

MFG225930

Understanding and Improving Your Results in Fusion 360 Simulation

Mike Fiedler
Autodesk, Inc.

Learning Objectives

- Learn the tools that Fusion 360 simulation provides to interrogate your simulation results.
- Learn some ways to increase your faith in your simulation results.
- Discover some of the common results-related issues.
- Learn how to correct some of the common results-based issues.

Description

In this course, we will focus on the results of the simulations. How do you get the most out of the results of your analysis? What are all the options that Fusion 360 simulation provides to review the results of the model? How do I know that the results can be trusted? Is it possible to improve the results that I am given?

Speaker(s)

As a designated support specialist at Autodesk, Inc., I help to provide proactive and reactive support in the area of simulation to Autodesk's Enterprise users. I obtained my bachelor's degree in mechanical engineering and have worked with locomotives, steam turbines, and sheet metal hydroforming prior to getting involved with finite element analysis (FEA). I have been helping FEA software users via technical support, training, and web content since 1999, and have been with Autodesk since 2009.

Learning Objective 1: Learn the tools that Fusion 360 simulation provides to interrogate your results.

Before we get to the specific tools in Fusion 360 simulation that help us evaluate our results and learn about our analyses, it might be beneficial to understand how this particular topic for a course came to be. First, as you can see from my biography above, I have been involved with finite element analysis for the past 19 years of my career. The topic of finite element analysis was then a given. Next, which of the products to use was the next question to be answered. Currently, there are offerings of Inventor simulation, Nastran In-CAD, and Fusion 360 from Autodesk. While I use all three at different times, and sometimes to validate against one another, I like to promote the Fusion product when I can. I find the program easy to use, yet has a good range of simulation capability. In addition to other attributes of the software, having access to the various workspaces (CAM, MODEL, RENDER and so on) all in one environment, I believe, could be very useful for a number of workplace environments. In short, I like the product. As this is a current product offering and is one that is relatively frequently updated, it would be good to note the version at the time I am writing this is 2.0.4801. It is possible that some appearances change, menus are restructured, or functionality is added or removed in future versions of the interface.

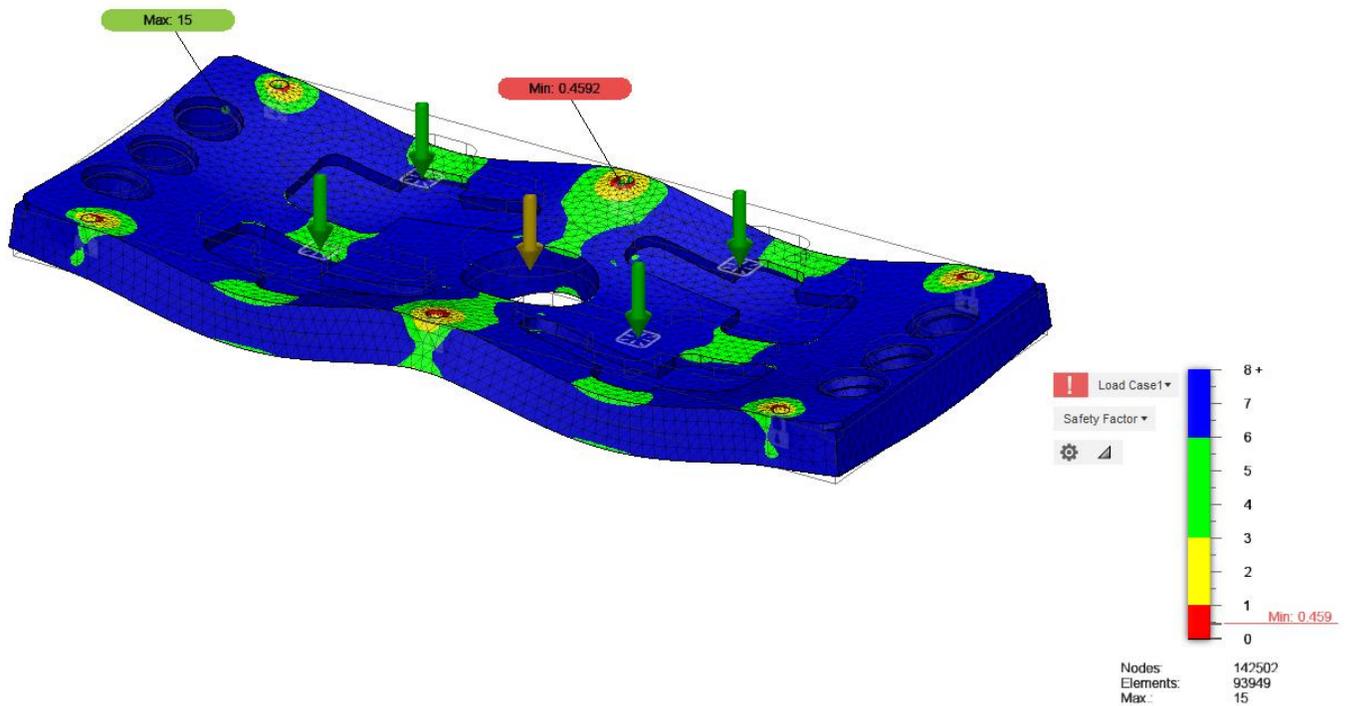
As someone who is reviewing the content, you are not necessarily here to be sold on a program, but rather because of the topic. Having settled on the particular field of study and program, we get down to the more granular level of topic. It has been my experience in product support that we have a lot to learn from the results, of course, why else would you run the simulation? If the simulation has been set up and run successfully, it would be wise to know the tools that the program provides for us to review the results and extract as much information as possible. If the results of our analysis would appear suspect, then there are also tools in the results environment that can potentially give us some insight in to what might have thrown our anticipated results off. We will look at the results tools from both perspectives; what is there to help us review a good analysis and what tools are there that might help us troubleshoot.

First stop – the display of the model and the legend.

Once the analysis has completed successfully, the Results View will display. The most obvious part of results of the analysis at this point are likely the graphical display of the model itself and the Legend. The default results that are displayed on the geometry and the options that you have available to switch between in the legend are a function of the type of analysis. We will cover now what the various analysis types provide from the viewpoint of display and the legend options.

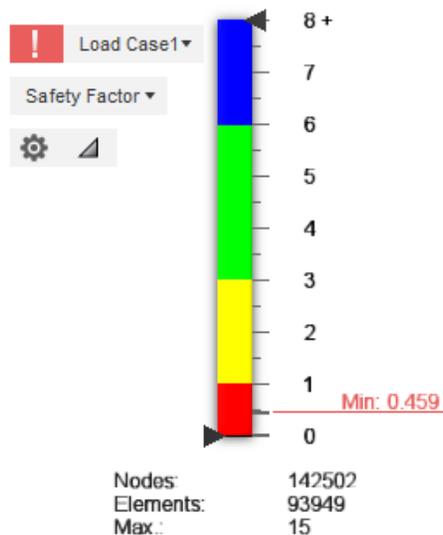
Static Stress

As can be seen in the image below, the default display of the results of a static stress simulation in Fusion 360 are the results of Safety Factor. The safety factor is the default display so that the user is able to make some quick determination about whether the geometry will be able to withstand the applied loads. Tags will automatically show the locations of the minimum and maximum results.



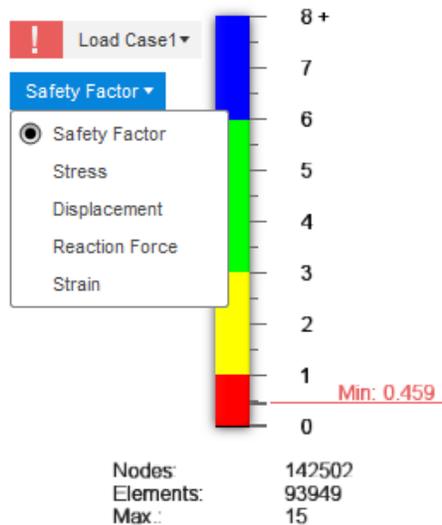
Default Display of Analysis Results in Static Stress – Showing Safety Factor and Min and Max Tags

We can access other results of the Static Stress analysis from the legend. If we take a look at the image of the legend below, we can see that there are drop down indicators next to both the option for Load Case and Safety factor, indicating that we can access additional options.



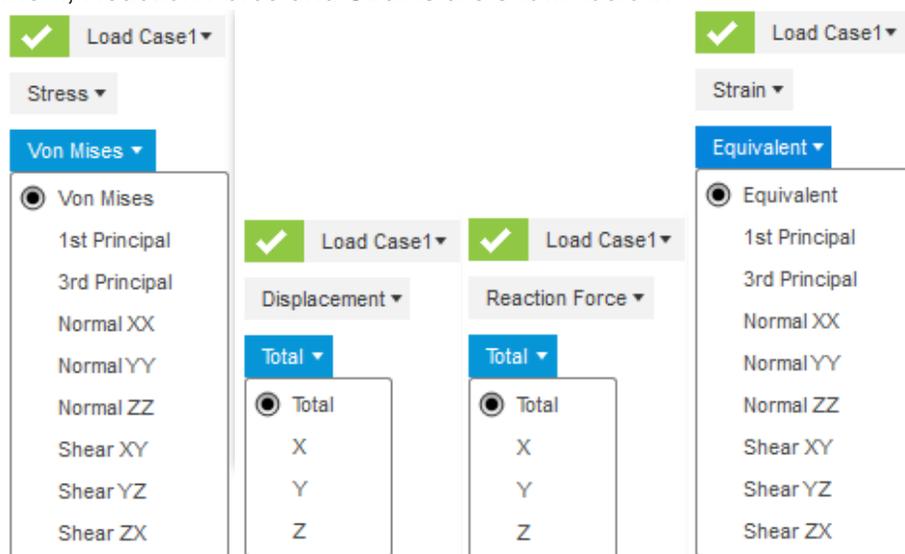
Detail of the Legend – Note Down Arrows Next to Load Case and Safety Factor

Presuming that we had set up multiple load cases prior to running the analysis, accessing the pull-down menu next to load case allows us to switch through the load cases. The pull-down menu for Safety Factor is how we can access different output results types. For static stress, these options include Safety Factor, Stress, Displacement, Reaction Force and Strain as shown in the image below.



Detail of the Legend – Options for Results Display in Static Stress

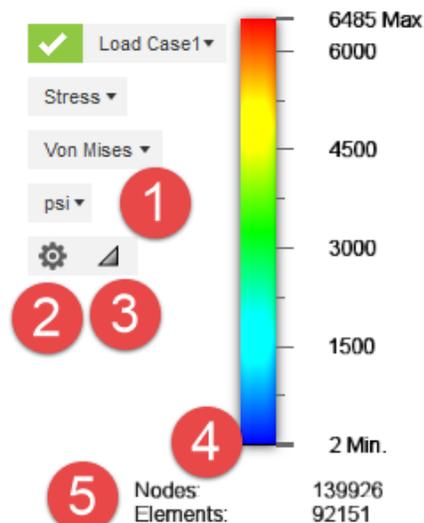
When we choose to display any of these other results options, then we will have additional options that we can select as sub-types. For instance, if we choose to display the option of Stress on the model, we will then obtain a submenu that allows us to choose Von Mises, Principal, and stress tensors. The various options for Stress display, Displacement, Reaction Force and Strains are shown below.



Detail of the Legend – Options for Results Display in Static Stress

Referencing the image above, it is possible to see that Autodesk Fusion 360 has a good range of results that can be explored. I don't doubt that people find the option to change the result type from Safety Factor to another option, such as Stress or Displacement, but I'm not sure how many people then also notice that opens the additional menus, such that one can change from viewing the default Von Mises stress, to – say, the Normal XX tensor for instance. It is good to know that these options exist. Detailing all of the specific results would make for a lengthy discussion that I will say is outside the scope of what we intend to introduce here. If there is interest in exploring more information about the various outputs, including stress tensors and the Von Mises stress, the Autodesk Fusion 360 documentation does a good job of providing additional details. You can find that documentation by accessing the topic Learning and Help from the upper right hand corner of the program, or, you can follow this link [Structural Results Help](#).

To round out the discussion of the legend options, at least for static stress, please reference the next image below.



Additional Static Stress Legend Items for Awareness

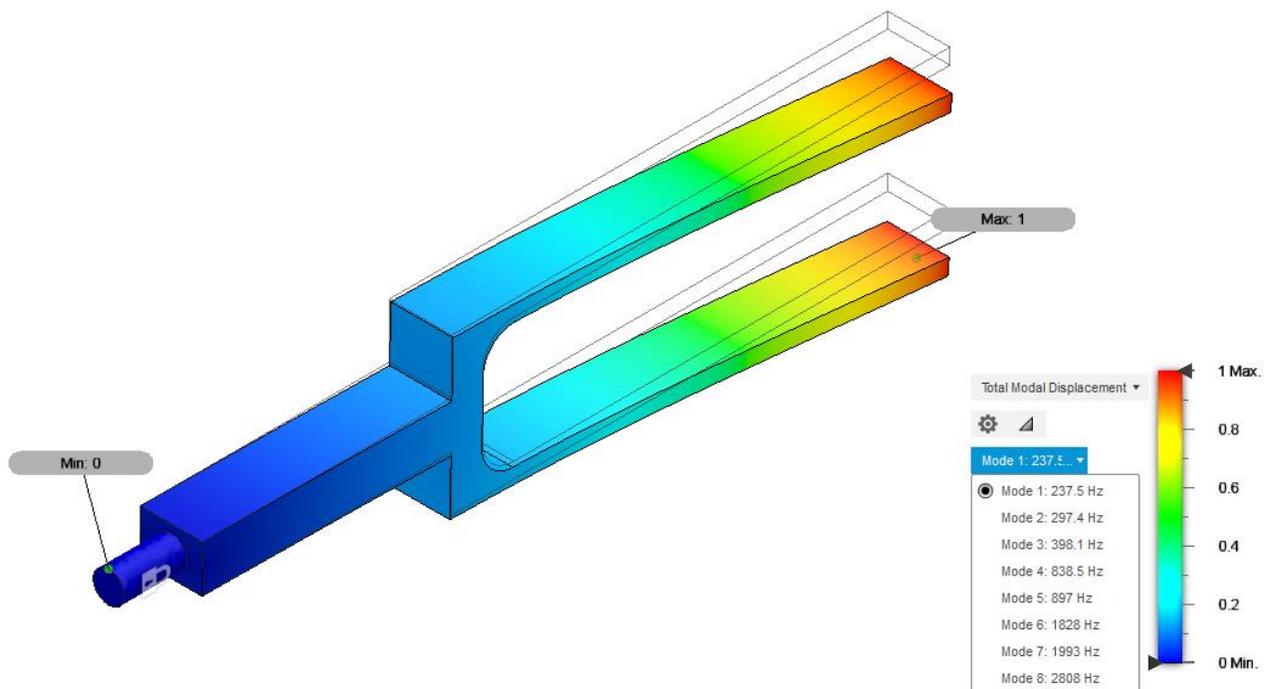
Of note with these numbered items:

1. When looking at certain results, such as stress, this drop-down menu allows you to change units “on the fly” between English and metric. This menu is not displayed by default as the units are not applicable to the Safety Factor result.
2. The gear looking icon is to access some additional legend options. There is a toggle for “Visible Only” which will make the range in the legend box dependent on what parts you have visibility turned on or off. There is an option that reads “Legend Size” that allows you to change the size of the legend between settings of Small, Normal and Large. Finally, the option “Color Transition” allows you to adjust the gradient between settings of Smooth and Banded.

3. The triangle icon that would be the lower right-hand corner of a square allows you to hide and show the legend.
4. Hovering over the top or bottom of the legend, you can grab on to the top or bottom of the scale and lower the range or move the bottom up, respectively. Doing so will hide elements with values greater than or less than the values beyond where you have moved the range. This is useful to allow you to concentrate on specific minimum or maximums in the model.
5. Finally, looking at the area of the legend, note that the program does report the quantity of nodes and elements in the model. As you utilize the program, you will gain a feel for how long models of a certain size will take and will be able to gauge the potential increase in solution time if you decide to make a mesh size change. The display of the count toggles on and off with the display of the mesh.

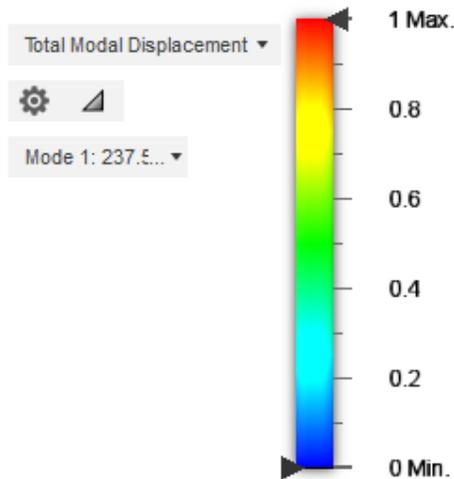
Modal Frequencies

As can be seen in the image below, the default display of the results of a modal frequencies simulation in Fusion 360 are the results of Total Modal Displacement. The total modal displacement does not give an indication of how much movement or displacement the object will experience at that frequency, but rather informs you where maximum and minimums occur. Tags will automatically show the locations of the minimum and maximum results.



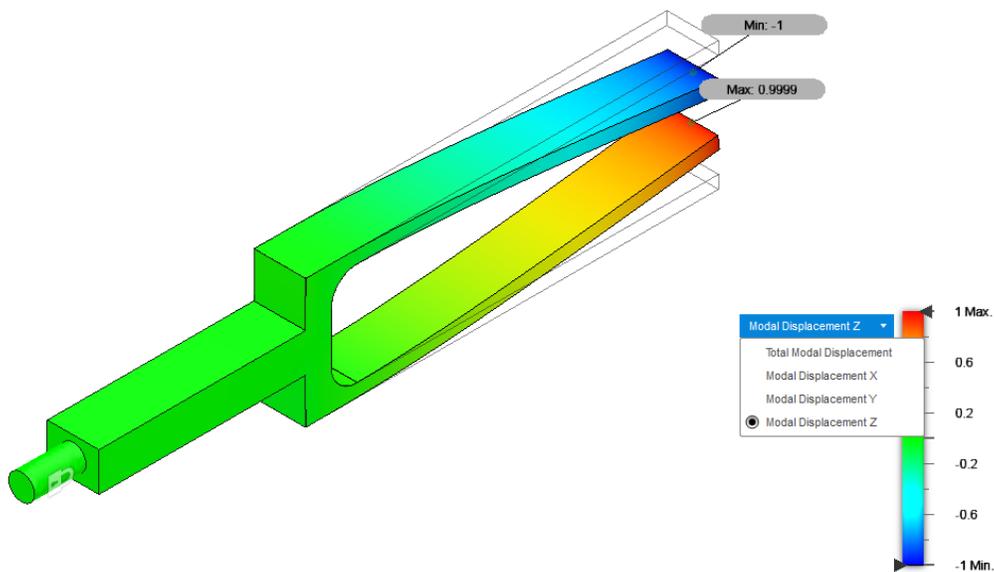
Default Display of Analysis Results in Modal Frequencies – Showing Total Modal Displacement

We can access a few other results of the Modal Frequencies analysis from the legend. If we look at the image of the legend below, we can see that there are drop down indicators next to both the option for Total Modal Displacement and next to Mode 1, indicating that we can access additional options.



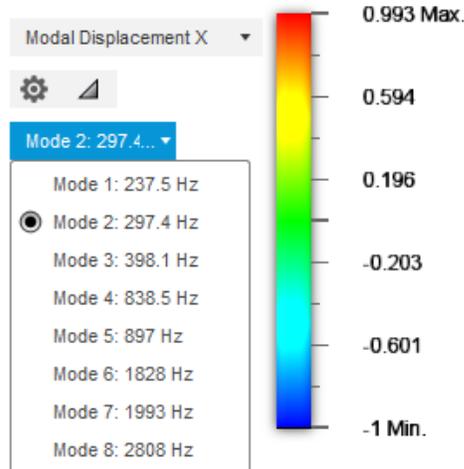
Detail of the Legend – Note Down Arrows Next to Total Modal Displacement and Mode 1

In addition to being able to look at the total modal displacement, from the drop-down menu it is possible to look at the components of the displacement in the x, y and or z direction. While the total modal displacement will always be positive, as a square root sum of squares type of calculation, the directional components can show positive or negative values depending on the direction of displacement as seen below.



Detail of the Legend – Options for Results Display in Modal Frequencies Showing Modal Displacement Z Lower Tine Shows Positive Value Upper Tine Showing Negative, Along Global Z Axis

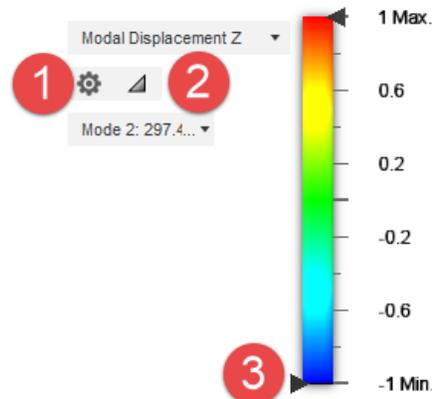
The other pull-down result option that you have here will allow you to see what the various frequencies are as a function of the analysis you have set up. In the below image, you can see that this tuning fork produced 8 modes with frequencies ranging from about 200 Hz to 3000 Hz for the first 8 natural frequencies. In addition to being able to observe the numeric value of the frequency, the model will also visibly display what that shape looks like.



Detail of the Legend – Display of Modes and Frequencies

I will offer two tips here regarding the results in the modal analysis. The first one is that you can control how many modes will be calculated – this can be done via the Manage pull down menu and then accessing Settings. On the General settings window, you will find the number of modes to be calculated input field. The second tip I would offer with respect to results is that it is generally helpful in Modal Frequencies analysis to see what that vibratory mode looks like. While we have not explored in to the pull-down menus beyond the legend area, yet, you can access the option to Animate from the Results menu in the ribbon. Again, this course is concentrating primarily on the results of your simulation, so I will not go much further in depth on the theory or setup of the modal analysis, however, if you would like to review the program’s documentation on the analysis type, you can follow this link [Modal Frequencies Analysis Help](#).

To round out the discussion of the legend options, at least for modal analysis, please reference the next image below.



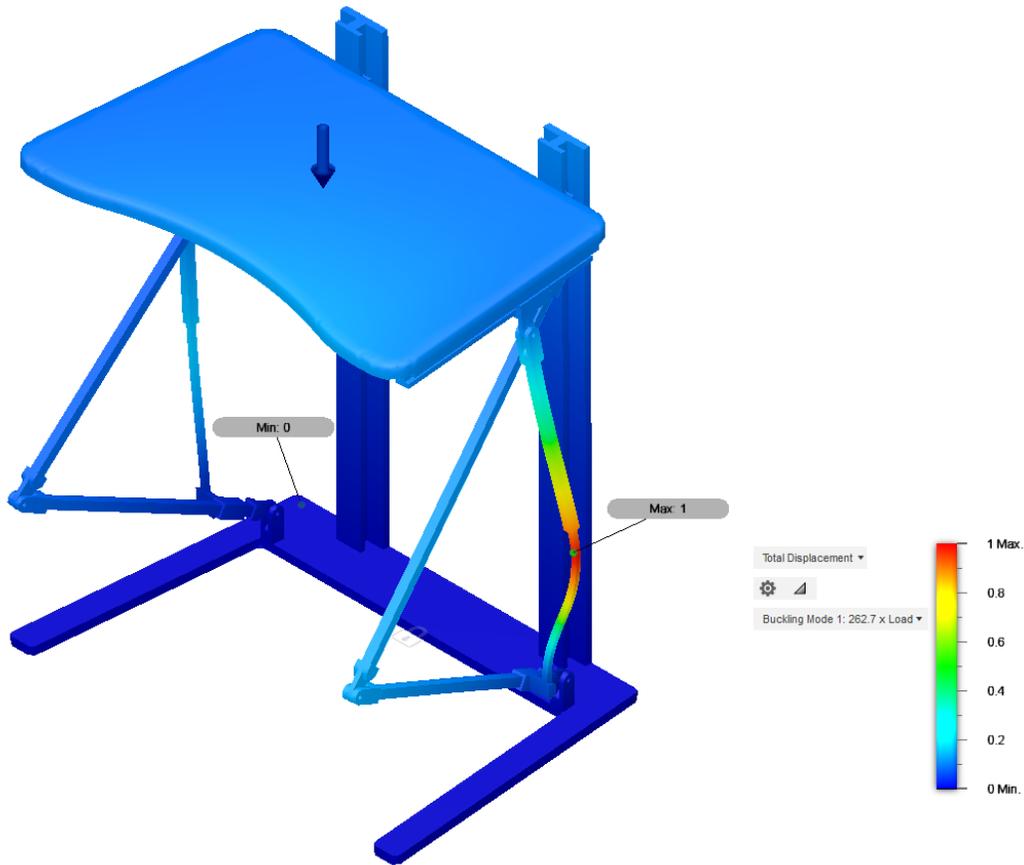
Additional Modal Analysis Legend Items for Awareness

Of note with these numbered items:

1. The gear looking icon is to access some additional legend options. There is a toggle for “Visible Only” which will make the range in the legend box dependent on what parts you have visibility turned on or off. There is an option that reads “Legend Size” that allows you to change the size of the legend between settings of Small, Normal and Large. Finally, the option “Color Transition” allows you to adjust the gradient between settings of Smooth and Banded.
2. The triangle icon that would be the lower right-hand corner of a square allows you to hide and show the legend.
3. Hovering over the top or bottom of the legend, you can grab on to the top or bottom of the scale and lower the range or move the bottom up, respectively. Doing so will hide elements with values greater than or less than the values beyond where you have moved the range. This is useful to allow you to concentrate on specific minimum or maximums in the model.

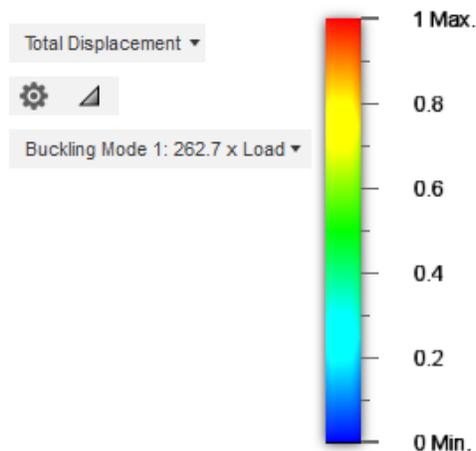
Structural Buckling

As can be seen in the image below, the default display of the results of a structural buckling simulation in Fusion 360 are the results of Total Displacement. The total displacement does not give an indication of how much movement or displacement the object will experience at buckling, but rather provides a general shape of the buckling and informs you where maximum and minimums occur. Tags will automatically show the locations of the minimum and maximum results.



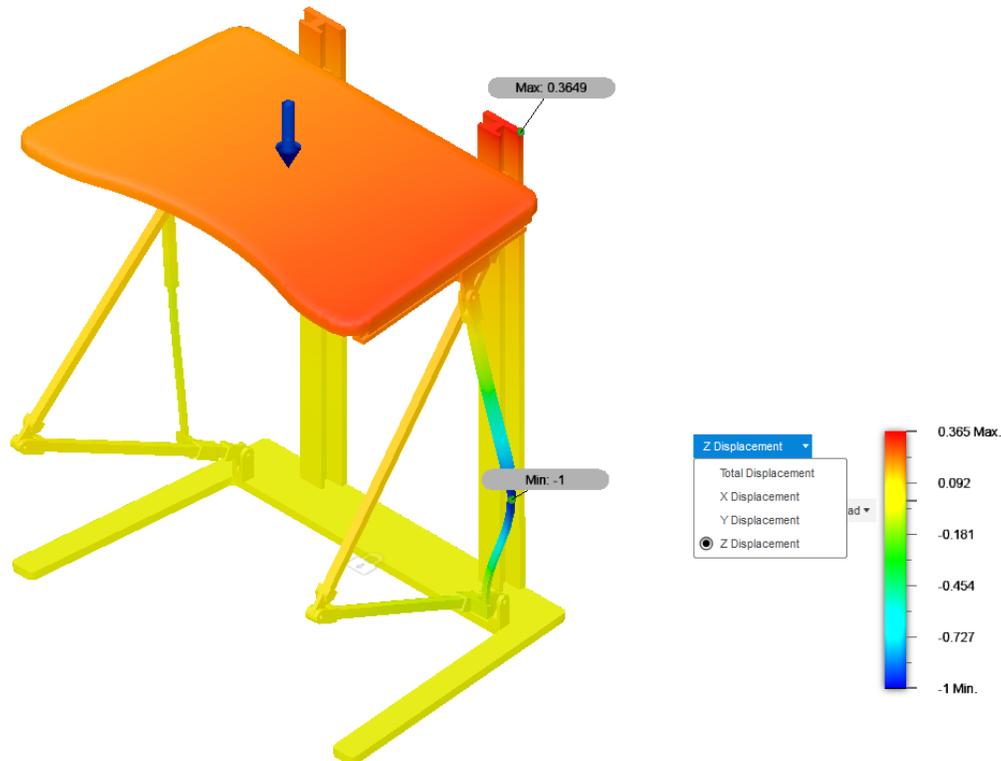
Default Display of Analysis Results in Structural Buckling – Showing Total Modal Displacement

We can access a few other results of the Structural Buckling analysis from the legend. Looking at the image below, there are drop down indicators next to the option for Total Displacement and Buckling Mode 1, indicating that we can access additional options.



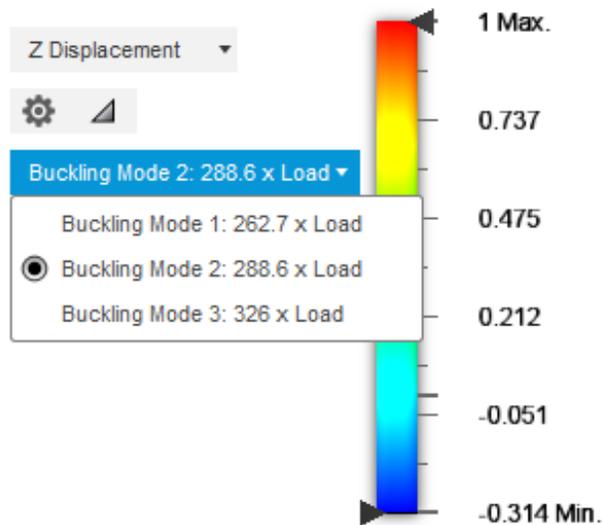
Detail of the Legend – Note Down Arrows Next to Total Displacement and Buckling Mode 1

In addition to being able to look at the total displacement, from the drop-down menu it is possible to look at the components of the displacement in the x, y and or z direction. While the total displacement will always be positive, as a square root sum of squares type of calculation, the directional components can show positive or negative values depending on the direction of displacement as seen below.



Detail of the Legend – Options for Results Display in Structural Buckling Showing Z Displacement

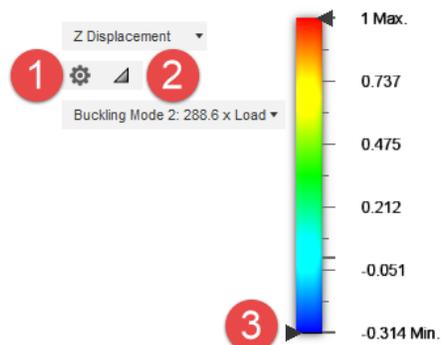
The other pull-down result option that you have here will allow you to see what the various buckling modes and factors are, as a function of the analysis you have set up. In the below image, you can see that this table produced 3 modes with load factors ranging from about 289 to 326 for the first 3 modes. In addition to being able to observe the numeric value of load factor, the model will also visibly display what that shape looks like. The load factor, when multiplied by the applied load on the model, gives an indication of what load would need to be applied to cause the structure to buckle. If you obtain a load factor less than 1, that would indicate that the currently applied load is more than sufficient to cause the structure to buckle.



Detail of the Legend – Display of Modes and Load Factors

I will offer a tip here regarding the results in the structural buckling analysis. That is that you can control how many modes will be calculated – this can be done via the Manage pull down menu and then accessing Settings. On the General settings window, you will find the number of modes to be calculated input field. As this course is concentrating primarily on the results of your simulation, I will not go much further in depth on the theory or setup of the structural buckling analysis, however, if you would like to review the program’s documentation on the analysis type, you can follow this link [Structural Buckling Analysis Help](#).

To round out the discussion of the legend options, at least for buckling analysis, please reference the next image below.



Additional Structural Buckling Analysis Legend Items for Awareness

Of note with these numbered items:

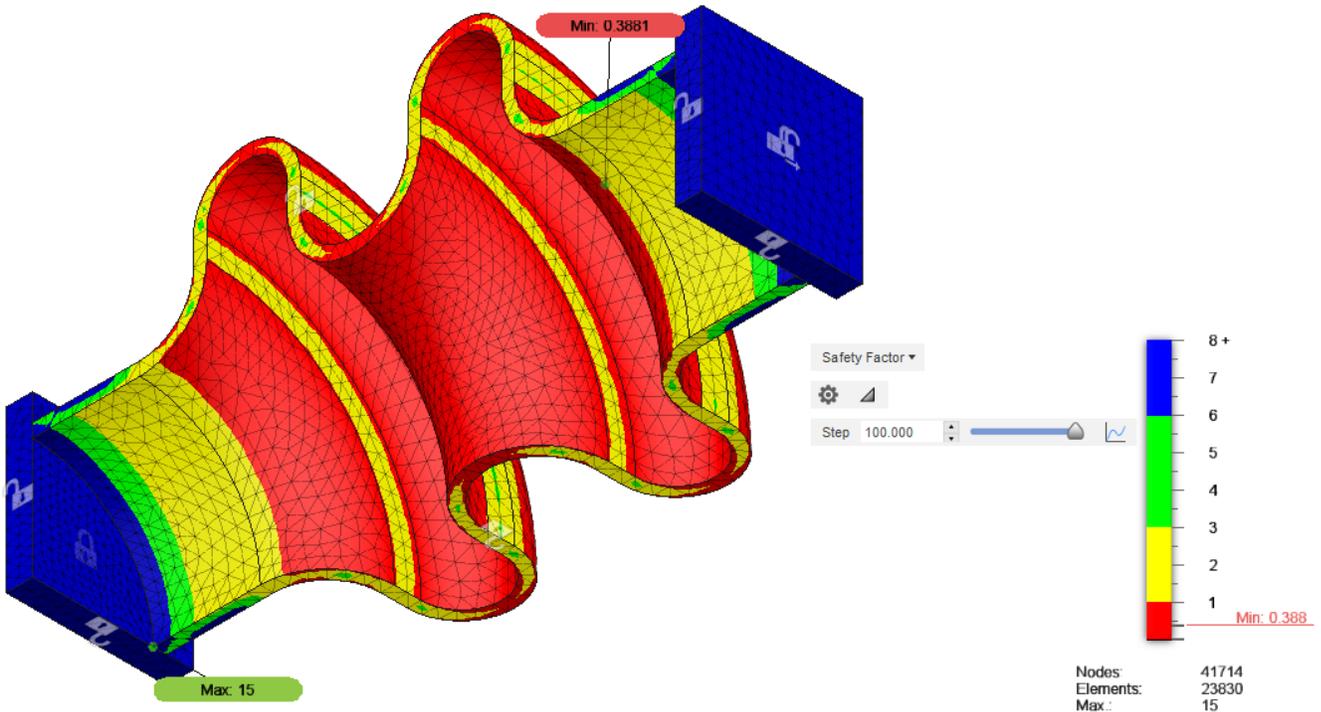
1. The gear looking icon is to access some additional legend options. There is a toggle for “Visible Only” which will make the range in the legend box

dependent on what parts you have visibility turned on or off. There is an option that reads “Legend Size” that allows you to change the size of the legend between settings of Small, Normal and Large. Finally, the option “Color Transition” allows you to adjust the gradient between settings of Smooth and Banded.

2. The triangle icon that would be the lower right-hand corner of a square allows you to hide and show the legend.
3. Hovering over the top or bottom of the legend, you can grab on to the top or bottom of the scale and lower the range or move the bottom up, respectively. Doing so will hide elements with values greater than or less than the values beyond where you have moved the range. This is useful to allow you to concentrate on specific minimum or maximums in the model.

Nonlinear Static Stress

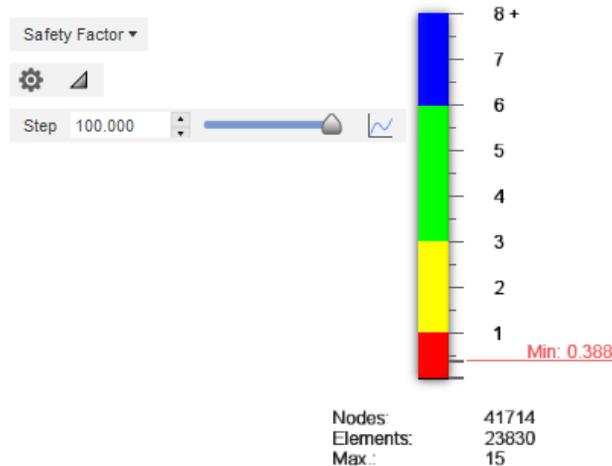
As can be seen in the image below, the default display of the results of a nonlinear static stress simulation in Fusion 360 are the results of Safety Factor. The safety factor is the default display so that the user can make some quick determination about whether the geometry will be able to withstand the applied loads. Tags will automatically show the locations of the minimum and maximum results.



Default Display of Results in Nonlinear Static Stress – Showing Safety Factor and Min and Max Tags

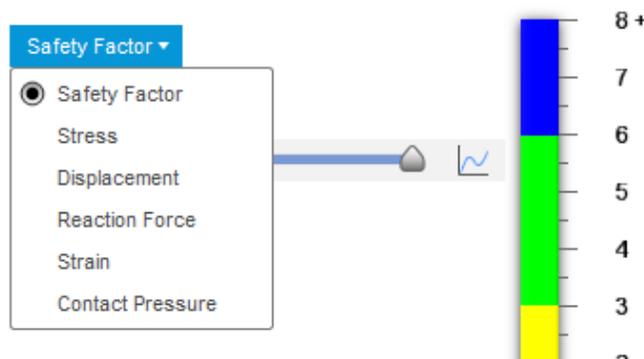
We can access other results of the Nonlinear Static Stress analysis from the legend. If we look at the image of the legend below, we can see that there is a drop-down indicator next to the option for Safety factor, and we also have a utility that we have not seen up

to this point yet, which shows Step (number) along with a slider and graph like icon (chart).



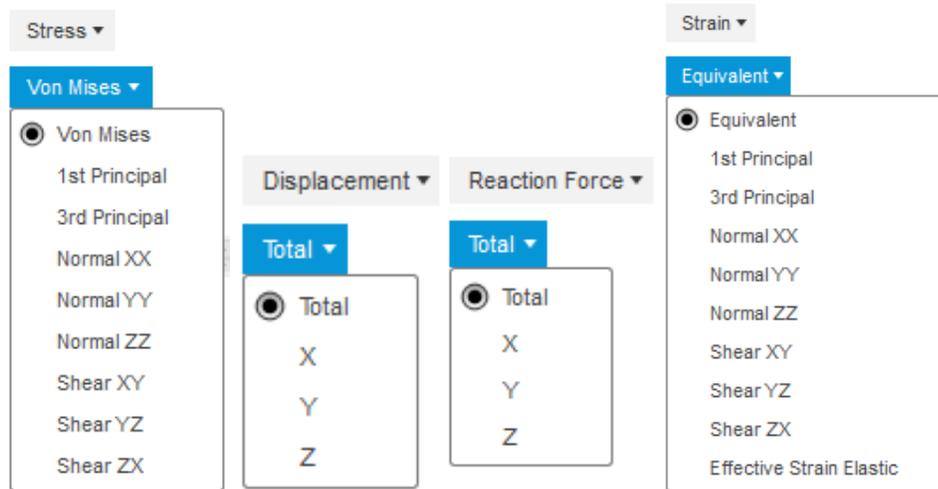
Detail of the Legend – Note Down Arrow Next to Safety Factor and Step

The pull-down menu for Safety Factor is how we can access different output results types. For nonlinear static stress, these options include Safety Factor, Stress, Displacement, Reaction Force, Strain, and Contact Pressure as shown in the image below.



Detail of the Legend – Options for Results Display in Nonlinear Static Stress

When we choose to display most of these other results options, then we will have additional options that we can select as sub-types. For instance, if we choose to display the option of Stress on the model, we will then obtain a submenu that allows us to choose Von Mises, Principal, and stress tensors. The various options for Stress display, Displacement, Reaction Force and Strains are shown below. Contact Pressure will just display that result on the geometry, it does contain a submenu.



Detail of the Legend – Options for Results Display in Nonlinear Static Stress

Referencing the image above, it is possible to see that Autodesk Fusion 360 has a good range of results that can be explored. It is good to know that these options exist. Detailing all these specific results would make for a lengthy discussion that I will say is outside the scope of what we intend to introduce here. If there is interest in exploring more information about the various outputs, including stress tensors and the Von Mises stress, the Autodesk Fusion 360 documentation does a good job of providing additional details. You can find that documentation by accessing the topic Learning and Help from the upper right hand corner of the program, or, you can follow this link [Structural Results Help](#).

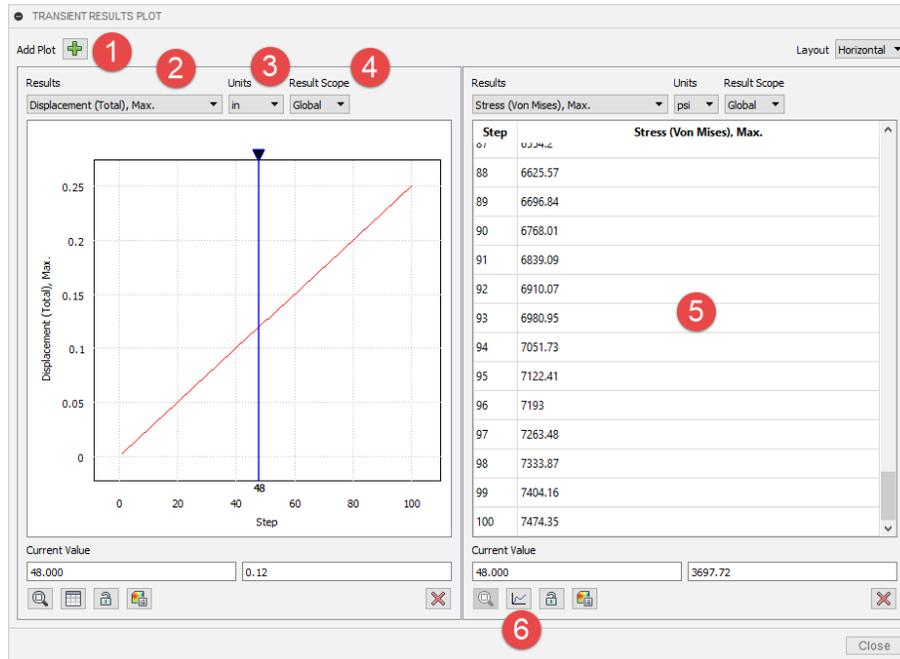
Legend items that we have not yet discussed in this document include the portion of the legend where the step, slider and graph icon (chart) are shown, as seen in the image below.



Detail of the Legend – Step, Slider and Chart

The nonlinear static stress analysis breaks the solution down in to several solution steps. The number of steps used in the solution are defined under Manage; Settings; General; Number of Steps when you are setting up the analysis. For this analysis, you can see that the I defined 100 steps for the solution. Using the up and down arrows, you can advance forwards or backwards through the results. You can also click in the field and type the specific step number you would like to review. The geometry will adjust to show the appropriate deformed geometry and output results for the given step and the values in the legend will be adjusted to the appropriate range for the given solution step. This slider is a quick way to adjust the step number, place your cursor somewhere along the line and left button click, or left click to grab on to the upward facing triangular handle and move it along the line to the step number you want to review.

Moving to the right along this utility, the last option here, which displays a graph like icon is for opening a 2D chart. The chart has several interesting and useful capabilities that I will highlight a few of in the image below.



Nonlinear Static Stress 2D Chart

When you first open the chart, you will be presented with a singular graph, such as the displacement plot shown on the left in the image above. Additional items;

1. If you press the Add Plot button you add additional plots, such as the item shown on the right, such that you can compare, say, displacements over the steps vs. how the stress changes over the solution steps.
2. The Results pull down menu allows you to change what is plotted, including stress, strain, reactions, displacement and safety factor.
3. The Units allow you to quickly toggle between metric and English.
4. Via Results Scope, you can choose to have the plotted result based on the Global results (all nodes of the model considered), or, change this to Probe and add a probe to the model for results at a specific location.
5. While we have a graph on the left, notice there is a table on the right (Step and Stress (global max von Mises), which I added via Add Plot.
6. Icon shown at the bottom of the chart or graph allows you to toggle the display of the information between table and graph display.



Additional Nonlinear Static Stress Legend Items for Awareness

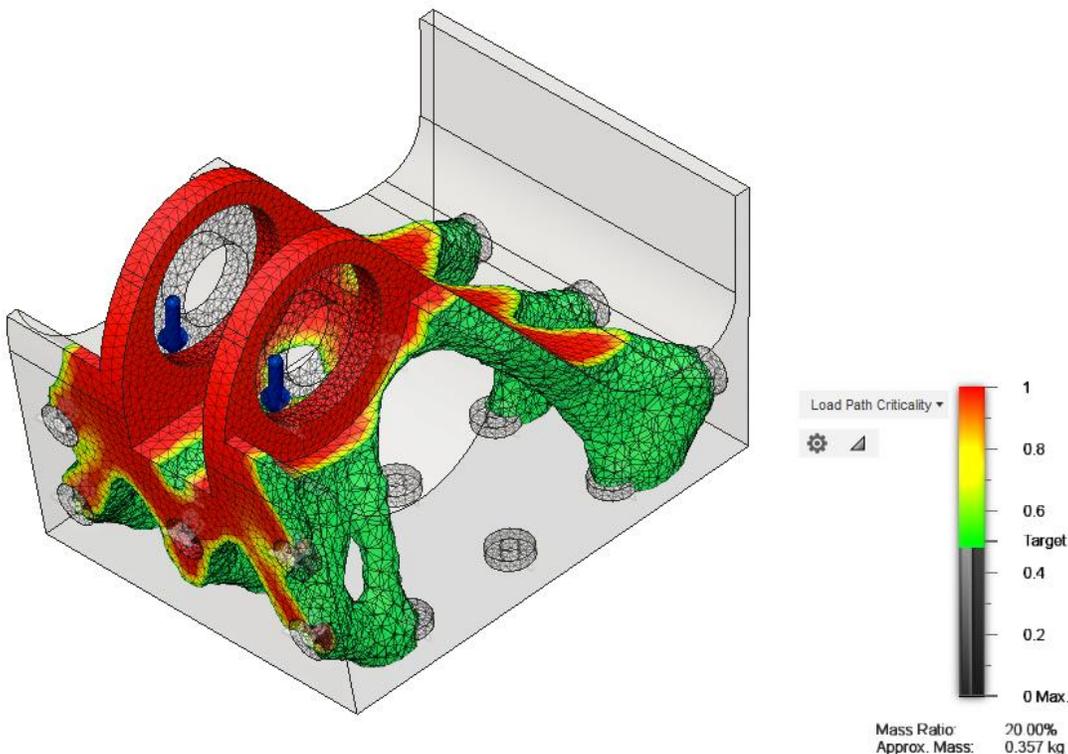
Of note with these numbered items:

1. When looking at certain results, such as stress, this drop-down menu allows you to change units “on the fly” between English and metric. This menu is not displayed by default as the units are not applicable to the Safety Factor result.
2. The gear looking icon is to access some additional legend options. There is a toggle for “Visible Only” which will make the range in the legend box dependent on what parts you have visibility turned on or off. The checkbox for “Update Range Per Step” controls whether the range of values on the legend bar itself update as the steps are changed or not. There is an option that reads “Legend Size” that allows you to change the size of the legend between settings of Small, Normal and Large. Finally, the option “Color Transition” allows you to adjust the gradient between settings of Smooth and Banded.
3. The triangle icon that would be the lower right-hand corner of a square allows you to hide and show the legend.
4. Hovering over the top or bottom of the legend, you can grab on to the top or bottom of the scale and lower the range or move the bottom up, respectively. Doing so will hide elements with values greater than or less than the values beyond where you have moved the range. This is useful to allow you to concentrate on specific minimum or maximums in the model.
5. Finally, looking at the area of the legend, note that the program does report the quantity of nodes and elements in the model. As you utilize the program, you will gain a feel for how long models of a certain size will take and will be able to gauge the potential increase in solution time if you

decide to make a mesh size change. The display of the count toggles on and off with the display of the mesh.

Shape Optimization

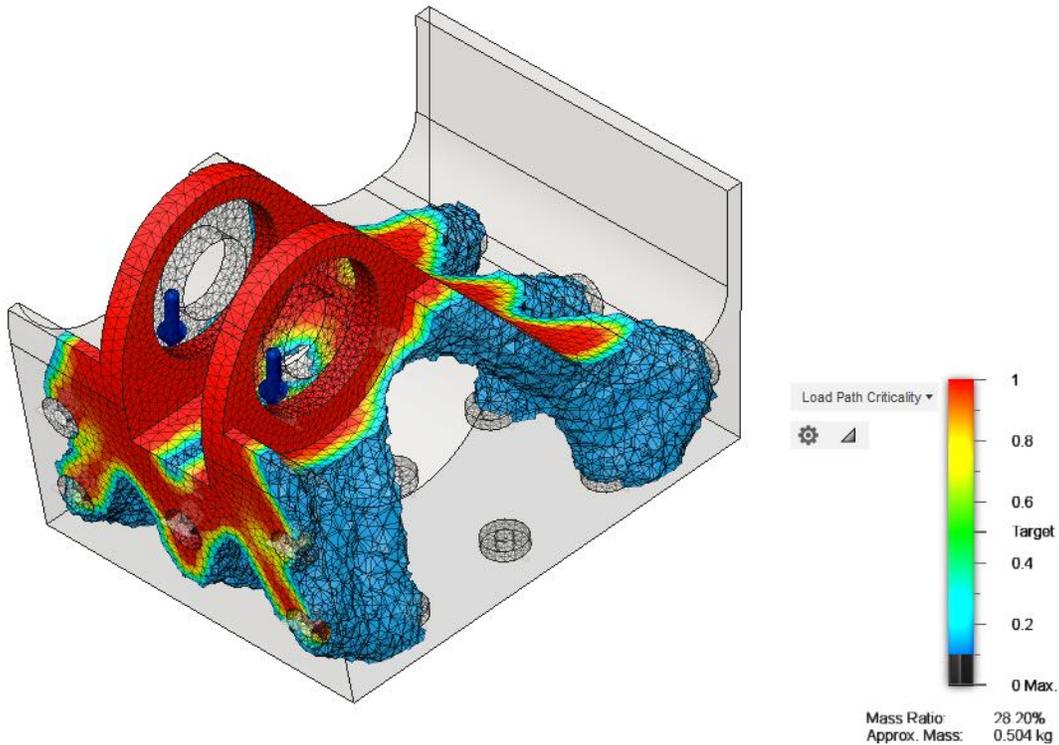
As can be seen in the image below, the default display of the results of a shape optimization simulation in Fusion 360 are the results of the optimized shape and display result of Load Path Criticality. The value will range from 0 to 1, where 1 represents the area of the model that is most critical to resist the applied loads and 0 the least. In setting up the analysis, one would have defined a target mass (e.g. reduce to 20% of the starting geometry's original mass), so the middle of the legend, where the lower slider is, should be that user defined goal, or Target as the legend identifies it.



Default Display of Analysis Results in Shape Optimization – Showing Load Path Criticality

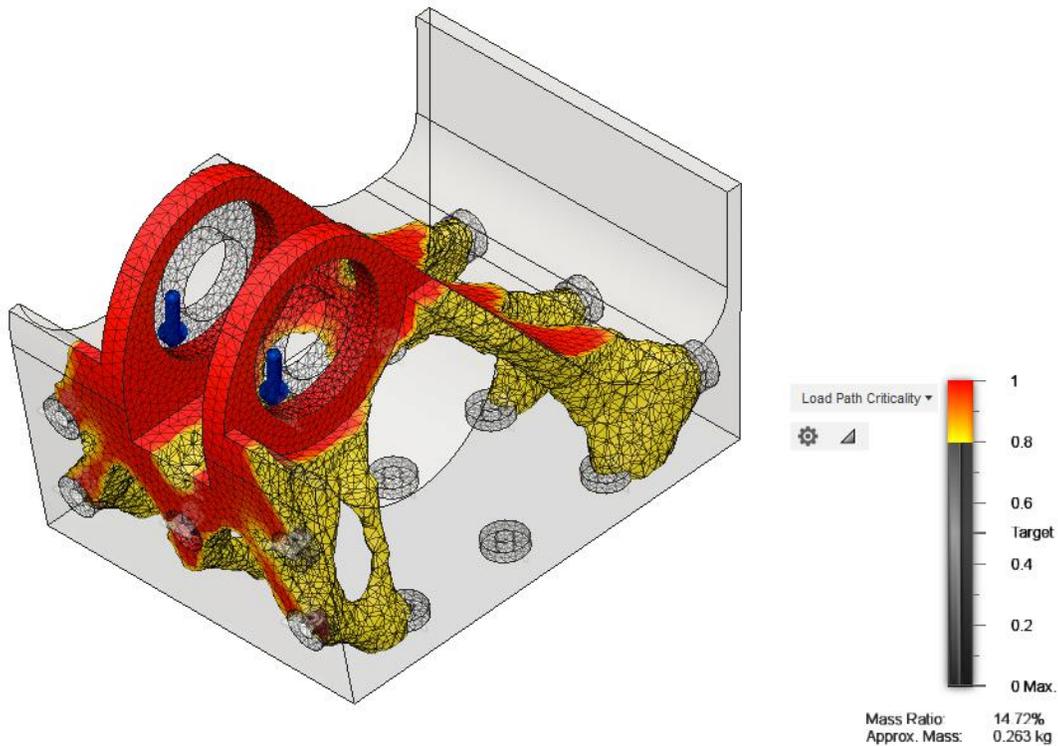
As can be seen from the image above, below the legend, the mass ratio and approximate mass is reported. For this analysis type, there are not too many options around the legend that would need to be explored. Likely, the most that one might do in the legend area in the results of a shape optimization study would be to adjust the range on the slider. If you move your cursor over the top of the vertical legend bar, you will notice that triangular arrow heads or handles will appear at the top and middle of the legend. We will first grab the lower handle and slide it further down the legend bar, moving it from the middle “Target” towards the lower end 0 value. If the original output of the analysis perhaps looked overly aggressive on the removal of material, moving the slider downward shows us material that was removed in the effort to get to the target

mass – material which was deemed less critical to the load path. You can see from the image below that where I have left the model now increases the mass ratio from 20% to approximately 28% and the approximate mass from .38 kg to approximately .5 kg.



Shape Optimization – Exploring Less Critical Regions

Alternatively, moving the grip further up beyond the Target, this will remove more material and we can see that the mass ratio in the below image is about 15% and approximate mass is .26 kg. As these regions are further up the scale on load path criticality, this would indicate that they are more critical pathways for carrying the applied load through the structure to the supports of the geometry. If you were going to machine away geometry to light weight a design, these are then likely areas of the original design that you would want to avoid removing material from.



Shape Optimization – Exploring More Critical Load Paths

I will offer a tip here that the size of the mesh that is utilized will have an impact on the results. For instance, imagine if you could use a very coarse mesh and mesh the original geometry using a total of 10 elements. In removing just one of those elements, we have reduced the mass by 10%, but that would also be a very coarse/obvious segment gone from the original geometry. On the other hand, if we mesh the geometry much more finely, the program can remove (or leave behind) elements in a more strategic fashion, resulting a more refined result. That said, additional elements also result in additional computation time. I would perhaps propose a mesh at the coarser end of the spectrum is run first to ensure that the end results make sense ... you have captured appropriate preserve regions for instance. Then, once you are satisfied, you can then explore several other iterations at smaller mesh sizes to see how this changes/refines the outcome. If you would like to review the program's documentation on the analysis type, you can follow this link [Shape Optimization](#).

To round out the discussion of the legend options for shape optimization analysis, please reference the next image below.



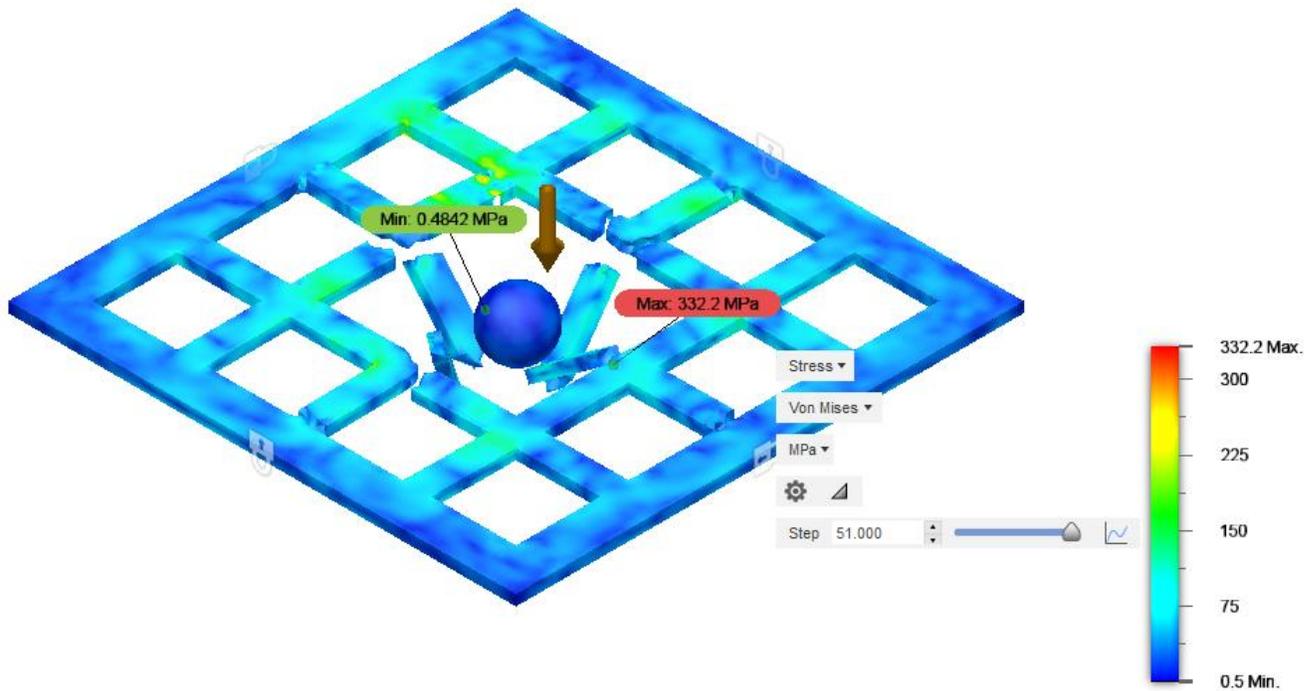
Additional Shape Optimization Analysis Legend Items for Awareness

Of note with these numbered items:

1. The gear looking icon is to access some additional legend options. There is a toggle for “Visible Only” which will make the range in the legend box dependent on what parts you have visibility turned on or off. There is an option that reads “Legend Size” that allows you to change the size of the legend between settings of Small, Normal and Large. Finally, the option “Color Transition” allows you to adjust the gradient between settings of Smooth and Banded.
2. The triangle icon that would be the lower right-hand corner of a square allows you to hide and show the legend.

Event Simulation (Technology Preview)

The Event Simulation analysis type, at the time of this writing is listed as a technology preview. That said, this is a powerful and robust analysis type that is accessible to the user, so we will detail the results that are accessible. Reviewing the image below, the default display of the results of an event simulation in Fusion 360 are the results of Stress (Von Mises). The stress as the default display allows the user to be able to make some quick determination about whether the geometry will be able to withstand the applied loads, has already exceeded the yield, or, give some initial impression about whether the event simulation was set up appropriately (that is, that the analysis results are within reason as to what one might have expected the outcome to be). Tags will automatically show the locations of the minimum and maximum results.



Default Display of Results in Event Simulation – Showing Von Mises Stress

We can access other results of the Event Simulation analysis from the legend. Referencing the numbered legend image below, I will cover (in shorter detail) the options that are accessible here in the legend area. These elements of the legend have been discussed and detailed in prior sections of the document, so I will not go in to too elaborate of detail here, but reference prior sections of this document where applicable.



Itemized Detail of the Legend – Event Simulation

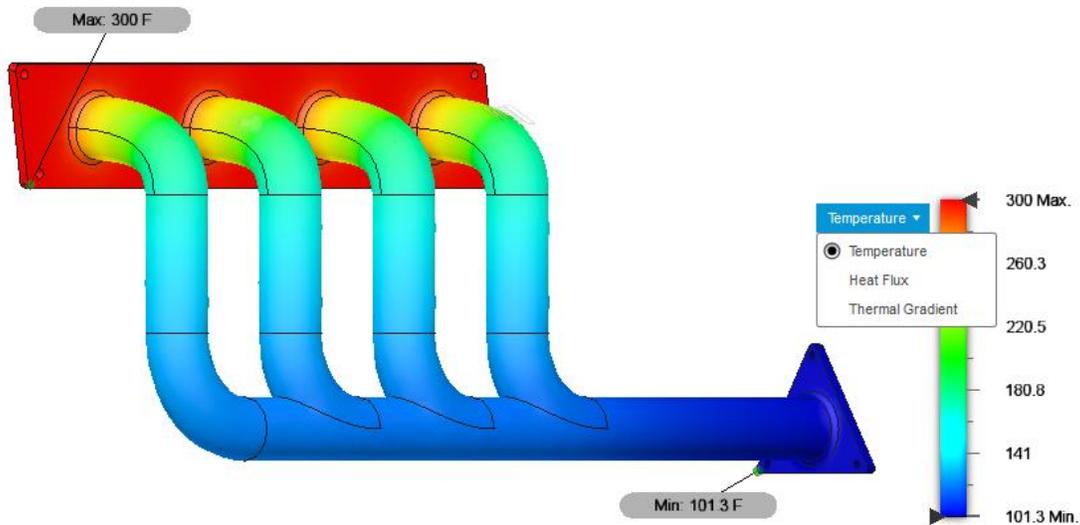
Referencing the image above, the legend elements for event simulation are;

1. The type of output display on the model and shows values of on the legend. Options here include Stress, Displacement, Strain, Acceleration, and Velocity.
2. Sub-menu for the output display of items listed in item one above.
 - When the display of Stress is chosen, options include Von Mises, Normal, Shear and Principal Stresses
 - When the display of Displacement is chosen, options include Total and X, Y, and Z vector displays.
 - When the display of Strain is chosen, options include Equivalent, Normal, Shear and Principal
 - When the display of Acceleration is chosen, options include Total and X, Y, and Z vector displays.
 - When the display of Velocity is chosen, options include Total and X, Y, and Z vector displays.
3. Option to change units on the fly between English and metric.
4. The gear looking icon includes the option for “Visible Only”, “Update Range Per Step”, “Legend Size” and “Color Transition”. The options for visible only, legend size and color transition have been consistent throughout the analysis types we have explored. The option for updating the range per step was outlined under the details for the nonlinear static analysis type.
5. This triangle icon will toggle between hide and show the legend.
6. This Step field and slider allow you to change what solution step of the analysis is displayed. This was detailed under the nonlinear static analysis section of the document.
7. This graph looking icon is for accessing the 2D chart, such that results can be graphed with respect to solution step. This too is covered extensively in the nonlinear static stress section of the document.
8. Hovering the mouse over the legend will expose the triangular handles at the top and bottom of the legend. You can grab on to the top or bottom of the scale and lower the range or move the bottom up, respectively. Doing so will hide elements with values greater than or less than the values beyond where you have moved the range. This is useful to allow you to concentrate on specific minimum or maximums in the model.
9. Node and element counts are displayed, so long as the mesh is displayed on the geometry.

If there is interest in exploring more information about the various outputs, or setup of the event simulation, I will direct you to the documentation. An entire course could be dedicated to the topic (like many of these analysis types), so I would encourage you, if interested to explore more on your own. I have, up to this point, been linking to the relevant portions of the User Guide. Please note that there are also many other resources to be found, such as videos and tutorials to help you. You can find the help documentation by accessing the topic Learning and Help from the upper right hand corner of the program, or, you can follow this link [Event Simulation Help](#) .

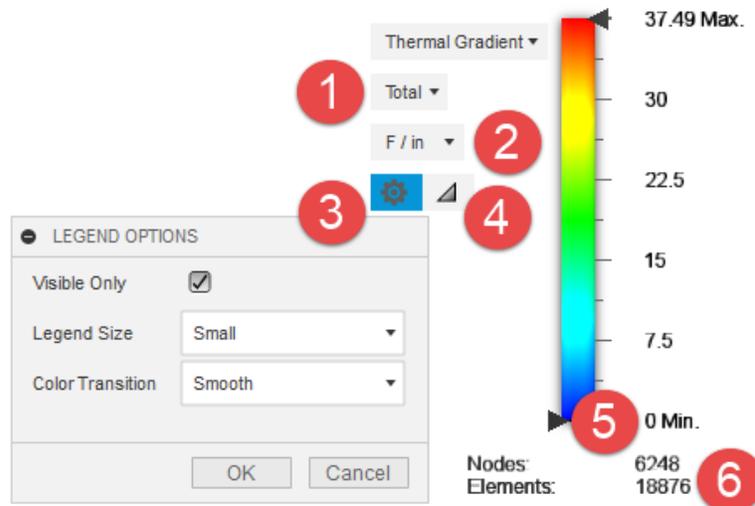
Thermal

Reviewing the image below, the default display of the results of a thermal simulation in Fusion 360 are the results of the Temperature and the temperature distribution is displayed on the model. From the pull-down menu that is shown, in addition to displaying temperature on the model, other results that can be displayed on the geometry include Heat Flux and Thermal Gradient.



Default Display of Analysis Results in Thermal – Showing Temperature

We will round out the discussion of the legend items for the thermal study type by referencing the image below and the respective sequence of numbered items.



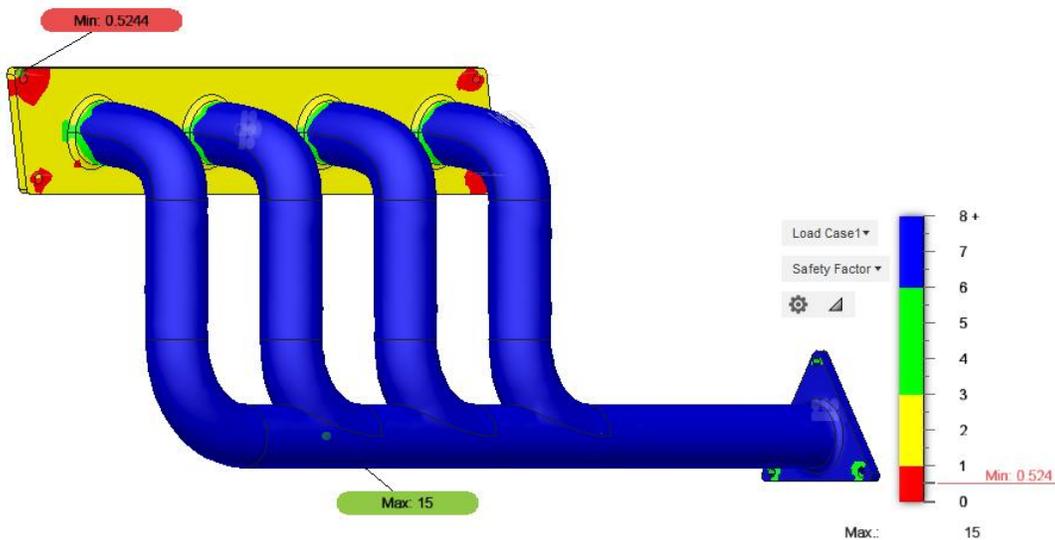
Thermal Analysis – Additional Items of the Legend

1. When looking at results of Heat Flux or Thermal Gradient, the default is to display the Total. From this pull-down menu, you can access the vector-based results of x, y or z directions specifically.
2. This drop-down menu allows you to change units “on the fly” between English and metric.
3. The gear looking icon is to access some additional legend options, as shown in the box menu. There is a toggle for “Visible Only” which will make the range in the legend box dependent on what parts you have visibility turned on or off for. There is an option that reads “Legend Size” that allows you to change the size of the legend between settings of Small, Normal and Large. Finally, the option “Color Transition” allows you to adjust the gradient between settings of Smooth and Banded.
4. The triangle icon that would be the lower right-hand corner of a square allows you to hide and show the legend.
5. Hovering over the top or bottom of the legend, you can grab on to the top or bottom of the scale and lower the range or move the bottom up, respectively. Doing so will hide elements with values greater than or less than the values beyond where you have moved the range. This is useful to allow you to concentrate on specific minimum or maximums in the model.
6. Finally, presuming that you have the mesh displayed, it is possible to see your node and element count.

If you are interested in learning more details about Thermal analysis, you can find the help documentation by accessing the topic Learning and Help from the upper right hand corner of the program, or, you can follow this link [Thermal Analysis Help](#).

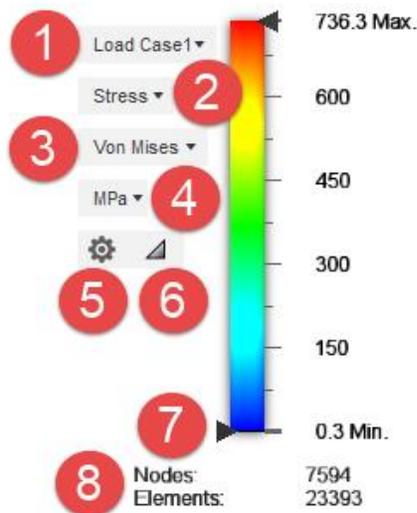
Thermal Stress

Our eighth (and currently final analysis type) to review what is available from a legend perspective is thermal stress analysis. The thermal stress analysis is essentially a combination of the analysis types of thermal and static stress, so, it would make sense the results that are available for study contains both thermal and structural centric results. Reviewing the image below, the default display of the results of a thermal stress simulation in Fusion 360 are the results of the Safety Factor, like what is seen when a stress analysis type has been run.



Default Display of Analysis Results in Thermal Stress – Showing Safety Factor

We will round out the discussion of the legend items for the thermal stress study type by referencing the image below and the respective sequence of numbered items.



Thermal Stress Analysis – Additional Items of the Legend

1. It is possible in this analysis type to put different types of loads or different magnitudes of loads in to different load cases and have them solved in a single analysis. To review the results of these different scenarios, different load cases, you would use the drop-down menu here to switch between load cases.
2. This drop-down menu allows you to change the type of results that you have displayed on the model. Options include Safety Factor, Stress,

Displacement, Reaction Force, Strain, Temperature, Heat Flux, and Thermal Gradient.

3. This menu allows you to choose the sub-type of result. Note that this is not displayed when the chosen result type does not require a sub-type, such as with safety factor and temperature.
 - a. Stress; Von Mises, principal, normal, and shear
 - b. Displacement; total or vector components (x, y, z)
 - c. Reaction force; total or vector components (x, y, z)
 - d. Strain; equivalent, principal, normal, and shear
 - e. Heat flux; total or vector components
 - f. Thermal gradient; total or vector components
4. When applicable, this will allow you to change units displayed between English and metric. Note that this is not displayed when units are not applicable, such as with safety factor.
5. The gear icon allows you to control three different things;
 - a. Visible only – when checked, the range in the legend box will be based only on the parts you have made visible.
 - b. Legend size – choose the scale of the legend from small, medium, or large.
 - c. Color transition – smooth or banded display of results on the geometry.
6. This triangle looking icon allow you to hide and show the legend.
7. Hovering over the legend, will cause these small triangle handles or grips to appear at the top and bottom. You can grab on to the top or bottom of the scale and lower the range or move the bottom up, respectively. Doing so will hide elements with values greater than or less than the values beyond where you have moved the range. This is useful to allow you to concentrate on specific minimum or maximums in the model.
8. Finally, presuming that you have the mesh displayed, it is possible to see your node and element count.

If you are interested in learning more details about thermal stress analysis, you can find the help documentation by accessing the topic Learning and Help from the upper right hand corner of the program, or, you can follow this link [Thermal Stress Analysis Help](#) .

First stop wrap up of the display of the model and the legend.

We have now reviewed in the pages above all the various results displayable for the eight different analysis types and how to access various results around the legend. As a reminder, our first learning objective was to “Learn the tools that Fusion 360 provides to interrogate your simulation results.” We have two additional locations that we can look to in the Results View to learn about our simulation results and those would be the ribbon, or Toolbar across the top and also the Browser tree on the left.

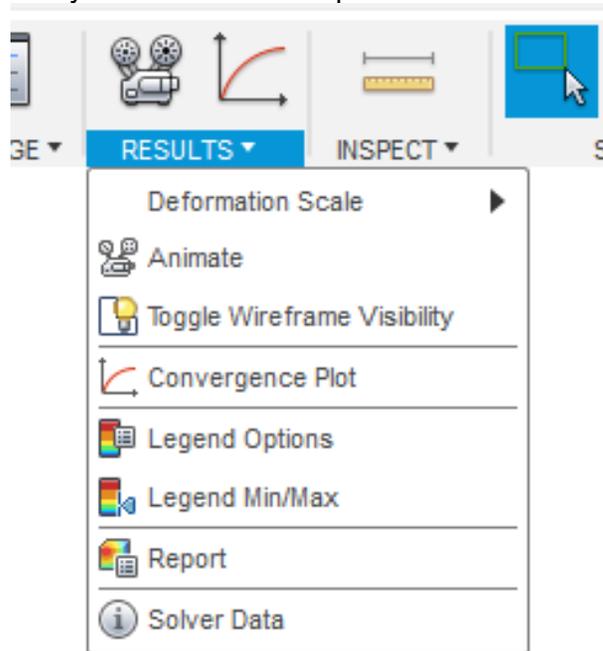
Second stop – results options from the Toolbar.

Across the top of the window is the toolbar and you have two primary locations here that are of interest when exploring the results on your analysis. Towards the right-hand end of the toolbar

you have the Results tool and the Inspect tool. Both can be helpful when reviewing the results, so we will take a look at the options that each of these provides for us.

Results tool from the toolbar

The list of post-processing options shown in the image below is almost entirely consistent across the eight analysis types that we have introduced, with very little exception. We'll briefly detail what these options do.



Display of Results Tool from the Toolbar – Captured from Static Stress Analysis

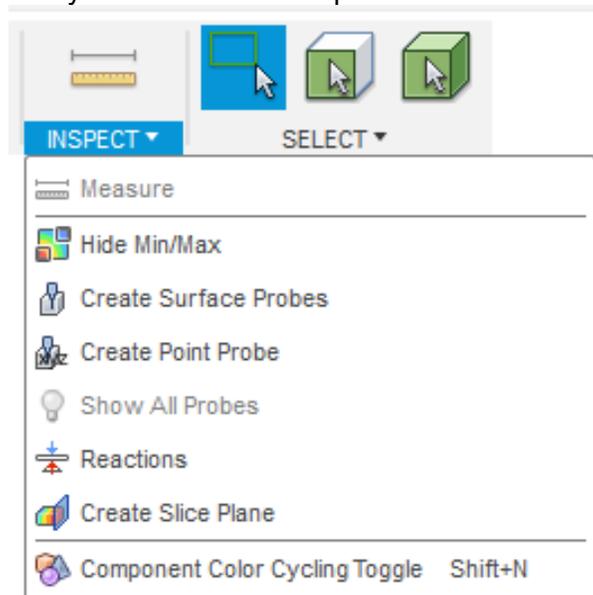
- **Deformation scale:** This allows you to adjust the deformed shape of the analyzed model. It can be set between undeformed, actual, adjusted 0.5x, adjusted, adjusted 2x and adjusted 5x. We will discuss some considerations for this in learning objective 2. This menu option is always visible, but will be not applicable to some analysis types, such as thermal analysis and shape optimization.
- **Animate:** This option allows to animate the deformed shape of the geometry from the original un-displaced shape to whatever deformation scale you have applied. Tip – if it does not appear to move at all during the animation, make sure you have the deformation scale set to something other than undeformed. Tip 2 – for generative design, it will animate the optimization process, or removal of material. Tip 3 – the animate dialogue has a record button so that you can share the animation via *.avi file with another party. Finally, the animate does not apply to and will not be visible in the results of a thermal analysis.
- **Toggle wireframe visibility:** Allows you to hide and show cad geometry wireframe. Can be hidden to make a cleaner results presentation. Displaying it

can help interpret how the geometry is deforming from the original un-displaced shape.

- **Convergence plot:** If you have run an adaptive mesh refinement study, this option will allow you to see how the stress or displacement changed over the various solution steps. Note that this result option will not be displayed in the menu if a mesh study was not run. Further, the adaptive mesh refinement study is not applicable to all analysis types.
- **Legend options:** This menu accesses the same legend options as the gear icon that is displayed near the legend itself and was covered in prior sections of this document. At a minimum, allows you to set the visible only option, control the size of the legend and the color transition.
- **Legend Min/Max:** This option allows you to set the minimum value in the legend and the maximum. This can be useful, for instance, with the banded color display to get values below or above a certain threshold within a band of color to make the results easier to interpret.
- **Report:** The program can generate a local html report, which is obviously a great way to share results with your colleagues.
- **Solver data:** This option allows you to access the solver output file. The solver output file may not generally be of interest to all, but it is possible to see when the analysis was run, can display insights in to warnings and errors and I have, at least on one occasion, seen it as a reporting requirement to be delivered as part of the consultancy work.

Inspect tool from the toolbar

The list of post-processing options shown in the image below is almost entirely consistent across the eight analysis types that we have introduced, with very little exception. We'll briefly detail what these options do.



Display of Inspect Tool from the Toolbar – Captured from Static Stress Analysis

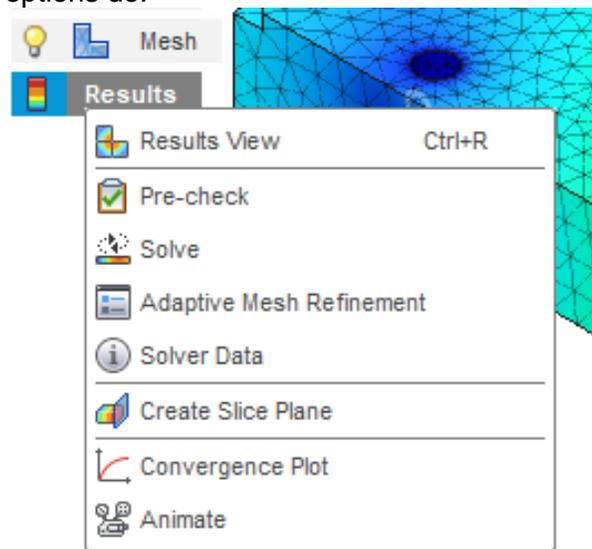
- **Measure:** The measure tool exists here on this tool menu, but is not specifically related to results, note how it is greyed out. The measure tool can be leveraged in the Model View.
- **Hide Min/Max:** As noted in the earlier sections, when you finish any of the analyses, the min and max locations of the displayed result type are highlighted by these probes. To clean up the model display, say – for capturing images, they can be hidden. Note when they are hidden, then this menu option will dynamically switch to read “Show Min/Max”.
- **Create Surface Probes:** This powerful results tool allows you to move over the geometry and it will dynamically display the value of the current result type. If you want to maintain the display at a location, left click and the probe will be attached to the model. If you change the result type, say from Von Mises to displacement, the value in the probe should automatically update to show the value for the new result type. Further, if you add any user defined probes, the Inspect menu tool will now include additional menu options of “Hide All Probes” and “Delete All Probes”.
- **Create Point Probe:** While the surface probes discussed above apply to the surface of the geometry, the point probe can be placed on the geometry and then repositioned in cartesian space using the grips or x, y and z inputs. This is useful for comparing results at a specific location across iterations of a geometry. E.g., if one was instructed to note temperature at 1 inch over, 2 inches up and .5 inches in from the lower right corner of a design, possibly to match where a thermal probe might be placed on a physical test.
- **Reactions:** This allows you to sum reaction forces and moments on the geometry. This is a great method for ensuring that constraints of the geometry “see” what your anticipated applied loads are. Tip: The values can be posted to a clipboard in case you need to paste them in to another program, such as an Excel spreadsheet. Note that reaction results are not applicable in modal, buckling, shape optimization, or thermal analysis. For event simulation, the calculation of reaction forces is a settings option that should be invoked before starting the simulation if this output is of interest.
- **Create Slice Plane:** When you select this option and move over the geometry, you will see preview planes based off the geometry surface you are moving your cursor over. When you have a plane and general location where you want the slice plane to be, left mouse click again to generate the slice plane. Once the slice plane exists, a menu will open to allow control over the plane; precise positioning, angle, style of display of results on the plane, and flipping the direction of the cut. You also have grips that will appear that allow you to rotate or drag the plane if you prefer.
- **Component Color Cycling Toggle:** Like the Measure command, also a tool not for results, but can be utilized in Model View. This will make different components different colors for a quick way to determine different components.

Third and final results stop – results options from the Browser.

Along the left-hand side of the canvas we have our browser. The browser contains history of and access to many of the items we have set up with our analysis. At the bottom of this list you will find an entry for Results.

Results from the browser

The list of options shown in the image below is almost entirely consistent across the eight analysis types that we have introduced, with very little exception. We'll briefly detail what these options do.



Display of Options Via Results from the Browser – Captured from Static Stress Analysis

- **Results View:** As this menu is accessible from Model View (and others), the Results View, of course, will take you to that view and display results for the model if they exist.
- **Pre-check:** Not directly results related but will perform a basic check to see if the minimum amount of information has been provided in order to execute an analysis.
- **Solve:** Begins the analysis if it needs to be analyzed.
- **Adaptive Mesh Refinement:** Note this is not applicable to all analysis types. This will access the adaptive mesh refinement setup, also accessible via Manage; Settings; Adaptive Mesh Refinement
- **Solver Data:** Accesses the solution data. Please see description earlier in document where the Result tool from the toolbar was discussed.
- **Create Slice Plane:** Accesses the slice plane tool. See prior detail regarding the Inspect tool from the toolbar.
- **Convergence Plot:** When an adaptive mesh refinement analysis has been invoked, will access a graph of the results, as discussed prior. Please see description earlier in document where Result tool from the toolbar was discussed.
- **Animate:** See description where Result tool from the toolbar was discussed.

This concludes the first learning objective of learning the tools that Fusion 360 provides to interrogate the simulation results.

Learning Objective 2: Learn some ways to increase your faith in your simulation results.

It has been my experience, in product support over the last (roughly) 20 years, that there are certainly times when someone knows exactly the results that they are looking for. Just some of the reasons that they are so certain about the results that they are expecting could be because they are reproducing a text book problem with a known solution, that they did the hand calculations themselves, they are re-analyzing something that was done some years prior, they are working to match results that a colleague did in a different, or the same, finite element analysis package, or they could have results from a physical test they are attempting to reproduce. On, the other hand, there are also certainly times that I have also spoken with users and they just are not sure what the results should be, which could be that it is a new complex design, perhaps have 2 competing results – but both seem plausible, perhaps they are a finite element analysis focused person, but the problem is not something typically in their field of study.

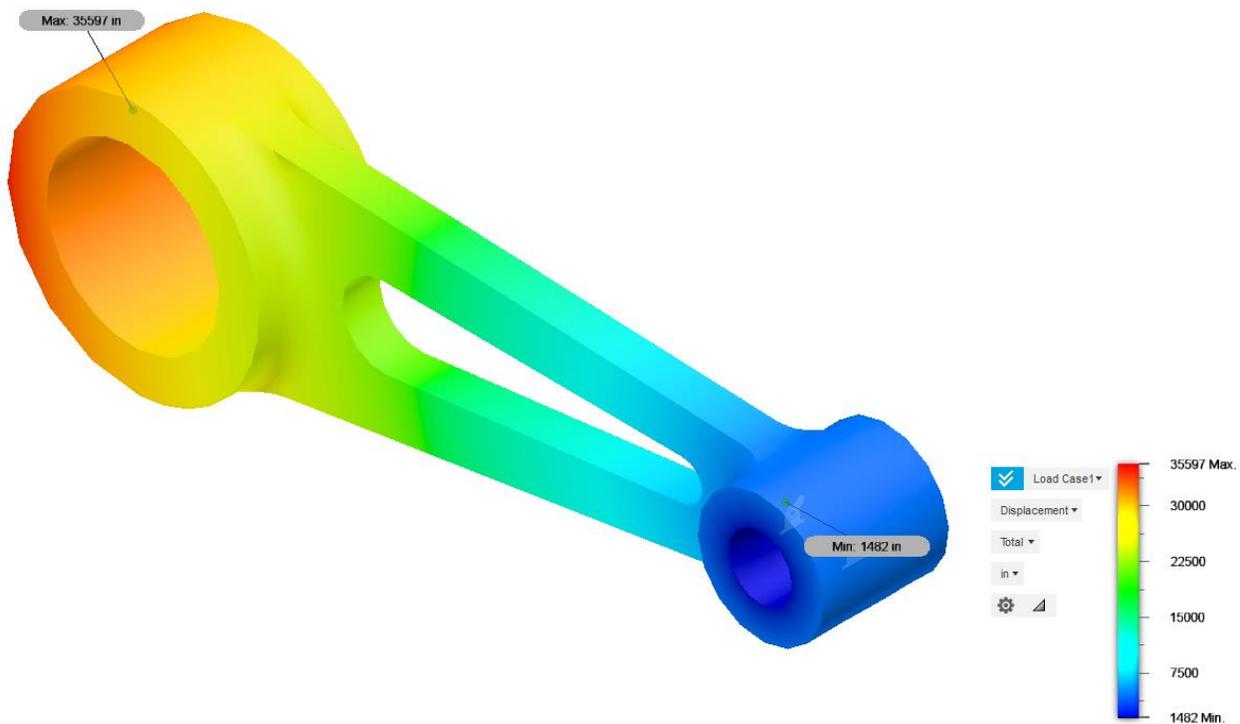
Whichever of these two above scenarios we have, I would say there is a good chance that at some point, you will have some results in front of you that either don't match what you expected to get, or, you just have uncertainty about what you have produced and whether the results are good. Well, the results are most likely always good for what we have entered in to the program, perhaps a better way for me to write that is - how can we help to ensure that the results reflect what we want and solve the problem we were anticipating solving?

As I begin to get in to my set of suggestions, I would only ask that the reader keep in mind that the overarching theme of this is around the Fusion 360 program and results. I hope to, below, provide a useful list of suggestions that will assist you in reinforcing your confidence in your results, but it may not necessarily be a completely exhaustive list. You may have some different practices that you employ to accomplish the same or similar results and, if those work for you, I would encourage you to continue to use them. If I share one item here that you keep in mind to leverage at some point, or reinforce techniques you already use, that would be fantastic. To follow are some of my recommendations, in not so much any particular order:

Review the Displacements

In the prior learning objective, we reviewed the various element types, the default type of result that it displays, and what other types of results are available. Over the years of my teaching classes on finite element analysis, I would suggest that it makes sense very early on in reviewing the results to review your displacements. There are a number of reasons that I would suggest to review the displacements as the first, or one of the first results;

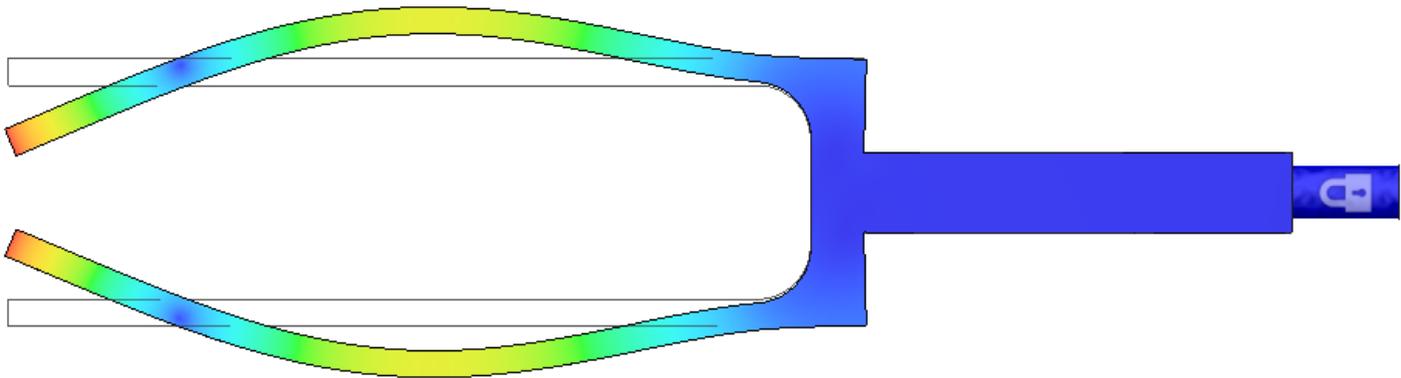
- FEA basically solves $\{F\}=[K]\{x\}$, most closely in linear static stress. The forces are known as they are applied by you in the setup of the model. The stiffness matrix can be determined by your geometry, the mesh, and the material that you have assigned. Therefore, the determination of the displacement is based on very little and things that should have been input correctly. It is easy for most finite element analysis programs to quickly converge on an appropriate displacement.
- If your displacements look “off”, then this is a good time to pause and review your simulation. The strain and stresses are computed based of the displacements that have been solved. So, if the displacements are wrong, there is a good chance that the strain and stress values will be also.
- If you are sure the displacements are off, to troubleshoot, go back to what goes in to the equation. Are the forces (or loads) of the correct magnitude and units? Are the material properties input appropriately? Are the constraints in the model where and what you wanted them to be? These would be my first checks at diagnosis.
- If the displacements still seem off, but loads, constraints and material all check out, then the next place that I would look is whether the model is an assembly, and if so, does it have contact? If the model has contact, did that work as anticipated or are there maybe changes that I need to make there?
- Keep in mind that one of the basic presumptions of linear static stress is small displacement, so, if the analysis you have is legitimately to produce large deflection/displacement, then you should consider a different analysis type.



Model Is Just a Few Inches Long – Displacements Unreasonable

Toggle the Wireframe

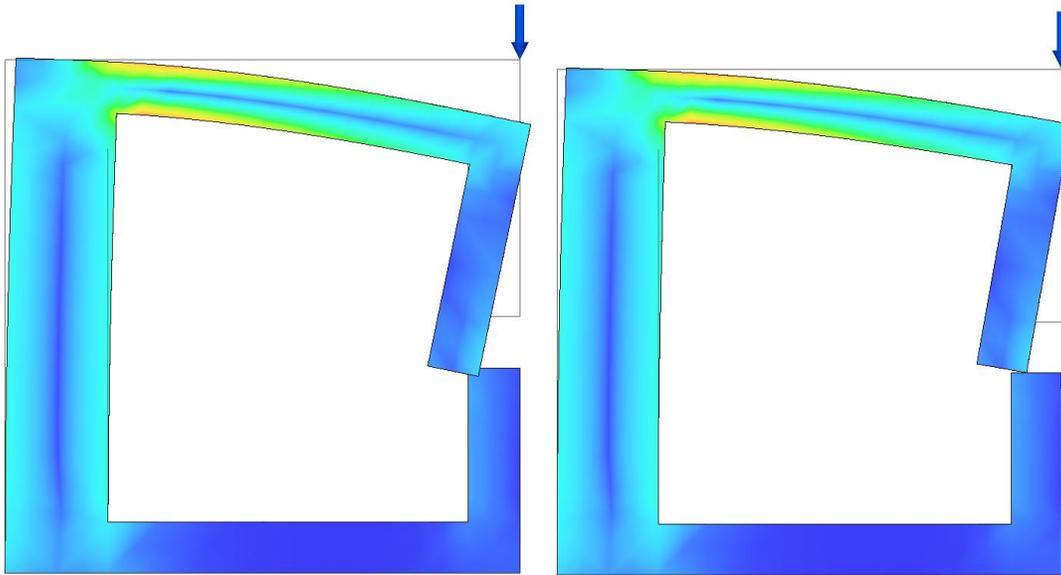
A seemingly simple suggestion, related to reviewing the displacements, I would propose that you try and toggle the wireframe on and off, while looking at the model from different views. Sometimes the wireframe of a complex model gets in the way of viewing the deformed geometry, causing too much clutter, making it difficult to see what is happening with the results. Other times, I have noted that toggling it off and on while looking at the displaced geometry gives perspective between the displaced and un-displaced geometry, which just might help to trigger the understanding. Recall you can access the option Toggle Wireframe Visibility from the Results tool on the toolbar.



Wireframe Can Help Us Understand Deformation – Such as with Interpreting this Vibration Mode

Adjust the Deformation Scale

A third and final recommendation around the idea of reviewing your displacements, recall that you have the Deformation Scale tool under the Results tool in the toolbar. At times, the deformation might be so small, that by really increasing the deformation scale, you can gain an understanding of the deformation trend and decide whether it is expected or not. In other situations, the deformation scale might be too large, giving the impression that something is happening that isn't. An example of this is when parts must close a gap prior to making contact. If one part, say, travels $\frac{1}{4}$ " prior to making contact and does make contact, but I exaggerate the deformation scale 2x, it will appear that the part has moved $\frac{1}{2}$ " and passed through the other object. In this case, reviewing that area of the geometry at actual scale is probably best, to determine if contact was made or not.

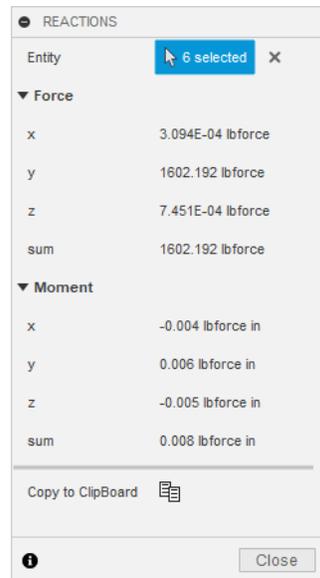


2x Adjusted Appears to Pass Through, Actual Scale Shows Contact

Review the Reactions

For analysis types where this result is an available output, a great and quick check is to look at the reaction forces at your constraints. Reactions are available from the toolbar under the Inspect tool. This check does a couple of things for you. First, if you made a mistake on applying the loads on the geometry, then you will find that the reaction forces don't add up to what you anticipated. It is possible to have simply typed in a wrong value, to have accidentally selected more or less surfaces when you applied the loads or to have accidentally deleted some of the loads you intended to have on the geometry. Another benefit of checking the reactions is when you have an assembly, the more complex it is, the more beneficial this check is – to make sure that the load is propagating through all the parts of the model. If not, it can be related to contacts not applied in the model where needed, or not working as you had anticipated, or there are gaps between parts, such that the bodies are not transmitting the forces from one to the next.

- Note that not all analysis types output reactions and recall that if you are running an Event Simulation, that you need to turn on the option to output reactions.

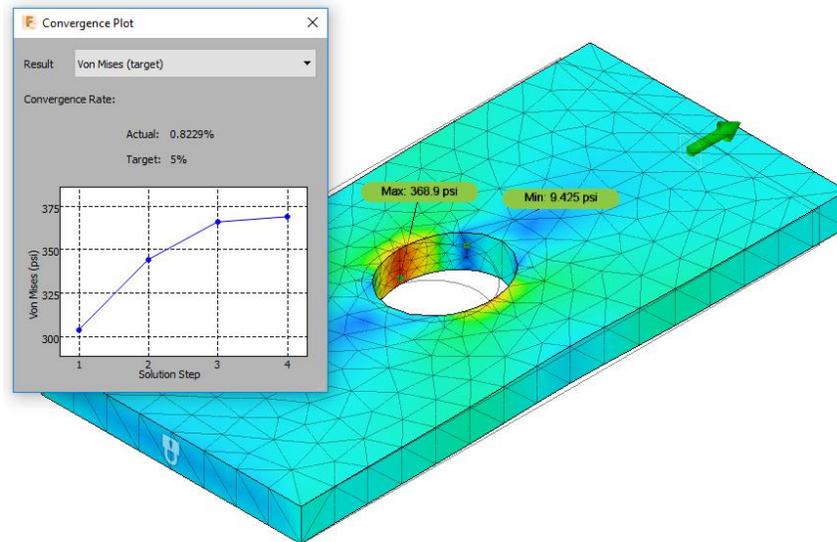


Reactions Menu Showing the Sums of Six Selected Surfaces

Leverage Adaptive Mesh Refinement

If the analysis type you are using allows for it, you can find the Adaptive Mesh Refinement under the Manage tool; Settings. Adaptive mesh refinement informs the program to run several analyses and, while doing so, to consecutively refine a portion of the elements and check the percentage of change between consecutive runs and to stop iterating when the tolerance has been met or the maximum number of iterations has been reached. This process can be stress or displacement based. At the end of the analysis, in the Results tool, it is possible to review the Convergence Plot and ensure that the analysis satisfactorily reaches a converged state.

- Tip 1 – this is excluded from some analysis types, such as a nonlinear static, because the solution times for this analysis type can already be towards the longer end. Adding the adaptive mesh refinement to an already computationally expensive run could make for excessively long solve time. It might be possible to run the geometry first pass as a linear analysis to help gain an understanding of appropriate mesh sizing and refinement and then run the nonlinear static analysis with a similarly configured mesh.
- Tip 2 – note with the adaptive mesh refinement slider that if you adjust it to the far right, that there is a Custom setting. This then “unlocks” the fields so that you can specify, if desired, you own inputs for maximum number of mesh refinements, the results convergence tolerance, and portion of elements to refine.



Result of Analysis with Adaptive Mesh Refinement and Convergence Plot

Hand Calculations

I am not here to judge you for whether you have hand calculations when you begin your simulation or not. That said, I can positively say there have been a few times over the last so many years of my supporting finite element analysis, that the end user and I would not have come to a satisfactory resolution of the question if one of us had not performed the hand calculations. At times, someone feels very strongly that the finite element analysis results are in question and it turns out the hand calculations support the fea results, which was perhaps counter to their intuition or perhaps what they were told the outcome should be. On the other hand, I can recall very well an incident several years ago where I felt strongly that there was no way the program could be producing wrong results on a seemingly very simple analysis. It turned out that the hand calculations supported the user's insights and not the program results. This resulted in us reviewing the model much more closely and us finding a very simple but not obvious at all geometry input error.

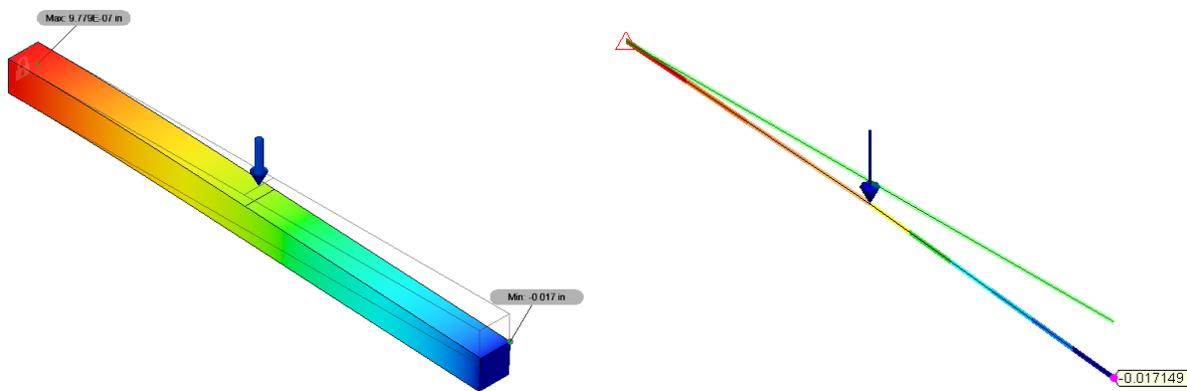
Other Programs

We are here, of course, to focus on Autodesk Fusion 360, but, this is not the only finite element analysis program that I have on my computer. From Autodesk, embedded in is the Inventor Simulation module, also embeddable within Inventor is Autodesk Nastran In-CAD and I also have the (now "sunset", as of 2018) Autodesk Simulation Mechanical. All these programs have the same purpose, to take you geometry, material, loads and constraints and to output what the resulting behavior is. While they might have differences in the interface or levels of capability, for the same geometry, set up in the same way, they should be able to produce very similar results. I will most certainly sometimes check the results of one of my simulations by setting it up in another to check the results. Granted, I could still be setting it up wrongly in both programs, making improper presumptions, but if I get the same results in both, it at least

increases confidence in the program. Note that Nastran In-CAD and Fusion 360 use Nastran solver technology, while Inventor simulation and Simulation Mechanical were based on two other programs, so leveraging different solver technology to arrive at similar results further increases confidence.

Other Elements

At this point in time, the Fusion 360 interface is heavily reliant on solid elements (bricks). This is useful for a wide array of simulations. Some other finite element analysis packages can leverage some other element types as idealizations of geometries, for example, using a line element to represent a beam, where the shape of the beam is entered in to the program mathematically, rather than modeled in full 3D. I would caution to approach this suggestion with some care to understand that different elements have some different particulars about their behavior. However, from time to time, I have found it useful to attempt to set up an analysis using different element types to see if I at least get somewhat similar results – to gain confidence in a result I am seeing. If you were just modeling a simple cantilever beam with an end load, this model can be analyzed in finite element analysis using the beam idealization, shell/plate elements, or – with care, solid brick elements. So, it could be useful in some instances, if someone has performed such an analysis in another program using beam or plate, that you could potentially get relatively similar results in Fusion 360 using solid elements to correlate and confirm some outcome.



Solid Model on Left, Idealized Beam on Right. 1in.x1in.x16in, 100lb. Mid-span Load. Both Arrive Right Around Theoretical Deflection of 0.017in.

Other Colleagues

Related to the above two suggestions, sometimes it is possible to get so in to a analysis that it almost becomes impossible to see what you are not seeing. If you have colleagues who have access or expertise with other programs and/or element types, they might be able set a similar analysis such that you can then compare the results and look for any differences, if there are any. In addition, whether your colleague has any finite element analysis experience or not, it can certainly be beneficial to talk through it with someone. They might ask a question about your setup or presumptions that uncover something that you might have overlooked or had forgotten to consider. If you are attempting to simulate something in production, they could know something about the actual installation or operating condition that is different than the

documented process, which leads to different measured performance, such that your finite element results are just representative.

Tutorials

There is a set of tutorials or hands-on exercises that can build your confidence in the program. While there is unlikely to be a tutorial that is the exact setup of the problem you need to solve (wouldn't that be nice), if you are new to the program or just get started with a different type of analysis (say, just starting in to thermal analyses or shape optimizations), the tutorials can provide some experience and perhaps some considerations that you had not thought about. The Hands-On Exercises are located here; [Hands-On Exercises](#)

- Tip: If you are feeling particularly confident, feel free to read a problem setup and description, go ahead and set it up and analyze it on your own and then go to the end of the tutorial and see if you were able to get the solution without reading or reviewing their step by step. If your results do differ, then you can back and review and find where you took a different path. This also then opens up tutorials from other software packages as well, if you stumble across them online or maybe find a manual around the office.

Verification Examples

To borrow from the documentation on the accuracy verification examples; "...written to assure customers that the Fusion 360 software performs consistently and accurately within its defined scope. It contains a number of Accuracy Verification Examples (AVEs), which compare analysis results with analytical solution for cases taken from NAFEMS benchmark publications. Other examples are taken from theoretical sources, such as Roark's Formulas for Stress and Strain. Each AVE page specifies the source of the reference benchmark." Let's face it, we are bound to encounter that one person who is suspicious of the program's results until you can show them proof that it can obtain a known solution. The AVEs can be found from here; [Accuracy Verification Examples](#).

- Tip: It can be a good learning experience just to pull one of your engineering texts off the shelf and try to reproduce a problem with a calculated solution from there. Knowing the target value from the book's solution lets you know when you are successful and attempting to go from paper description through finite element analysis setup can build up your analysis skills.

Experience and Observation

Your own life and job experience, what you have observed over time should not be discounted just because the program has given you a result. Based on my own experience, what people would sometimes call "gut feel" can also be valid, as it is your experience and exposure telling you so. If you perform an analysis, the results come up and you suddenly "feel" that the results appear to be unexpected, it is certainly worth a few moments to double check some things out. Take some time to leverage some of the methods prior mentioned; review displacements, reactions, your inputs and so on until your comfort level returns or you discover the source of your unease.

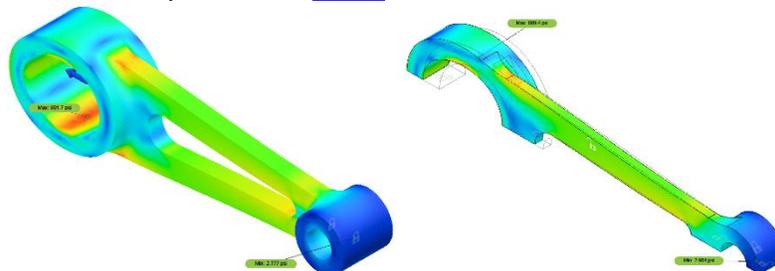
Physical Testing

I would say, and I think most people would agree, that one goal of finite element analysis would be to reduce the amount of physical testing required, but not to replace physical testing altogether. If you can run several simulations to test out several design alternatives and rule a couple of them out based on the results, that is fantastic. Perform the physical testing or further studies on the remaining candidates to help select the best design. A counter argument I have heard is that simulation is so slow that someone can physically set up and test something before you can get those simulation results. Use that to challenge yourself to become more proficient with the program and find more efficient ways to perform the simulation. Using Fusion 360, you can send multiple scenarios to cloud... what is their capacity run multiple physical test concurrently? But, again, they are not at odds. The physical testing results and finite element analysis results should obviously correlate. Note that there can remain differences due to how something must be set up in finite element analysis, or how it must be set up in the physical test or production environment. Learning the source of these discrepancies only helps you to understand your product better and explain things more quickly in the future.

Symmetry

When someone has run a simulation, has some question about their results and I review that model and see that a full model has been used, but planes of symmetry exist, such that they could analyze it again as either a half or quarter model, I love to be able to recommend it.

- Setting up the model again as a symmetric model gives you the opportunity to go through your setup again, which potentially allows you to uncover a possible misstep that you took the first time through.
- Using a symmetric model, you now have a smaller volume to deal with, which means that you can leverage a finer mesh size and still fall within the same or shorter amount of analysis time.
- Constraints that are applied to the symmetric faces can have the bonus of helping to restrain an otherwise under constrained model. Imagine, for example, that you want to see the stresses in a soccer ball when inflated. Attempting to constrain this geometry without influencing the results is a tricky prospect for the full model, however, in 1/8th symmetry, the symmetry constraints you need to add would sufficiently constrain the ball without adding unwanted constraint. There is a sample of using symmetry in one of the accuracy verification examples here; [AVEs](#)



Full Model vs Equivalent Quarter Symmetry Model – Von Mises Compare Within 1%

Nastran Solver Technology

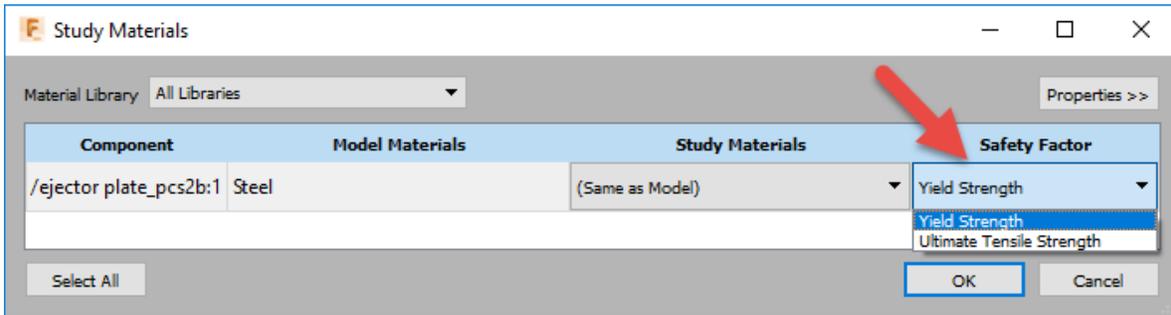
Perhaps more for your helping the confidence of others in your simulation results, but as a relatively newer simulation software, it is possible that some long-time analysts question the technology. As pointed out in a prior section, the solution leverages the Autodesk Nastran solvers – a function of the acquisition of NEi Nastran product in 2014. According to Wikipedia (https://en.wikipedia.org/wiki/NEi_Nastran), the first commercial version of NEi Nastran for use on PCs was released in 1990, so the tested technology has been around for a while, while the Fusion interface for it is significantly more recent. The Nastran product has name recognition that many who have been involved with finite element analysis would recognize, but of course, also you can leverage some of the tips above as well, such as running some accuracy verification models to validate.

Learning Objective 3: Discover some of the common results-related issues.

In this section of the document, I would like to highlight some of the issues that you might encounter during your usage of finite element analysis. More specifically, again, since the overarching theme of our document focuses on the results, I am concentrating on a set of issues or problems that might arise, where the problem could be discoverable by reviewing the results presented to you in the Results View of Autodesk Fusion 360. In our final learning objective, later in this document, we'll then address how to resolve these items. This current learning objective is just about the discovery of these items.

Safety factor review

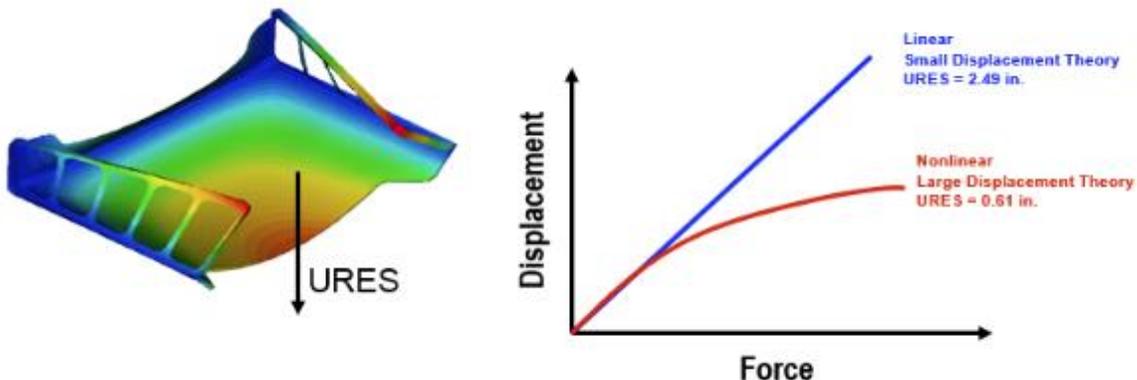
As the results of several of the analysis types present this result to us as the first result once we have run the analysis, we will discuss this first. The outcome of the analysis, as shown to us through the safety factor, will be that our design is either below, at, or somewhere above the yield strength (the default criterion). If the minimum safety factor is less than 1, then we know that we have areas of the model where the currently applied constraints and loads, for the given material, cause the design to experience stress beyond yield (presuming default was not changed). Obviously, at 1, stresses are equal to that yield and safety factors greater than 1 mean the computed stresses are less than yield. The equation utilized to calculate the safety factor is $\text{Safety Factor} = \text{Material Strength} / \text{Actual Stress}$, where the material strength is either the material's yield strength or the material's ultimate tensile strength. Whether it is based on yield or ultimate is defined when you define the material, referencing image below. Certainly, if the safety factor is at or less than a value of 1, this is cause for investigation. At the same time, if the design comes back as something marginally safe, it is probably still a good idea to investigate further, particularly if the adaptive mesh routine has not been utilized. Also, keep in mind that you can hover over the legend to grab one of the handles to adjust the range of mesh that is displayed on the model. This can help you to isolate where in the model, perhaps a particularly low safety factor is showing up, such that you can determine whether this is a cause of concern in your design, or perhaps an artifact of how the model is constrained or loaded. In our fourth and final learning objective, later in this manual, we discuss some items that could be looked at in regard to an unusual safety factor.



Safety Factor's Material Strength Choice of Yield Strength or Ultimate Tensile Strength. Choose When Defining Material of Components

Displacement results unusually large (or smaller)

As has been brought up in prior sections of this document, I have always suggested that one of the most important things to do when reviewing the results of a finite element analysis would be to take a good look at the displacements. I think it is not unreasonable, particularly when one is getting started with FEA, that they would want to quickly jump to the stress results as a quick check to see where the results may fall with respect to yield, or within their imposed stress limits, but, reviewing the displacements of the model can tell you much about the model. For a linear static stress analysis, keep in mind that it is one of the limitations of linear static stress analysis that you need to have relatively small displacements and rotations. As a rough guide, keep the rotations around 10 degrees or less and the deformations it is usually suggested to follow the 1/10th rule (displacement 1/10th the overall scale of the geometry for a global deformation or 1/10th the thickness for a local deformation). There is some information in the Autodesk Nastran In-CAD documentation concerning discerning limits [Linear vs Nonlinear](#). Of course, the opposite of having that problem would be reviewing your displacements and finding that they are much lower than you had anticipated. There are a number of things that could be looked at in regard to that as well, and we'll outline them in our fourth and final learning objective at a later point in this manual.



Stress Stiffening Captured by Nonlinear. Note Graph, Linear's Incorrectly Large Displacement. (Image courtesy Autodesk Knowledge Network Nastran In-CAD)

Reviewing your stress results

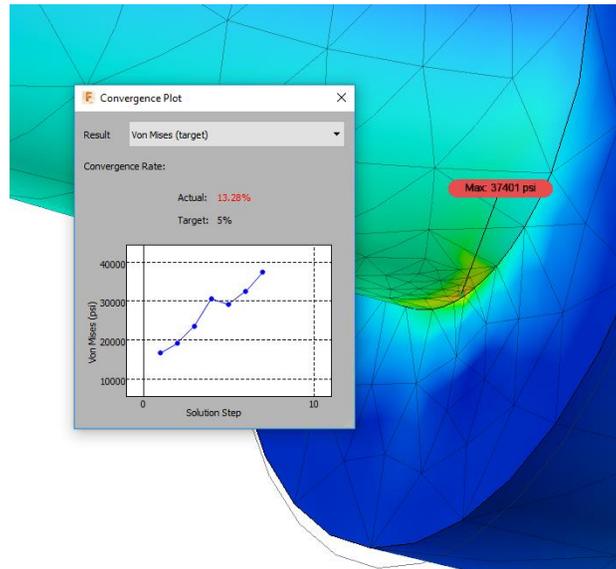
After you have looked at the safety factor and the displacements and deemed those to be reasonable with regards to what your expectations, calculations or experimentation tells you, now it is time to review the stress results. The stress results could end up being higher than anticipated, lower or within reason. Certainly, being much higher or lower than expected would be cause for further investigation, but what about the case where the results seem reasonable? I have seen it often the case that by “reasonable”, people often mean that the stresses don’t appear to be ridiculous and provide a reasonable safety factor. But are the reasonable stresses correct? This goes back to some of the things that we discussed in our second learning objective... what sort of data do you have to compare these results to? If you have hand calculations or experimental data and your findings from the finite element analysis match those results, well, then that is fantastic. On the other hand, if you have nothing to compare these results with, I might propose then that the best thing to do (barring hand computation or experimental) would be to leverage some of the prior suggestions, such as leveraging a symmetry model if you can, using adaptive mesh refinement or comparing against another program, such as Autodesk Nastran In-CAD to confirm. Please also keep in mind that there are a variety of stress results to choose from (Von Mises, principal, normal and shear). The Von Mises stress will always be a positive, as a SRSS type of computation. Tip - When comparing results that you obtain against any other program, ensure that you are looking at the same stress result type. For more details on the variety of stress results and how they are computed, please refer to the following Fusion documentation; [General Structural Results](#)

$$\sigma_v = \sqrt{\frac{(\sigma_{xx} - \sigma_{yy})^2 + (\sigma_{yy} - \sigma_{zz})^2 + (\sigma_{zz} - \sigma_{xx})^2}{2} + 3(\sigma_{xy}^2 + \sigma_{yz}^2 + \sigma_{zx}^2)}$$

Von Mises Stress Calculation. (Image courtesy Fusion Help on General Structural Results).

Singularities

As a subset of reviewing the stresses, it would be wise to consider the fact that some finite element models will produce very high stresses in some localized locations of the geometry. In identifying when you have encountered a singularity, it would be noted that if you continue to change the mesh size (making the mesh size finer) that the resulting stress will continue to increase in magnitude. Singularities can arise from a few different conditions, but if the abnormally high value stress is localized around one of the following regions; including the location of a point load, sharp corners or boundary conditions, there is a fairly good chance you are dealing with a singularity. Again, refining the mesh in this area to find that the stresses only continue to increase would provide further confirmation that this is what you have encountered. We will consider some options we have when looking at our fourth learning objective.



Singularity on Stepped Diameter Shaft. Not Converged after Seven Iterations.

Stress concentration

A stress concentration may initially look like a singularity, where you have a larger stress in a rather localized area, however, stress concentrations will eventually converge on a solution with a refined mesh. The stress concentration would typically be a function of the geometry. Classic examples of this include around a hole in a flat plate, or notch in the side of a flat plate, or in the shoulder of an increasing/decreasing diameter tensile specimen. Fortunately, this one is relatively easy to find and remedy in current finite element analysis programs.

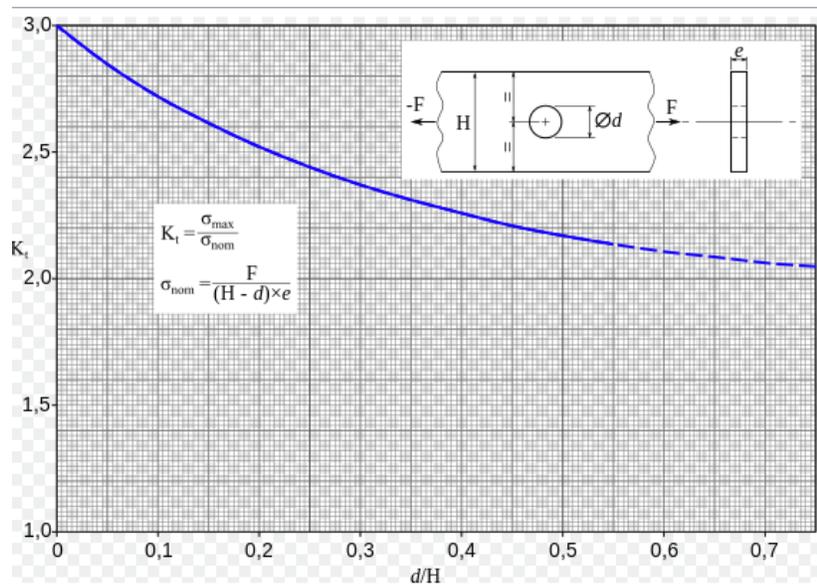


Chart of Theoretical Stress Concentration Factors (K_t). (Web image – labeled for reuse)

Learning Objective 4: Learn how to correct some of the common results-based issues.

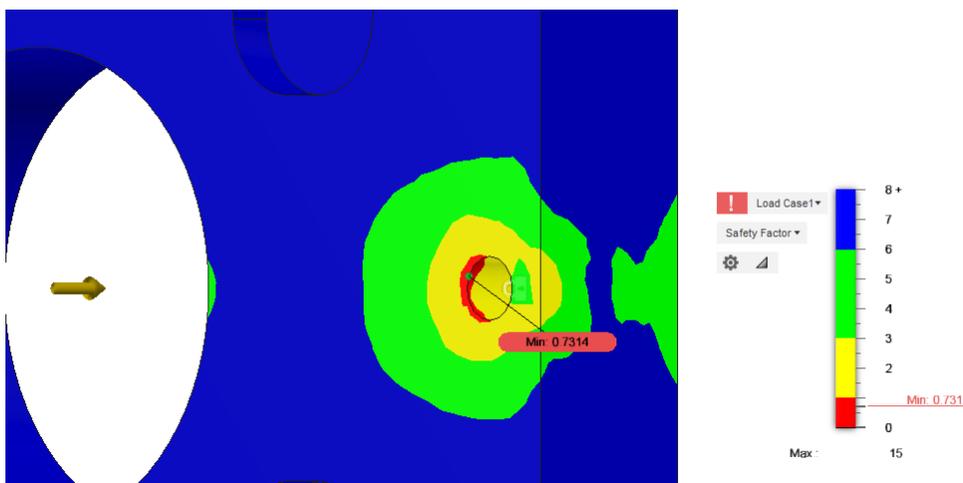
In this final section of the document, we will revisit the items from learning objective 3 and determine what we can do to correct the issues when they occur or avoid them in the first place.

Safety factor

It is probably apparent that the concerns with respect to safety factor, probably in order, are being under 1, then right around (or less than set criteria for the company's set limit), and finally, considerations for exceeding.

Safety factor less than 1

If the resulting safety factor coming out of our analysis shows value less than 1, what we know at this point is that the geometry we have put in to the program, along with our defined material, loads and constraints has produced resulting stresses that exceed the material strength that was input – which should correspond to the material property of either the yield or the ultimate strength. I think in this case, that one of your first next steps would be to look at the geometry and determine where this is happening and what percentage of the geometry we are talking about. If this happens in a very finite location, we can consider whether it is a function of perhaps how we have constrained the model or how we have loaded it (e.g. the load is at a vertex, an edge or very condensed region). Based on what we observe there, we can consider whether it warrants further investigation or able to be dismissed as a local and/or artifact that will not have considerable impact on the production component(s)' performance.



Safety Factor <1. Localized at Sharp Corner Where Hole is Fixed

In the example image above, it is likely that if we just constrain the model slightly different that we can eliminate that localized issue. On the other hand, if we look at our

geometry and large portions are less than 1, when this is unexpected, then I would propose that there are a couple of things that can be done. I know that the first thing I would do is go back to the model environment and review my setup. Look to see if the load you applied agrees with what was specified to you (check units and make sure that there are no input typos), double check the material to ensure that the material chosen is what should have been and that the properties for the given material look appropriate. Thirdly, take a good look at the constraints on the geometry and make a determination if they are appropriate for what the real world or in-service conditions will be. Hopefully you find that there is a simple input mistake to correct and then a subsequent analysis proves a good design. If, however, verification shows that all your input is correct, and if these results come from a linear static stress analysis, keep in mind that this result is telling you that the stresses have exceeded the yield. A linear static stress analysis does not have the ability to produce appropriate post-yield behavior or results. If you are going to need to discuss the findings of such a design, you might want to consider setting up the geometry as a nonlinear analysis to be able to share some insights as to what the design will look like in the post-yield state.

Safety factor greater than 1, but still marginal

As noted in the product documentation learning material [FOS](#), “Autodesk, Inc. cannot dictate what safety factor you should achieve in your design work. It is the customer’s responsibility to research safety factor guidelines and to ensure compliance with applicable codes, design standards, and acceptable practices.” So, it is up to the designer to ensure that they are aware of any codes, standards or design practices (perhaps internal to the company) and target results that would fulfill that criteria. Presuming that your design is close to this criterion, but perhaps a little too low or a little too close to the lower threshold and so it is decided that a design change will be made. Consider that the safety factor is just a ratio of the material strength divided by the calculated stress. We know from basic engineering that stress is equal to force divided by the area. Or, more general, the stress is equal to the load divided by the area. From this, we can take away that if we need to strengthen our design, we can impact this by changing the load or the area (geometry change to the design). Along with that, the stress and strain results are dependent on the deflection, which we know, based on Hooke’s law is force equals stiffness multiplied by displacement. The load may not be changeable (we need the design to sustain a certain load), but the stiffness is a function of the material and the geometry of the part. When we combine these considerations with the example image just above (where the constraints caused a local low safety factor), then we then have the following choices to influence the outcome; load, material, geometry, and how the model is constrained. Probably more than one of these are non-negotiable in your design criteria (e.g. the material must be ‘x’), but likely something else can be changed.

Safety factor greater than design criteria

When our design produces large values for the safety factor, this doesn’t necessarily indicate any problem, it means the calculated stresses are all low, so you may be wondering why this is contained as a point in a learning objective of how to correct some common results-based issues. The first thing that I would say about this is that it is

always wise to double check or triple check our results. We know that there are multiple ways to do this, and we outlined many of them in learning objective 2. Presuming that our validation tells us that it is, in fact, a very safe design – you may be done with your work, or you can think more about what that means. Do you have an overweight over-designed part or assembly? Can you change to a lower strength lower cost material? Could you run a shape optimization analysis and find locations where material could be removed from the design? Less material, of course, could result in savings on material cost or shipping costs.

Displacement (deflection)

It has been mentioned a couple times prior in this document, so you can probably tell by now, one of my favorite “go to” results to judge the outcome of the analysis is the displacement result, showing the deformation or deflection in our design. It should be easy for the program to arrive at rather accurate displacement results, again – based on Hooke’s law, there is little that goes in to contributing to the calculation of the result – the stiffness and the load. So, very quickly, often we can tell if we have a problem with our setup or not.

Smaller than expected

It is not too often that I encounter designs in my years of supporting finite element analysis where people have asked for help in investigating deflections that they suspect are too low. When I have, it is typically the case that we closely review the input and find that a load was missed or input of the wrong (too small) magnitude or that it is a material input error. Both go directly in to the Hooke’s law equation, right. Occasionally, this can be mesh related, but it must be a grossly coarse mesh for that to be the source. My suggestion would be that these are the first things to closely review in this situation. Another possibility is that, at least in some cases, people look at the results, see that the displacements are nice and low (within their limits) and say to themselves “great” and move along with the review and process with the presumption that everything is well, when it may not be entirely accurate. One of the conditions above could exist (load or material input problem), or, there is also the question of whether the model is over-constrained. Consider whether the boundary conditions placed on the model represent the actual conditions or if maybe they have been used in an over-aggressively fashion. The nice thing about finite element analysis is that if you have some hesitation about that and some consideration about how you might alternatively constrain it, it is simply a matter of cloning your study (to preserve the results of the first analysis) and then make the changes in the new study, re-analyze and review the results.

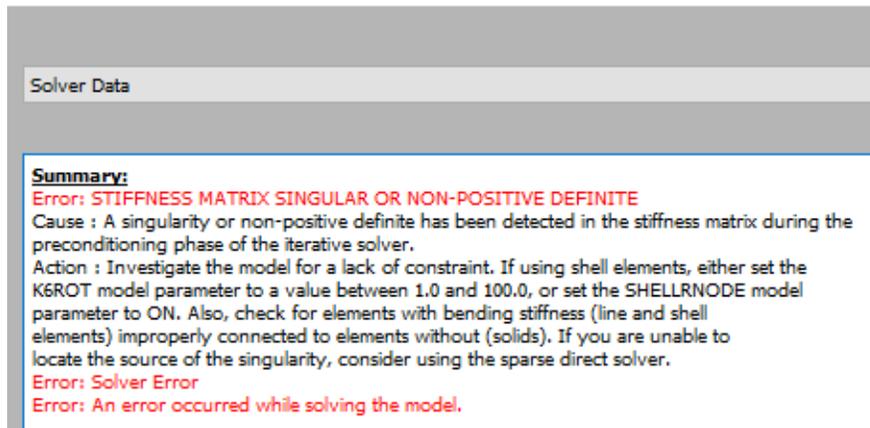
Larger than expected

When dealing with displacements that are larger than expected, the degree of how much larger (not always, but often) can point to the source of the problem. For instance, if you accidentally type in a value of 1000lbf for a load that should have been 100lbf for a reasonably substantial design, the deflection will likely give you some indication of that. Obviously, if your design is less substantial, having a load 10x of what you intended will make an obvious impact. Of course, regardless of the magnitude, if you are nearly certain that the displacements seem unreasonably large, you should investigate, but to

my point, I think that you could probably make two buckets of “the displacements appear large” and the “the displacements appear very large”. In the first bucket would often be the usual suspects of loads and materials. Having improperly large loads or less stiffness in the material than intended in your setup will lead to large deflections. In these cases, the solution is to correct the input. Perhaps less obvious is a case brought up earlier and that would be when the wrong analysis type has been utilized. Recall, for example, that linear static stress analysis anticipates that the deflections will be reasonably small. When you have conditions that call for a nonlinear analysis, such as when the material properties really warrant it (e.g. hyperelastic materials) or there are stress stiffening effects that would come in to play, the solution is to check it with both analysis types. The proper nonlinear material model will better approximate the stiffness of the material and bring the displacements in to reason and the iterative stress stiffening will produce smaller deflections than the static solution.

In the bucket of really large deflections, or displacements in some cases, these are more commonly going to result from something like lack of boundary conditions, which could be an oversight – which causes the model to move off in to space along some direction, or to perhaps rotate when it was not foreseen. The obvious solution in this case is to add the intended constraints. Keep in mind that symmetry can also be leveraged to help stabilize a model. If the geometry is an assembly and there is large deflection or displacement, you are going to want to review all things about the contact. Has the contact been defined, is it the right type of contact, what about the penetration type, are the parts close enough for contact to be considered... to name a few things. In short, when there are really large displacements, there is typically something missing from the setup that should have held it... whether that is a boundary condition or another part in contact. Generally, if the lack of constraint is too much and there is rigid body motion, at least in a linear analysis, the program will error with a message about it, as seen in the image below. In this case, the assembly contained an internal component that was not touching any other bodies and, as such, was free to move around. Finally on this topic, it is worth noting that the Fusion program has an option in Manage:Settings to allow for Remove rigid body modes. This option can help stabilize a model when it is difficult to constrain a model without artificially limiting or preventing deformation that would occur naturally. You can find documentation on this option here [Remove rigid body](#) .

F Solver Data



Solver Data from Under-Constrained Model. Note Action Statement Reads “Investigate the model for lack of constraint.”

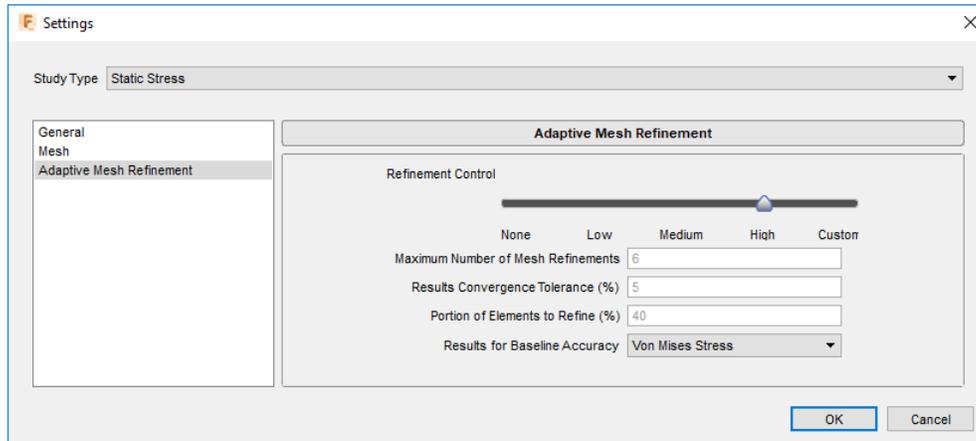
Stress results

With the stress results, we’ll be a little more brief here since the usual suspects covered under the immediately two prior sections apply (always check material, loads and constraints for issues). Further, we had a thorough discussion in the safety factor section and the stress results are what goes in to the safety factor. If your stresses are low, the safety factor would be high, so what was written in that section would generally be applicable. Likewise, if your stresses are very high, that would make the resulting safety factor low and then those discussions would be applicable. I will add some additional considerations here though.

Stresses are low

Let us first eliminate any issues with the inputs of the analysis (load, material, constraint) since we have already discussed these several times. As a first consideration, we’ll imagine that you have already reviewed your displacements, and everything looks good there. The displacements perhaps match your hand calculation and/or test data so that is all fine and well, you proceed to look at the stress results and they are lower than expected. Again, working off the presumption that everything else is appropriately set up, your suspect in this situation would likely be the mesh size. Having a very coarse mesh can potentially make the model overly stiff. If we rearrange Hooke’s law, the displacement is solved from the forces divided by the stiffness. In a little more appropriate fashion, we could write this as $\{x\}=[K]^{-1}\{f\}$, such that the displacement vectors are determined by multiplying the inverted stiffness matrix by the force vectors. Ultimately, the wrongly low displacements are used to solve for the strain and the stresses in the model. The first action I would take here is to refine the mesh. There are several aspects about Autodesk Fusion 360 that I like when it comes to this. Since we can cloud solve, it is possible for you to create multiple studies of different mesh sizes and send them off to the cloud for solving with no real impact to your own machine and your productivity. Another thing is that located under the advanced settings of the mesh, you will note options for element order options of parabolic and linear. In addition is an

option for curved mesh elements. The options will help increase the accuracy of your solution and more can be read here [Mesh Quality](#). Finally, the program contains the adaptive mesh routine, which can help ensure an appropriate mesh refinement is utilized.



Adaptive Mesh Refinement

Stress results too large

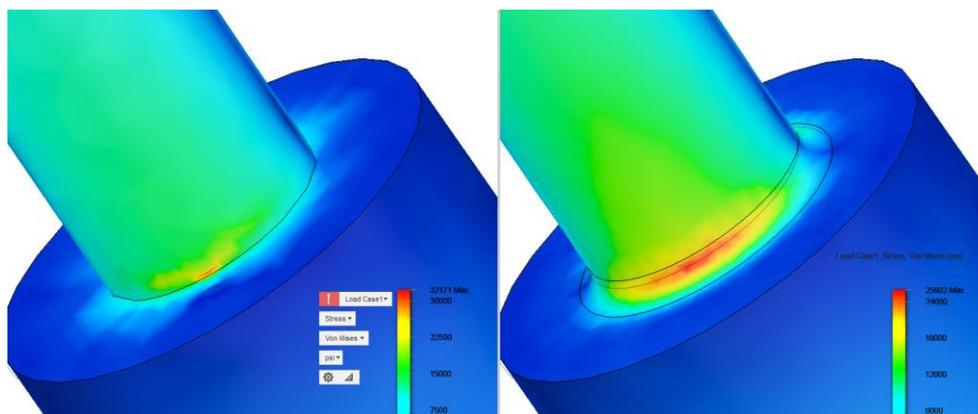
If the resulting stress results are too high, we know to check the standard items (loads, constraints and materials). Keep in mind that a way to check the loading, in the post processing of the model, that is – in Results View, would be to sum the reactions and ensure that they agree with what we know the applied load should be. The discussion that has been had on analysis type applies here as well. If the stresses of the linear analysis are exceeding the yield of the material, you should consider re-investigating the model again with a nonlinear static stress or event simulation analysis. Additionally, if the stresses are rather large, but still potentially below yield, it might also be wise to run a buckling analysis. Geometry is susceptible to buckling, especially if it can be classified as a long slender profile, even when the loads applied have not caused it to go beyond the material's yield strength. Finally, as we'll investigate in the next two sub-topics, singularities and stress concentrations can also give rise to larger than anticipated stress results.

Singularities

Singularities were first brought up in learning objective 3 and there we noted that singularities can come from a few different situations, these include sharp corner, point loads and boundary conditions. Singularities are generally pretty identifiable because of there being something unique about where it occurs (e.g. the sharp corner, the edge of your boundary conditions), they are typically very concentrated (e.g. to a point where the point load is, or edge of the end of a constraint) and a third way is that this will give rise to increasing stresses as the mesh is refined.

Singularities in Fusion 360

So how do we deal with singularities and, perhaps more specifically for our interests, what can Fusion 360 do for us? You have several different choices you can make with regards to the singularity and perhaps depending on the time you have to deal with it, what the cause is and the flexibility you have to alter the design with it will influence the path taken. First, I will note that I have brought up the mesh refinement a couple of times. This useful tool in Fusion 360 can help clue you in to a possible singularity, as the singularity is likely to be the location with the highest stress in the model, therefore the region that gets refined and, ultimately, reviewing the convergence plot will show you that the stress is not leveling out. As to dealing with a singularity, let us consider a couple possible common approaches and how Fusion helps. Imagine you have a rectangular plate with a rectangular hole in it and you load the specimen in tension. The four internal corners of the cutout are where your singularities will arise. One way of dealing with it, as is sometimes suggested is to take note, be aware of the cause but then to ignore it. By using Fusion 360's surface probe, you can see that the stress will drop off quickly as you move away from the location of the singularity. In a simple example model I have in front of me, the maximum stress at a center node is 17ksi, at the corner of the element it is 12ksi and by the time I move one element away, it is a little less than 9ksi. So, the stress drops to almost half within the span of moving one element away. So, the probe can be used to realize and interrogate this and find some more appropriate stress values a little further from this point, where the stress is more uniform. An alternative to ignoring it is dealing with it, which would involve, if possible, some modification – such as adding a fillet to the previously sharp corner (reference image below), or to make a region to distribute the force instead of making it a point, or adding something like an extension or component to move the boundary condition causing it away from the area of interest. Fortunately, with the Model space in Autodesk Fusion 360 or the Simplify tools, it is very easy to make some geometry modifications.



Singularity Left Image, Fillet Added on Right. Stress Reduces to 26ksi from 32ksi

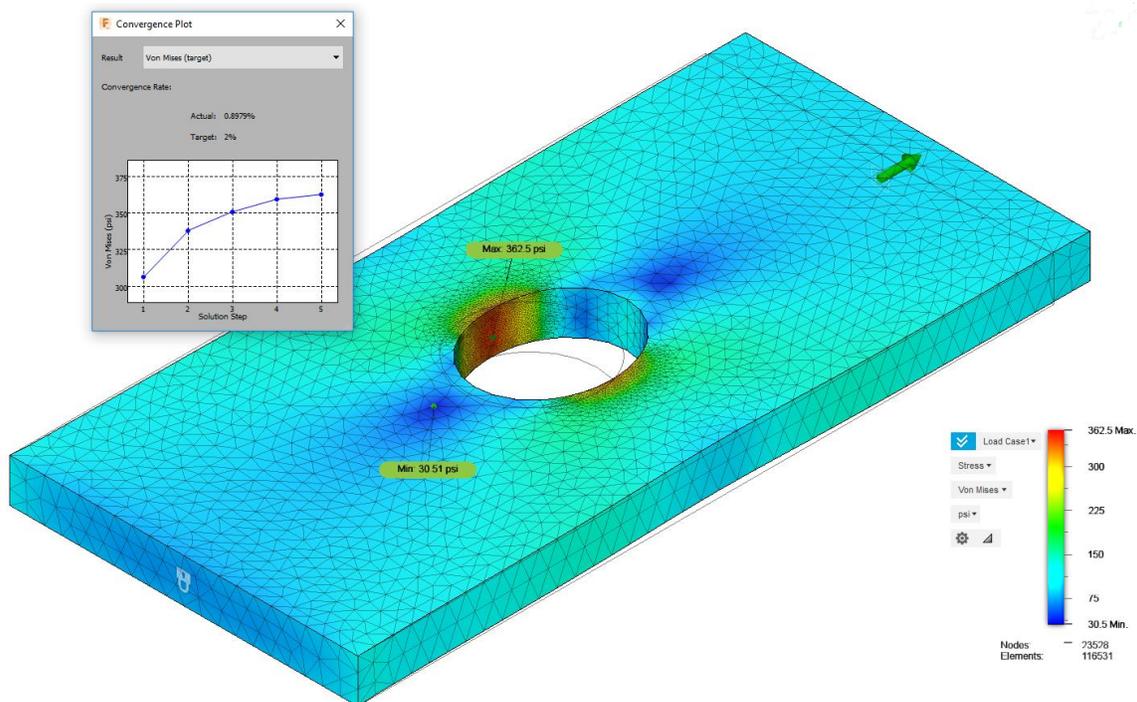
Stress concentrations

Finite element analysis has been employed to help solve geometry with stress concentrations for a number of years now. Historically, this could have been an expensive endeavor from the

standpoint of time and computer resources. Autodesk Fusion 360 has some useful functionality when it comes to this sort of solution.

Designs with stress concentrations

Initially, if stress concentrations exist in the model, it may look as if you are dealing with behavior like a singularity. That is, you may obtain one stress result and then once you refine the mesh, you find that the stress has gone yet higher. It was, classically, some work to set up and run an analysis to obtain a result, but then to go and make the mesh finer and run the analysis again. With a stress concentration, what you have at this point are two data points which are not necessarily the same, which of course then requires a third simulation to see what is happening with the trend in the stress... is it still increasing or is it beginning to taper off, to converge towards a value. Fusion 360 (a.) has parabolic elements and curved elements which can help us get to the solution faster than linear elements (b.) contains the adaptive mesh refinement which takes a lot of the manual labor of the mesh setup away (c.) can leverage the cloud solve so that the iterations do not consume computing resources and (d.) the converge plot helps give us confirmation about whether the solution is converging or not.



Adaptive Mesh Refinement Solution of Classic Type Stress Concentration. Five Total Iterations Cloud Solved in Under Two Minutes. This Uses Linear Elements. Parabolic Converged on Second Iteration.

This concludes the handout for Understanding and Improving Your Results in Fusion 360 Simulation. While I am certain that it is not an exhaustive list of things that are possible to

review in regard to your results, I hope it goes a long way to helping you consider some different things that you are able to do with the Autodesk Fusion 360 program and that you learned some useful information from it.