MFG 124427

Nonlinear Simulation in Autodesk Fusion 360

Mike Fiedler
Autodesk

Lee Taylor
Autodesk

Learning Objectives

- Gain an understanding of the limitations to linear static stress, and know when to employ nonlinear analysis.
- Learn how to describe the 3 primary types of nonlinearity that would lead you to perform a nonlinear versus a static stress type of analysis.
- Become familiar with the advanced material models that are available for nonlinear analysis.
- Discover what actions need to be taken in order to set up your own nonlinear analysis within Fusion 360.

Description

This course will focus on the nonlinear study types within the simulation capabilities of Autodesk Fusion 360. Simulation should be an integral part of the design process to ensure that the design will perform as expected. Knowing when and how to leverage the nonlinear study types will help build your simulation skills. To answer the question of “when”, we will consider the limitations of static stress and how Nonlinear Static Stress and Event Simulation go beyond those limitations. In order to answer the question of “how”, we will examine the advanced material models (such as elasto-plastic and Mooney-Rivlin) and the options that we can and should utilize within the interface.

Speaker(s)

Mike Fiedler

As an enterprise simulation specialist at Autodesk, Inc., Mike Fiedler helps to provide proactive and reactive support in the area of simulation to Autodesk's Enterprise users. Mike obtained his bachelor's degree in mechanical engineering, and he worked with locomotives, steam turbines, and sheet metal hydroforming prior to getting involved with finite element analysis (FEA). He has been helping FEA software users via technical support, training, and web content since 1999, and he has been with Autodesk since 2009.

Lee Taylor

Lee Taylor is a Distinguished Research Engineer at Autodesk. Dr. Taylor is the author of numerous explicit dynamics finite element codes and has 30 years of experience in finite element analysis (FEA) development. He is the author of Autodesk's explicit dynamics software (formerly the NEi Explicit FEA product). He was the original principal developer of the ABAQUS/Explicit FEA product as well as the original principal investigator for Sandia National Laboratories ASCI code development project. Lee received a bachelor's degree from the University of Texas, Austin in Civil Engineering. He also holds a master's degree from the University of California, Berkeley, in Structural Engineering and Structural Mechanics, and PhD for the University of Texas, Austin in Aerospace Engineering and Engineering Mechanics.

Objective 1: Gain an understanding of the limitations to linear static stress, and know when to employ nonlinear analysis.

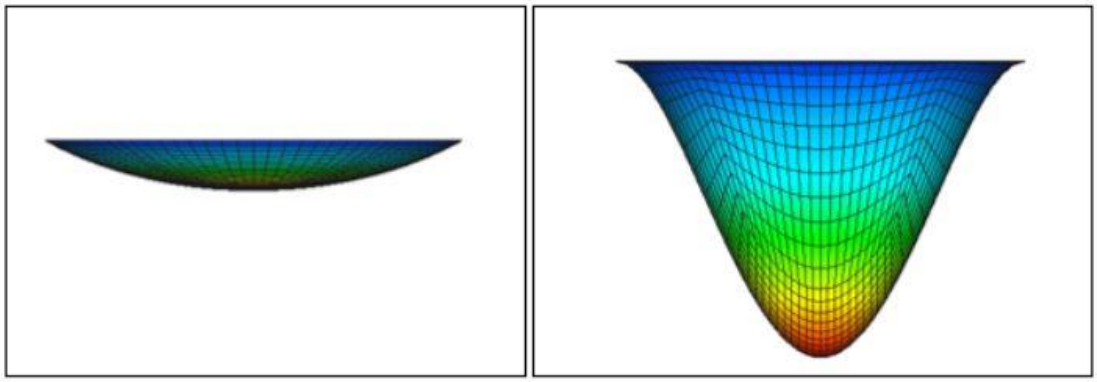
In preparing to gain knowledge about nonlinear static stress, and event simulation analysis types (both of which are capable of performing nonlinear analysis), it might make the most sense to take a moment and understand why we should take the time to study this topic. In the various objects or mechanisms that we might want to simulate or study with finite element analysis, we can often gain a more complete understanding of those studies and more accurate results in some cases. The answer to how this is accomplished is with nonlinear analysis. The answer to what specifically can be done is using the iterative based approach that is employed in these various analysis options and the material models that they contain.

The Limitations of Linear Static Stress

To expand upon what was written above, it is possible that we gain a more complete understanding of those objects we need to analyze, and better results, using nonlinear analysis due to the possibility that our analysis contains nonlinearity. In these cases, if we stick with a linear static stress analysis, these effects will not be properly accounted for. We become better analysts if we know where the limitations of linear static stress are. So, let us consider the following limits;

Deformations should be small in linear static stress

In linear static stress, the deformations should be small. If they are not, this is an indicator that you should likely be leveraging a nonlinear analysis. At this point in the discussion, it makes sense to ensure that all are clear on making the distinction between deformations versus displacement, or – at least I make the distinction when having this discussion on finite element analysis. When an entire object or body moves from its original location (and this could be a linear movement or rotational), then I would consider the study an ideal case for changing to a nonlinear analysis. On the other hand, deformation is a bending, elongation, compression or twist of body or component due to the loads on the geometry. Linear static stress is applicable to solving these deformations, until those deformations become considered large, at which point a nonlinear analysis should be leveraged.

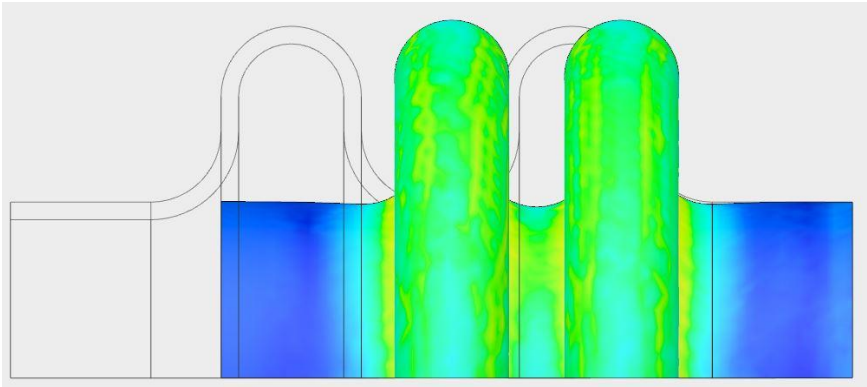


Classic Stress Stiffening (Nonlinear) Result (on left) vs Linear Static Results (on right)
Image Courtesy of the Autodesk Nastran In-CAD 2017 Help

To summarize the point, it would be wise to leverage a nonlinear analysis if there will be displacement (rigid body motion), or large deformation and cases where stress stiffening will come in to play (typically thin geometry with large deformation). It would be reasonable to wonder where the limits are when large deformation is mentioned. Regarding the deformation, you can find a variety of various guidelines on it if you were to search online the topic of large deformation or large deflection. I tend to utilize the 1/10th rule, where if my displacement is 1/10th of the characteristic length of my geometry, I am going to solve it with a nonlinear analysis. Even as those limits are approached, it doesn't hurt, other than some solution time (if you have any doubt about your displacement results) to change it to a nonlinear analysis and test.

Strains and rotations should be small in linear static stress

Large strains come up in different types of analyses in a variety of ways, and is frequently accompanied by a large deformation type of result. If you think of the compression of a rubber bellows or compression of an o-ring, for instance, these analyses typically will have both large strain and large deformation. If the analysis is expected to lead to permanent deformation, this again would likely be a large strain analysis. A linear static stress analysis will likely produce inaccurate results or even fail to converge in these instances. Regarding the small rotations, pay particular attention to when forces may cause twisting in a body or when a part (such as a pin) is free to rotate within a hole.

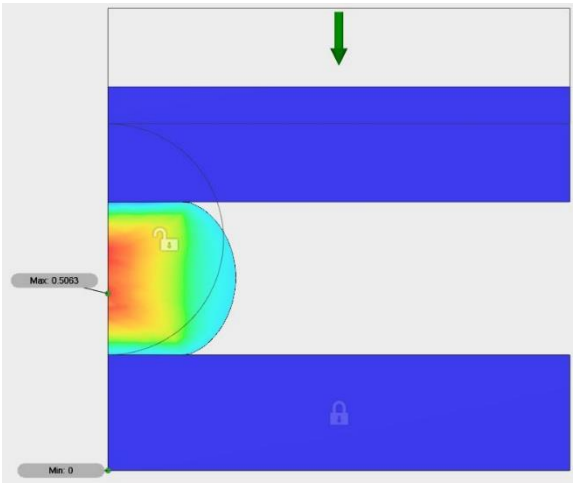


Rubber Bellows Compression Analysis (Strain Max. 14.5%)
Note Un-displaced Shape for Comparison – Actual Displacements

To summarize the point, it would be wise to leverage a nonlinear analysis if there will be large strain or rotation in the model. Some rules of thumb with what constitutes large strain would include strains beyond the elastic limit, or, that which would lead to permanent deformation. If the material you are using (such as a hyper elastic) is capable of large strain, even while maintaining elasticity, a nonlinear analysis is likely necessary in order to appropriately enter the material properties. This will be discussed further, later in this document. Regarding rotations, it would be suggested that if your rotation exceeds any more than about 10 degrees, then it is likely time to start considering a nonlinear solution.

Changes in stiffness throughout the model should be small in linear static stress

A linear static stress analysis is going to presume that the stiffness throughout the model is fairly uniform. This can be considered in a few different ways. The first that we might consider is that the stiffness of the various parts with respect to one another all have relatively similar values. Drastic changes in stiffness (imagine an assembly of steel and soft foam) will likely result in errors or failure of the analysis to complete. Another way to interpret this is how the stiffness of the model changes throughout time. If you were to consider the classic example of a guitar string, you know that by tightening the string, the frequency increases. If you were modeling that in terms of a structural fea, you would be applying more tensile load to the string, which causes the stiffness to change (load stiffening), which is not accounted for linear static stress. A final consideration for this might be abrupt changes in stiffness, such as when the yield of the material is reached or buckling occurs. Neither of those events can be appropriately captured by linear static stress analysis.



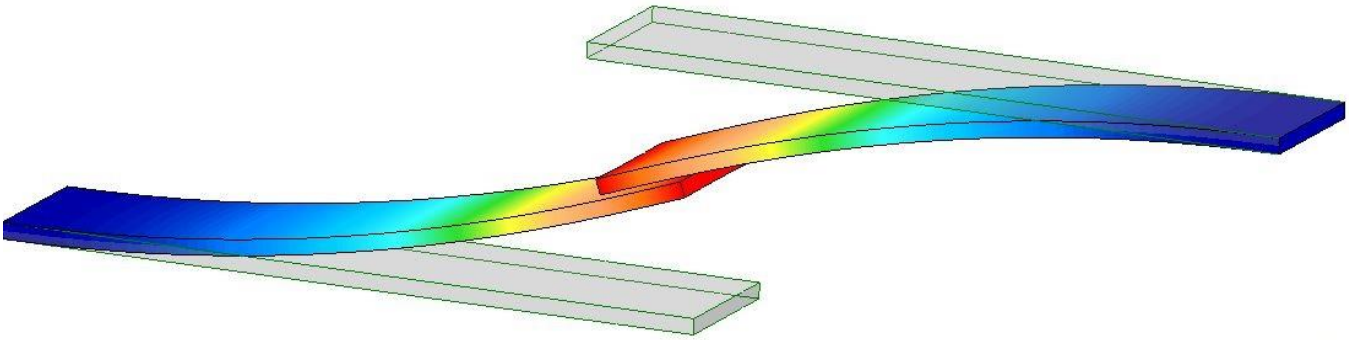
Hyper elastic O-ring Compressed Between Steel Plates
Event Simulation Showing 50% Strain in O-ring

To summarize the point, if you have drastically different material stiffness in the model, it is more likely than not that at least one of those materials (such as the foam I mentioned as an example) needs to be appropriately put in to a FEA program using a nonlinear material – in order to describe that material's behavior under load appropriately. If the stiffness is going to change over time, such as with the guitar string example, all the nonlinear analysis types under consideration for this topic update the stiffness of the geometry over the course of the solution. Finally on this point, combining to two aforementioned points, the appropriate material model with the ability of the analysis type to update the stiffness of the geometry, will allow a nonlinear analysis to be able to simulate up to the yield (and beyond) and approach buckling solutions.

Changes in boundary conditions are small

With the concept of small changes in boundary conditions within linear static stress, it is possible to look at that in terms of 3 different scenarios or applications of the meaning. First of all, if we consider treating that phrase literally, and consider a boundary condition, the boundary condition is unchanging throughout an analysis. When you add some fixity to the model, such as applying a Tx or Ux boundary condition, you are telling

the program to add a high stiffness to that location in the direction specified, such that it does not allow the node to move in that direction. These should be unchanging and that is typically the desired outcome. On the other hand, you might want to have some boundary change over the course of the event. For instance, if I let a straight edge overhang the edge of my desk and push down hard on the end that is on the desk, to make it a cantilever, at the opposite end I can apply some force to deflect the end of the straight edge. Now, I might not readily know what that force is, however, I should be able to easily determine how much I have deformed the free end, plug that in to my analysis and back out the required force. The ability to add some known displacement is done with prescribed (or enforced) displacements, which are found under the Constraint menu in Autodesk Fusion 360. With linear static stress, however, I must keep in mind our earlier limitation that the deformations should be small. So, while it is something that can be leveraged in linear static stress, I must do so within the confines of small deformation. The third way that this may be interpreted is in terms of contact. If I have a part whose edge or boundary is in a position that is dependent upon the contact with a different part, this boundary could change if these parts are in sliding or separation contact, but note that with linear static stress, the amount of movement between parts in contact is expected to be within the distance of an element or less.



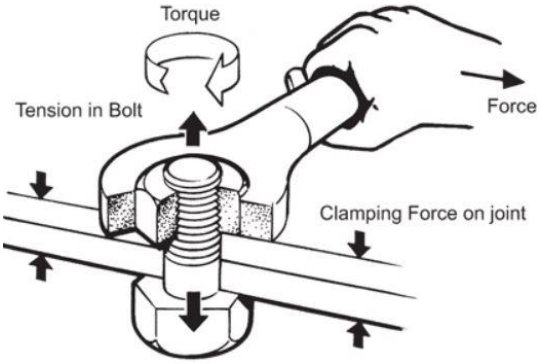
Two Cantilever Beams with Opposing Attracting Forces Applied (Note Shaded Un-Displaced Position)
Nonlinear Contact Allows for the Beams to Settle to Equilibrium Position

In summary of this point, I think that we can agree that there is really no problem with fixed boundary conditions fixing edges or surfaces. Where this has the ability to change though, is in nonlinear, particularly in event simulation, where it would be possible to apply a prescribed displacement that is time dependent, such that the boundary's displacement becomes a function of time. Regarding the second point, as mentioned prior, nonlinear analysis overcomes the limitations of small displacement, so there is also freedom to apply prescribed displacements that impose the displacement over greater distances, even taking the model in to the nonlinear (or permanent deformation) state of the analysis. Lastly, nonlinear analysis, due to its iterative nature, is able to deal with sliding or separation contacts over large distances and so it is suitable for analysis such as snap fit or press fit.

Changes in load direction with deformations are small

Imagine that you have a wrench on a nut with the end of the wrench pointing to the 3 o'clock position (if you envision a clock face overlay) and you are applying torque to tighten the nut. At first your force is direction downward towards the 6 o'clock position and by the time the wrench is at 6, you are likely repositioning your hand and now

changing the direction of the force toward the 9 o'clock position. In short, the force has to change direction. In another example, imagine a flat balloon on a desk and you blow in this. At first, when the balloon is flat, or near flat, most of the pressure is directed vertically or horizontal, as the majority of the surface of the balloon is oriented parallel to the plane of the desk. However, as you continue to blow in to the balloon and it inflates to near final shape, the pressure is now basically oriented radially outward. In both of these examples, the direction of the load (force or pressure) is changing over time. In linear static stress, this does not happen. You apply a force and you are assigning it to a specific vector direction or aligned to some edge, for example, and regardless of how that causes the geometry to deform, the force will always stay in that initial direction. In the case of the pressure, you likely apply it normal to some surface and, again, it stays in the orientation that it was initially applied. If you have an analysis where the load would need to change direction to follow the deformed or displacing geometry, such as inflation, torque, large bending analysis, then using a load that does not change direction is going to limit the accuracy of your deformed results and subsequent other result output types.



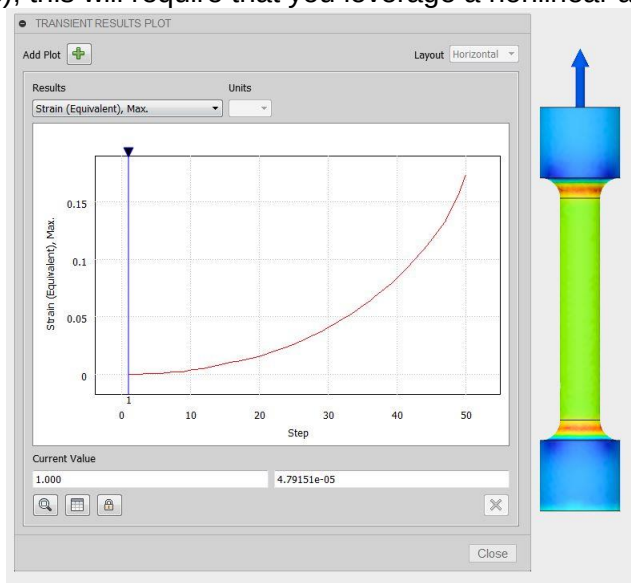
Direction of Force Must Change Over the Course of One Revolution

In summary of this point, it should be noted that the concept of being able to apply a force which follows the deforming geometry (often called a follower force) is entirely achievable in a nonlinear analysis, again thanks to the fact that the analysis is broken out in to a series of steps or iterations, at the time of the writing of this document, it is not something that can be currently applied as a load type in Autodesk Fusion 360, but is on the roadmap for inclusion. That said, in the interim, until this load modifier is added in to the program, I would not necessarily suggest that you completely avoid these types of simulations without first consulting some others with finite element experience or giving some thought to alternative approaches. It is entirely possible that there may be a temporary work around that can be leveraged, such as using a prescribed displacement to push a form or die in to the body, where the shape of the form would apply the loads in different directions, instead of relying on a single user applied force which changes direction over time.

Material remains in the linear elastic range

It may not require much explanation on this point, but we can break this out in to two different considerations – linear and elastic. The first consideration is just as this technically reads and that is that linear static stress is expecting that you remain in the linear elastic range. There is a linear relation between the stress and strain produced in the model and the slope of that line is defined by the material property you select from

the library, or define on your own, via the modulus of elasticity. If you have a material that has no linear region on its stress strain curve (such as some hyper elastics) or a very small linear region which you expect your results will fall beyond (such as some plastic), then the linear static stress analysis is not equipped for you to enter the appropriate material information to describe the behavior of the material in those regions. It is relying, again, on the modulus of elasticity to determine the slope and relation between the stress and the strain. In consideration of the second point of this statement, linear static stress is for analyses that remain in the elastic portion of the stress strain curve. If you anticipate that your model will experience plasticity, or you are performing the analysis with the intent of taking it in to the plastic region (such as with a forming analysis), this will require that you leverage a nonlinear analysis.



Tensile Specimen Analyzed in Fusion 360 Using a Standard Nonlinear Material (Cu – Low Strength)
Note Nonlinear Strain Results While Force Applied Linearly

In summary of this point, nonlinear analysis contains a variety of material models so that you can accommodate entering in the material information for those materials which have no practical linear portion to their stress strain curve, as well as material models that can accommodate the information necessary to describe behavior in the plastic region of the curve.

Objective 2: Learn how to identify the 3 primary types of nonlinearity that lead you to perform a nonlinear analysis

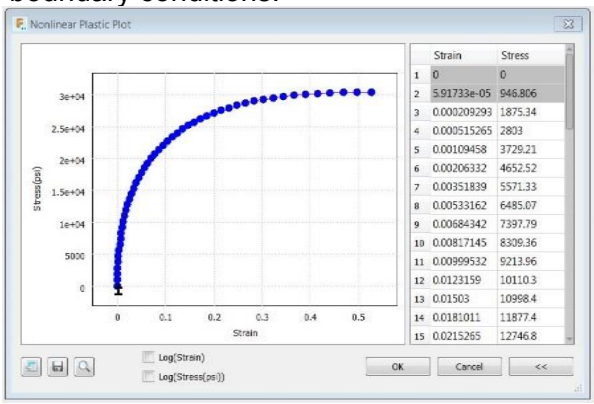
Generally speaking, when discussing the need for nonlinear analysis, there are three broad categories of nonlinear effects. When beginning to set up an analysis, or if during your review of your results you have questions about those results, it can be helpful to think through these nonlinear effects and see if any might apply to the simulation you are solving – if so, then it would be wise to consider utilizing a nonlinear analysis to account for the one or more nonlinearities impacting your solution.

The 3 primary types of nonlinearity

As discussed in the prior pages, there are some limitations to linear static stress. When these limits are reached, very often a nonlinear analysis can be called upon to account for the behavior desired, but unachievable in a static stress analysis. The good news is that the six limitations of linear static stress that we identified in the prior section are all specific situations of the 3 different nonlinearities that we are highlighting here. So, if you have a good understanding of those limitations that were presented, the concepts we are about to present here are just, essentially, the headings under which those particular cases fall.

Material Nonlinearity

Material nonlinearity is summarized in the Fusion 360's documentation on nonlinear static stress analysis theoretical background as such: "One key assumption held by Hooke's Law (F=kx), is that the material stiffness, k, is held constant through the analysis. Depending on the material in question and the degree to which the material is stressed, this can be an inaccurate assumption. In Nonlinear Static Stress studies, the stiffness is updated at every equilibrium iteration. Therefore, determining the global response of the structure becomes an iterative process, requiring the incrementation of applied loads and boundary conditions."



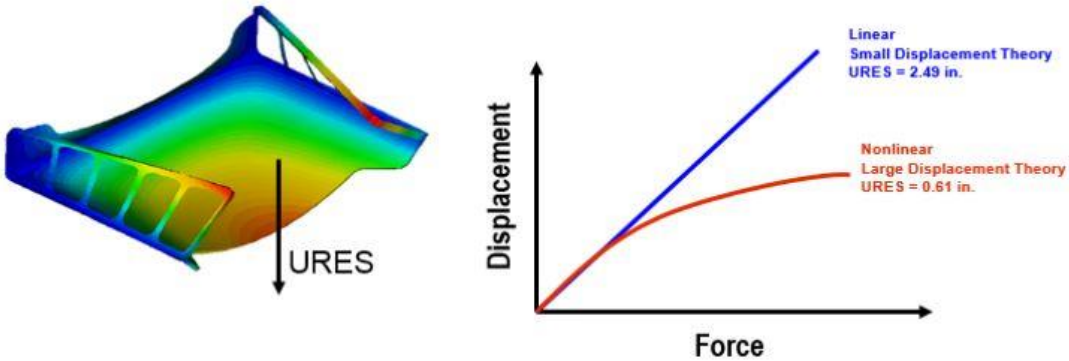
	Strain	Stress
1	0	0
2	5.91733e-05	946.806
3	0.000209293	1875.34
4	0.000515265	2803
5	0.00109458	3729.21
6	0.00206332	4652.52
7	0.00351839	5571.33
8	0.00533162	6485.07
9	0.00684342	7397.79
10	0.00817145	8309.36
11	0.00999532	9213.96
12	0.0123159	10110.3
13	0.01503	10998.4
14	0.0181011	11877.4
15	0.0215265	12746.8

Strain Stress Data Input Screen in Autodesk Fusion 360
Strain and Stress Data Used in Preceding Tensile Specimen Image

In summary of material nonlinearity, the sources of the nonlinearity come down to needing to see what happens in the plastic region when a model is loaded beyond the linear elastic region of the curve, or, when you are utilizing a material that has a highly nonlinear material curve right from the start. Nonlinear analysis has the material models to allow you to accommodate these various strain and stress data input and we will cover those in greater detail in an upcoming section.

Geometric Nonlinearity

Geometric nonlinearity is summarized in the Fusion 360's documentation on nonlinear static stress analysis theoretical background as such: "Hooke's Law also assumes a linear (small displacement) response of the structure to the applied loads. Once again, depending on the material in question and the degree to which the material is strained, this can be an inaccurate assumption. In large geometric deformations, strains become a nonlinear function of the displacement gradients. Any load increment must be solved for the deformed structure, not the original structure."

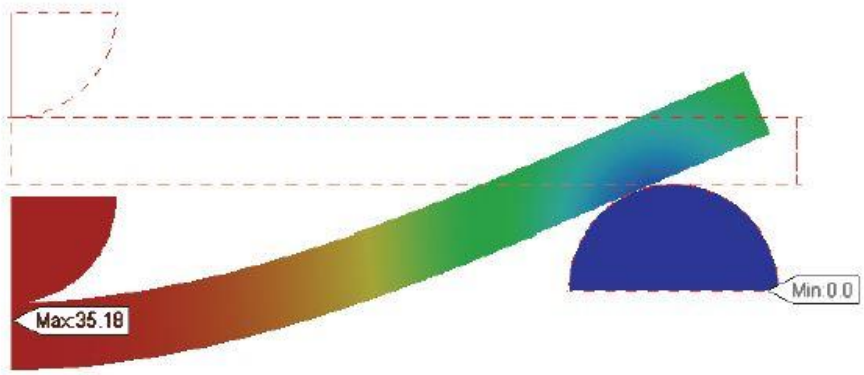


Linear Model Predicts a Proportional Response, Where Nonlinear Shows Stress Stiffening
Image Courtesy of the Autodesk Nastran In-CAD Help

In summary of geometric nonlinearity, we can summarize it using the description that the Autodesk Nastran In-CAD help documentation uses, which is: "The geometric nonlinearity becomes a concern when the part(s) deform such that the small strain assumptions are no longer valid. The large displacement effects are a collection of different nonlinear properties, such as: large deflections, stress stiffening/softening, snap-thru, buckling, [and] large strain".

Force & Boundary Condition Nonlinearity

Force and boundary condition nonlinearity can come from a couple of different sources, but I think can be distilled in to 3 different buckets and those are; are the forces required to follow large displacements, will the boundary conditions change during the course of the event due to something like a prescribed displacement, and/or, will the boundary conditions of some object within the analysis change due to the fact that the current and final boundary are dependent upon contact with another body – and that boundary will change by a relatively large amount over the course of the simulation.



Flexural Test Model (Using Half Symmetry)
The Beam's Contact Boundary Has Changed Significantly Over the Course of the Solution

In summary of the force and boundary condition nonlinearity, keep an eye out for situations where the force would need to change direction in order to appropriately apply the load to the large deformation geometry, situations where you have large displacement prescribed displacements, and sliding contacts that move a distance greater than the length of an element.

Objective 3: Become familiar with the advanced material models that are available for nonlinear analysis.

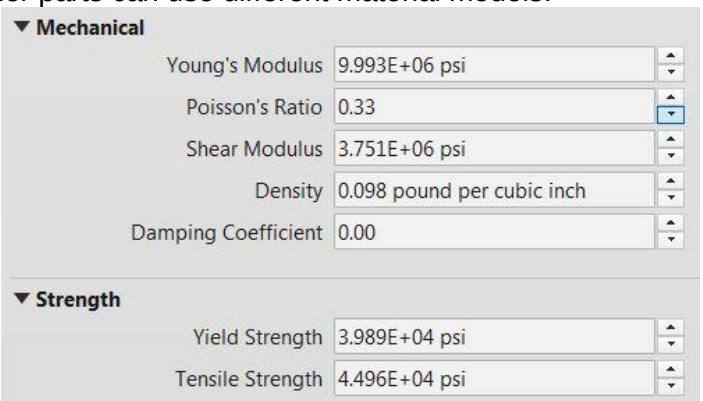
One of the great advantages of utilizing a nonlinear analysis over a linear analysis is the ability to take advantage of a variety of material models. The various material models that are available in nonlinear analysis allow for the ability to input material data for materials that might not behave linearly at all (or have a very small linear portion on the stress strain curve), as well, to take advantage of exploring what happens when an material goes beyond the elastic limit in particular loading situations.

The Nonlinear Materials

There are, at the time this document is being compiled, five different material models that can be leveraged with the nonlinear analysis types in Autodesk Fusion 360. The models are; linear isotropic, hyper elastic (Mooney Rivlin), nonlinear elastic, nonlinear elasto-plastic (bi-linear), and nonlinear plastic. Let's explore these different material models in a little more detail.

Linear Isotropic

The linear isotropic material model is the default material model. Sometimes I have seen people presume that because they have changed from a linear static stress analysis to, for instance, a nonlinear static stress that suddenly, if the model is loaded adequately high, they could see plastic behavior. This is not the case, due to the fact that the linear isotropic model is being used. This is the same material model that is used by linear static stress. It is perfectly fine to use if one or more parts in the simulation will stay within the linear portion of the stress and strain curve. If it is an assembly, other parts can use different material models.



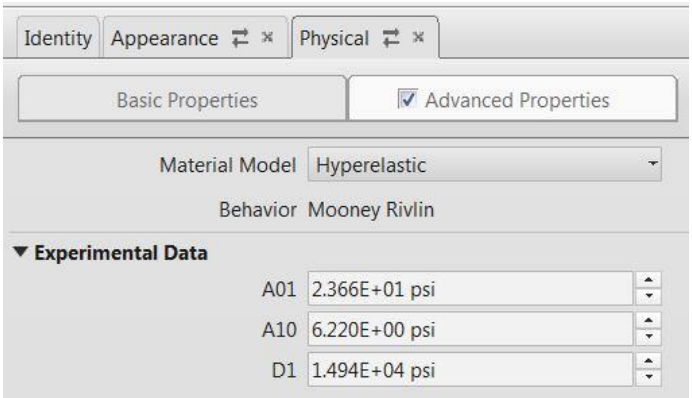
Linear Isotropic Properties of Fusion 360's Aluminum

We can see standard properties here that include Young's modulus, Poisson's ratio, shear modulus, and density. There are properties populated for this material that define the yield strength and tensile strength, but these are only utilized in this material model to calculate the safety factor. Note that there is nothing to describe what the curve looks like beyond the yield, so, if using this material model and you obtain stress or strain values beyond the yield, the values tell you that you have exceeded the yield, but are likely inaccurate and you should

change to a nonlinear material model in order to arrive at more precise calculations. As mentioned prior, if one or more parts are expected to stay in the linear range, it is a perfectly acceptable material model to use for those parts. Further, from time to time, if I have an analysis that is having some difficulty converging and uses one of the complex material models, on occasion I will temporarily switch parts to linear isotropic to see if this analysis runs. This can give insight that the material could be the source of the troublesome analysis and potentially identify bad material data has been input for the more advanced material model.

Hyper elastic (Mooney Rivlin)

The nonlinear stress-strain characteristic of hyper elastic materials, such as rubber, are complex. In Fusion 360, we currently model hyper elastic materials using a 2 constant standard Mooney-Rivlin material model. It should be noted that the hyper elastic material simulation is based on elastic behavior. All materials using this model, like the linear isotropic above, would return to their original shape when unloaded, that is, no permanent deformation occurs.

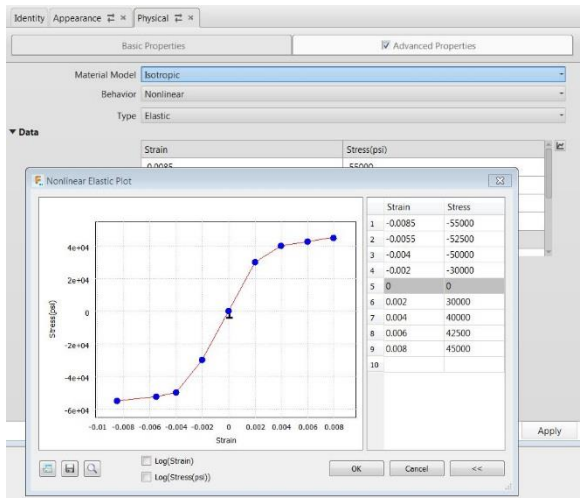


Hyper elastic Mooney Rivlin Data Input Screen

As mentioned above, the hyper elastic material model uses a 2 constant standard Mooney-Rivlin type of input. To characterize a hyper elastic material, typically there are a number of different tests that can be performed (such as uniaxial, equibiaxial, planar and more) to determine the stress and strain values. The values from these various tests can then be curve fit and the material constants determined. A01 is the first constant, A10 the second and D1is a constant related to the volumetric deformation, equal to half of the bulk modulus. Much more detail can be found in the Fusion 360 help files, either by searching the topics on the keyword of hyper elastic, or, go to Simulation->Concepts->Analyses, General->Advanced Topics->Hyper elastic Materials. A link to the Fusion help system will also be posted at the end of this document.

Nonlinear Elastic

Nonlinear elastic materials spring back elastically. Just like the prior two material models that were explored, there is no permanent deformation created when the material goes plastic. The stress vs. strain data is defined in a table and that data should be true stress vs. true strain, with the slope between any two points not being negative.

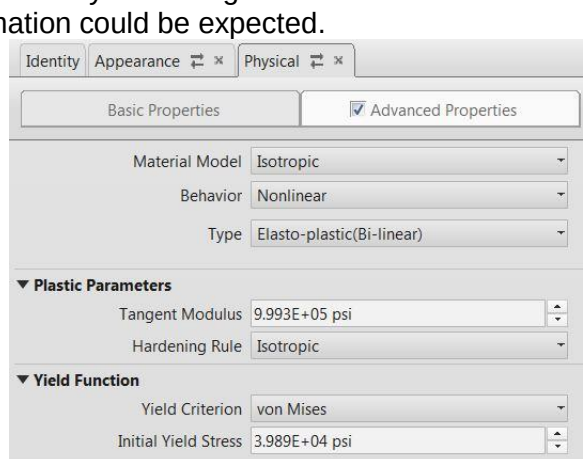


Nonlinear Elastic Data Input Screen

As can be seen in the image above, the nonlinear elastic material model requires that the user provides the stress and strain data, filling in the table in order to define the material's nonlinear elastic behavior. One of the advantages of this material model is that, as also can be seen in the image, different values may be entered for the tensile properties versus the compression properties, which is helpful for materials such as cast iron, with different tension and compression behavior. The values must pass through (0,0). If the values for compression are not given, it will be assumed to be identical to the tension values and the first row has to be (0,0). The values may be imported from a *.csv file if that is easier for the user, using the icon at the far bottom left of the image above.

Nonlinear Elasto-plastic (bi-linear)

Nonlinear elasto-plastic (bi-linear) is the first material model we have examined thus far that is capable of exhibiting plasticity. That is, with the appropriate data and loading, if the model exceeded the yield strength and then were unloaded, residual stress and permanent deformation could be expected.



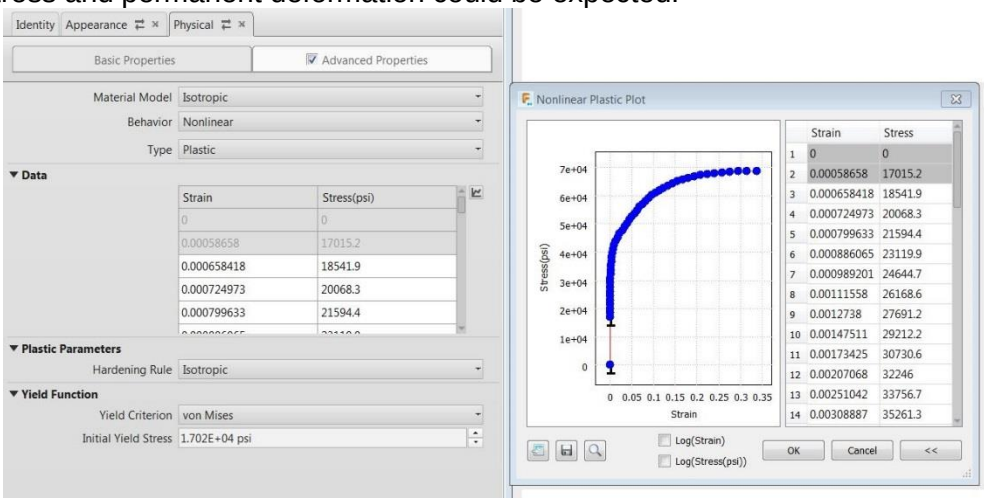
Nonlinear Elastic Data Input Screen

As can be seen in the image above, the nonlinear elasto-plastic material model has a relatively easy to define set of data input. You will specify the tangent modulus of the material. This

value is used post-yield to define the slope of the stress-strain curve in the post-yield range. The material model also requires that the initial yield stress is defined. With this information, the program now has the data it needs to know whether to follow the slope defined by the modulus of elasticity or that defined by the tangent modulus, depending where the resultant values lay. The modulus of elasticity is defined on the Basic Properties tab, as was shown for the input with the linear isotropic material model. Finally, because this is a material model that incorporates plasticity, you can specify the hardening rule, which is set to isotropic by default. In addition, you have kinematic hardening and isotropic + kinematic. Please see the Autodesk Fusion 360 help if you would like additional information on the hardening rules. You can navigate there via Simulation->How-To->Analyses, General->Setup Simulation->Materials->Advanced Material Properties->Define Nonlinear Materials.

Nonlinear Plastic

The nonlinear plastic model is the second (of two) material models we have examined thus far that is capable of exhibiting plasticity. That is, with the appropriate data and loading, if the model exceeded the yield strength and then were unloaded, residual stress and permanent deformation could be expected.



Nonlinear Plastic Data Input Screen

As can be seen in the image above, you will need to enter points in a stress-strain table to define the material behavior at stresses above the initial yield strength. The first two data points are automatically generated based on the zero stress and initial yield-stress points. The additional data points define how the material work hardens as it is strained in tension beyond the initial yield strength. Values with a negative slope should be avoided. Finally, because this is a material model that incorporates plasticity, you can specify the hardening rule, which is set to isotropic by default. In addition, you have kinematic hardening and isotropic + kinematic. Please see the Autodesk Fusion 360 help if you would like additional information on the hardening rules. You can navigate there via Simulation->How-To->Analyses, General->Setup Simulation->Materials->Advanced Material Properties->Define Nonlinear Materials.

Objective 3: Become familiar with the advanced material models that are available for nonlinear analysis. Supplemental information.

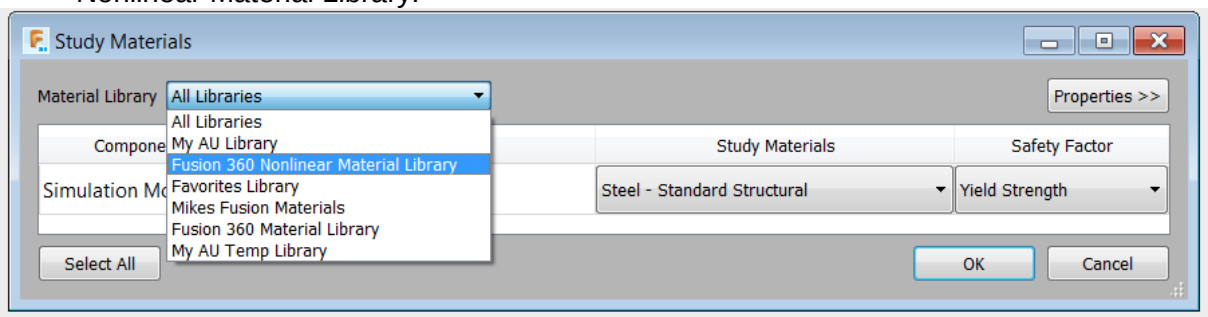
We have spent the prior section discussing the various material models that are currently available with Autodesk Fusion 360. As the application of the material model is a key component of nonlinear analysis, we'll take a bit of the document here to outline how to access the nonlinear materials.

Applying a nonlinear material to your analysis

It was mentioned earlier in this document that the default material model is a linear elastic isotropic material model. If you have explored this far in to the study on nonlinear analysis, it is likely that you currently do or soon expect to be applying this knowledge to an analysis of your own and I would presume you would like to utilize a nonlinear material model. To do so, you can either access one of the pre-defined nonlinear materials, or more likely, you'll need to know how to define your own material and access the material model that you want to utilize.

Pre-defined nonlinear materials

Once you are in the Simulation work space of Autodesk Fusion 360, to define the material for the part, you can go to the Material pull-down menu and choose Study Materials. The top left hand corner of this input window lists the material library that it is allowing you to select materials from, by default this is likely listed as All Libraries. This should be changed, if you want one of the pre-defined nonlinear materials, to Fusion 360 Nonlinear Material Library.



Accessing the Existing Fusion 360 Nonlinear Material Library

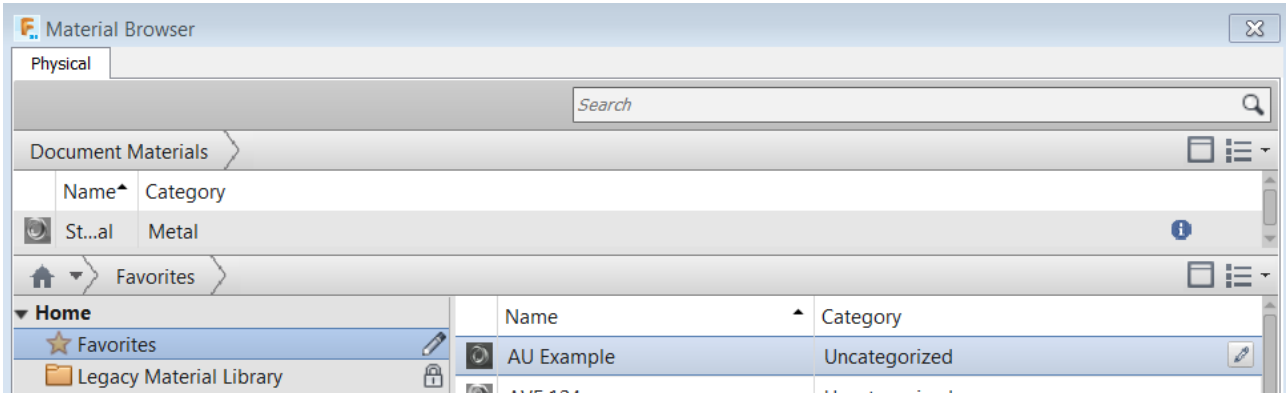
The pre-existing nonlinear material library, at the time of this writing, currently contains approximately a dozen metal materials – varieties of aluminum, copper, steel, and titanium. If you have a different metal or different material, you will need to generate your own material. We will make a brief introduction to that process.

Creating your own nonlinear materials

Autodesk Fusion 360 allows the users to generate custom materials. There are a number of knowledge base articles and videos out there (if you do an internet search) and those might be helpful. My instructions that I am going to give here will be taking us directly to being able to define a nonlinear material, where some of the other online

content may be a little more general information about adding custom materials. It should also be noted, for those attending the course or watching a recording of the AU presentation, this workflow will be demonstrated during the session. But, for reference, here are the quick steps to getting to define your own;

1. I typically like to start in the Simulation environment (or work space).
2. Access the Material pull-down menu from the ribbon.
3. Select Manage Physical Materials, the Material Browser window will open.
4. On the left hand side panel of the Material Browser, if not expanded, expand the Fusion 360 Material Library.
5. I then typically click on the category of Metal from under the Fusion 360 Material Library.
6. You will find a list of existing materials, we need to copy one of these so we can use it as a base to edit/modify it to make it our own.
7. Typically, I'll just take Aluminum from the top of the list, right click on the name of the material and choose "Add to" and add it to the Favorites library so we can edit it there. You cannot edit a material in the Fusion 360 Material Library, but once copied and added to the Favorites, you can edit.
8. Now that it has been copied, click on Favorites from the left hand panel so that we can see what is in that library. We should see the aluminum (or whatever material it was you copied).
9. I will typically, at this point, immediately rename it, so that I know it is my own material. So, right click on the name Aluminum of the material in the Favorites library and choose "Rename" and type a new name in. For this example, I will name my material "AU Example". Once you press enter to accept the name, please note that it will immediately alphabetically sort itself in your favorites list. So, if you have multiple materials there, when you go to edit your new material, make sure to find the appropriate one in the list.
10. Hover over your newly named material. In the center of this Material browser you have two columns; Name and Category. As you hover over your new material, the row with these two columns is highlighted in blue and a pencil icon will show up on the right hand side of the Category column, go ahead and click the pencil icon.

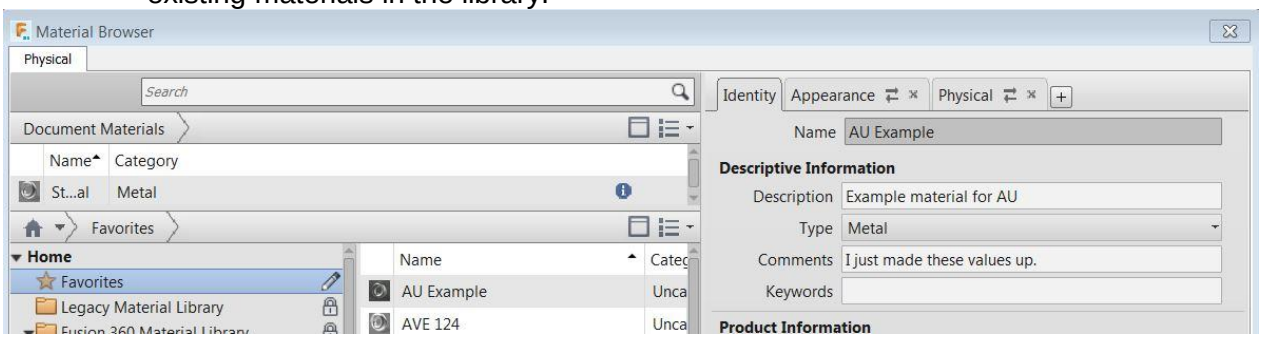


Edit Material

11. Once we click that pencil icon, our Material Browser window will expand and we can begin to enter in the data that we are really trying to get to. The expanded window

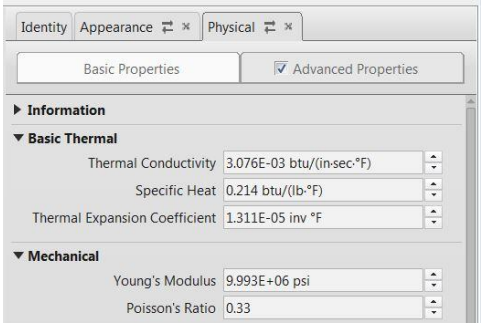
has added a pane to the right and now contains three tabs; Identity, Appearance and Physical.

12. The tab of Identity allows you to add some description and comment, manufacturer information and so on. I typically at least enter in some description and comments to denote that this is my own custom material, so that I can differentiate it from the existing materials in the library.



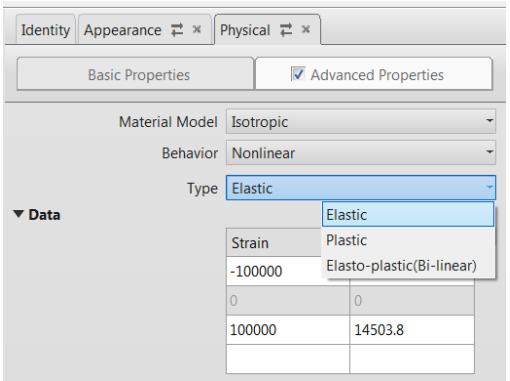
Edit Material

13. You can click the Apply button at the lower right hand corner of the Material Browser window to apply these descriptive changes, but to keep the window open.
14. The tab labeled Appearance is not critical to the material properties for simulation, I typically do not change it.
15. Move over to the tab labeled Physical. This tab has two sub-sections, “Basic Properties” and “Advanced Properties”. The Basic Properties is where it will be by default and includes the basic thermal, mechanical and strength properties. To get to making this a nonlinear material, click the check box on the “Advanced Properties” button and then click on the Advanced Properties bar/panel itself.



Basic and Advanced Properties Tabs
Currently Showing Basic Properties

16. Once on the Advanced Properties tab, you have two sub-options here; Material Model and Behavior. If you select the pull-down menu on Material Model, you can toggle that between Isotropic and Hyper elastic. If you choose Hyper elastic, then the Behavior is automatically set to Mooney Rivlin and you can enter the appropriate material data for your model.
17. If you leave the Material Model set to Isotropic, but change the Behavior setting to Nonlinear, then that gives you access to all the other material models discussed in this document (Elastic, Plastic and Elasto-plastic (Bi-linear).



Access to Nonlinear Material Models

18. Chose the appropriate type and enter in the appropriate information for the type that you have selected. E.g. if you choose elastic, you enter stress and strain data points, if you choose plastic, enter in stress and strain data post yield and identify what that yield stress is.
19. After the data is entered, press the Apply button again to save your information.
20. Last steps I would do, if not done already with other materials, I would suggest – would be to create your own library and copy your material to that library.
21. From the extreme lower left hand corner of the Material Browser window you can select the pull-down menu and choose the option Create New Library.
22. Assign a location and name to the library.
23. Once your new library appears in the list, go back up to Favorites, locate your new material, right mouse click on the name of it and choose “Add to >” and add it to your new library.
24. You can now exit the Material Brower.

Your new material now exists within your new library. Going forward, once you have your geometry in the Simulation work space, when you go the Material pull-down menu and choose Study Materials, when the Study Materials dialogue opens, you can now access your new material library from the pull-down and any new materials you have created should be available. Presuming that the analysis type you have selected for your simulation supports nonlinear materials, then when you choose to use your nonlinear material, the nonlinear properties will be leveraged. On the other hand, if you select to, say, perform a Linear Static Stress analysis, then the linear properties (or Basic Properties) will be utilized.

Objective 4: Discover what actions need to be taken in order to set up your own nonlinear analysis within Fusion 360

In this section of the handout, we will outline the steps that need to be taken to set up a simple analysis in both nonlinear static stress as well as event simulation. The Fusion archive file Bellows v1.f3d you will need to download from the following link; https://autodesk.box.com/s/ks6ps0ffl1hzq4oniaxnjr5getnrq9pi to walk through the exercise on your own. These actions will be presented in brief as we will walk through the entire setup during the live and recorded presentation.

Model Setup Description

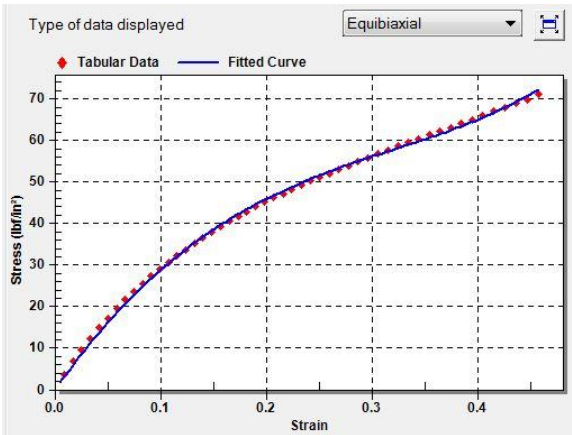
For these setup demonstrations (and those that will be performed in the live and recorded AU session), we have a ¼ symmetry model of a bellows, with metal plates on either end. The bellows will use the hyper elastic Mooney Rivlin material. The steel plate at the bottom of the model (-z direction) will be fixed and bonded. The steel plate at the top of the model (+z direction) will be in a contact (separation) condition with the bellows and a prescribed displacement will be applied to the metal plate at the top end to compress the bellows by ½ inch. The model will utilize an absolute mesh size of 1/16th of an inch (0.0625). We'll review how to set up in both nonlinear static and event simulation and discuss the differences.

Nonlinear Static Stress Example

The steps to perform are as follows;

1. Download the required file (Bellows v1.f3d) to a location on your computer.
2. Open Autodesk Fusion 360.
3. On the Data Panel, you may desire to create a new folder to put the model in to, e.g. clicking the button on the data panel labeled New Folder and naming the folder AU 2017.
4. Once you have the folder structure you want, select the Upload button on the data panel and then browser for and upload your Bellows v1.f3d model. Note that the Upload's "Select Files" button allows you to import this Fusion archive file as well as many other file types from other cad packages.
5. Once you have located and selected your file, press the Upload button in the Upload dialogue window and the file will be loaded in to your cloud repository.
6. The Job Status window will appear showing you the progress of your upload and once it shows a status of Complete, you can press the Close button of the Job Status window.
7. An image of the bellows model will appear shortly after in your Data Panel.
8. Double left mouse button click on Bellows v1 in the data panel and the model will open in Fusion 360.
9. Typically the model will open in the Sculpt work space. This gives you the opportunity to make changes to the model in Sculpt or change it over to the Model work space and make cad changes there.
10. There are no cad model changes that need to be made, so we can go to the pull-down on the ribbon and change to the Simulation work space.

11. As we move in to the Simulation environment, a window appears that will allow us to pick the New Study type. We will select the Nonlinear Static Stress type and press OK.
12. Once the model comes in, we can access the Material Browser to create a new material and define that as a hyper elastic Mooney Rivlin. How to define a new hyper elastic material was outlined in the prior section of this document under the topic of "Creating your own nonlinear materials".
13. Once you have changed the Material Model from Isotropic to Hyper elastic, the program will expect you to enter in the first and second constants, A01 and A10, as well as the volumetric deformation, D1. For this exercise, A01=23.66 psi, A10=6.22 psi and D1=14,940.
14. These constants came from material test data. For a particular rubber material, it was tested to find stress and strain data from equibiaxial testing, pure shear testing and simple tension testing. Those 3 sets of stress and strain data were loaded in to a curve fitting program and the curve fit was generated. The fitting program also produces the constants that can be utilized for the input to the program. The below images show the curve fit of the stress and strain data and the resulting constants. Note that the curve fit routine I used produced output for a 5 constant model and the bulk modulus of elasticity. We then used those first two constants for direct input to Fusion 360 for our sample model here and ½ of the bulk modulus for input of the volumetric deformation.



Curve Fit Results of the Equibiaxial Data

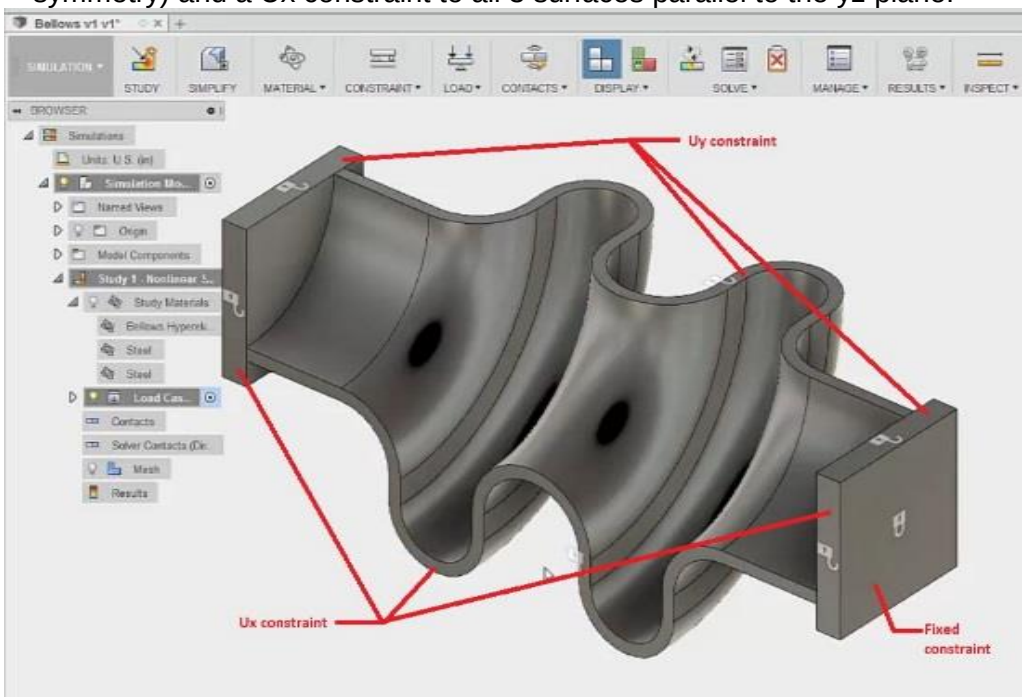
Results of Curve-Fitting Scheme

Material Model Constants

	lbf/in²		lbf/in²
C10	2.36559E+1	K1	29876.051
C01	6.220151E+0	K2	0
C20	-1.987396E-1	K3	0
C11	-9.531732E+0	K4	0
C02	4.27569E+0	K5	0
C30	0	K6	0
C21	0		
C12	0		
C03	0		

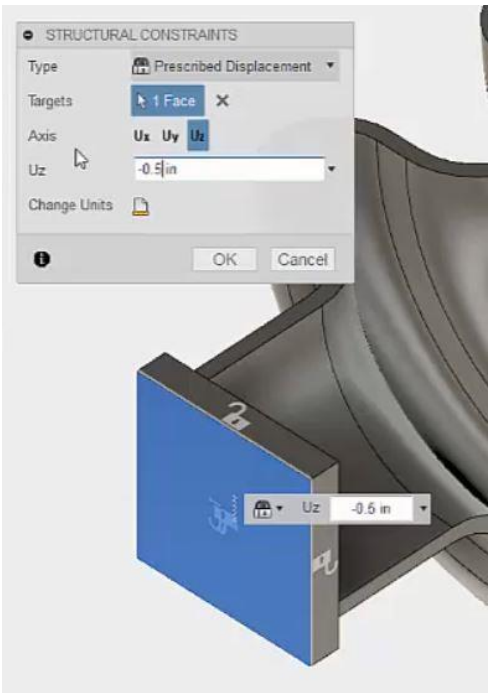
Constants Output

15. Save the new material to your own library (also outlined in the prior section).
16. Once the material has been saved in your library, go to the Material menu and access the Study materials.
17. Leave the two metal parts (Base:1 and Top:1) with the default defined (linear isotropic) steel material.
18. Change the Material Library drop-down menu to access your own material library (where you have just recently saved your new Mooney Rivlin) material and assign that material to the bellows part. In the simulation file, the bellows is identified as component “Simulation Model 1:1”.
19. Next we can add a fixed constraint on the bottom most (-z direction) surface of the metal base plate,
20. Add a Uy constraint to all 3 surfaces that are parallel to the xy plane (to account for symmetry) and a Ux constraint to all 3 surfaces parallel to the yz plane.



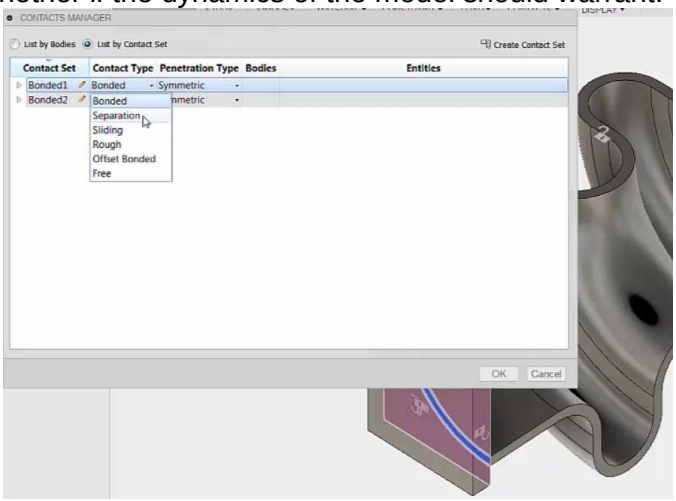
Constraints for NLSS

21. Rotate the model around so that we can view the top most surface of the metal plate in the +z direction and then add a prescribed displacement to that surface. The prescribed displacement is accessed from the Constraint menu, select Structural Constraints and then change the Type from Fixed to Prescribed Displacement. Input a value of -0.5 inch in the Uz direction. Uncheck the Axis toggles for Ux and Uy.



Adding the Prescribed Displacement for NLSS

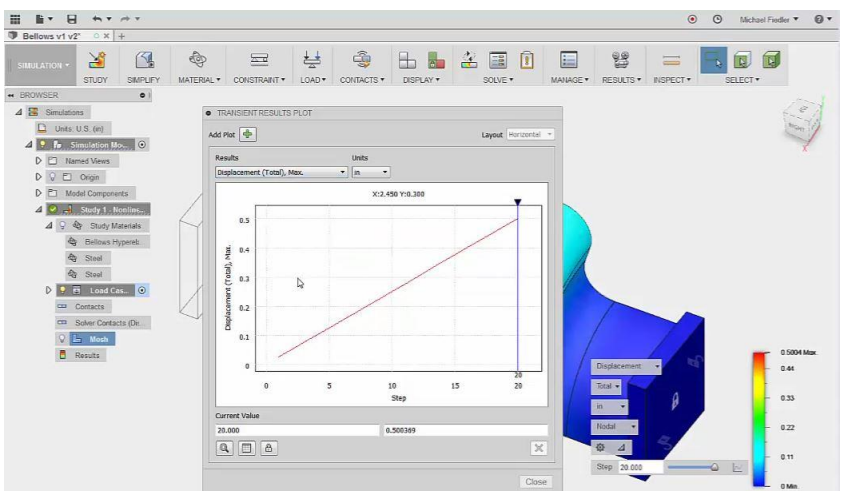
22. Go to Contacts in the panel and then click Generate to have the program automatically generate the contact pairs.
23. Once the contact pairs have been generated, locate the Contacts in the Browser on the left hand side of the canvas and then click to edit by hovering over the Contacts in the browser and then selecting the pencil icon.
24. You should see two bonded contact pairs. One pair is for the contact between the bellows and the bottom plate and the other is for the contact between the bellows and the top plate. Clicking on either row in the Contacts Manager dialogue box will highlight the pair on the model. Find the pair for the top plate and bellows and then change the contact type to Separation, which will allow the parts to slide or separate from one another if the dynamics of the model should warrant. Press OK to close.



Setting the Contact for NLSS

25. Because this is a nonlinear static stress analysis, the solution will be solved in increments or steps. Access the Mange pull-down menu from the ribbon and click on Settings to open the Settings dialogue box.
26. The only change we are going to make here is to increase the number of solution steps from the default of 10 to 20. To help contact models converge, it is recommended that you use at least 10 steps. With models that have material nonlinearity, a range between 20 to 40 should be a fine start. Press ok to close the Settings dialogue.
27. Going back to the Browser, right mouse click on Mesh and access Mesh Settings. In the Mesh Settings dialogue, click the radio button to use an Absolute Size.
28. Type in a value of 0.0625 inches and press OK to close the Mesh Settings.
29. Right mouse click on Mesh in the Browser again and choose Generate Mesh.
30. The model is now ready to solve. In the ribbon bar, click on the Solve button.

Once the analysis has completed, you will be automatically moved to a results view. Note the solve time to get to results depends on many factors, but could take between an hour and two for this analysis. The default of the results display is the safety factor. Use the pull-down menu to review other results, such as the displacement and strains. Note the slider that is available near the legend. Because we chose to solve the model with 20 steps, we can move the slider back and forth and review results over those steps. There is a small graph icon to the right of the slider. If you click that graph like icon, it will open a Transient Results Plot window where you can plot the various results versus solution step. The nonlinear static stress analysis is now complete!



Transient Result Plot for NLSS

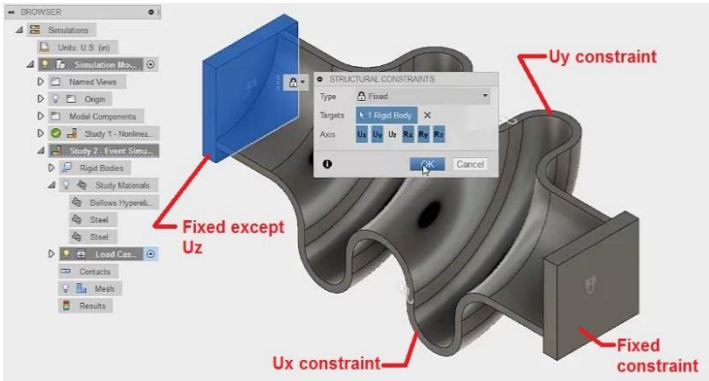
Event Simulation Example

We will pick up with performing the Event Simulation from the end of the nonlinear static stress analysis. The steps to perform are as follows;

1. We will switch from the results display to setup, so access the Display drop down menu in the ribbon and change to Model View from Results View.

2. In the Browser bar on the left, right mouse click on the heading for Study 1 – Nonlinear Static Stress (located just above the Study Materials) and then choose the option for Clone Study. This will duplicate the study, including the geometry, materials, loads and constraints. The study type is also duplicated, we will need to change the study from a nonlinear static stress to an event simulation.
3. Back up in the ribbon bar, choose the Manage pull down menu and access Settings.
4. At the top of the Settings window that opens, you can see the Study Type is set as Nonlinear Static Stress. Use the pull down to change this to Event Simulation.
5. You will obtain a warning about loads and constraints being removed from the model and be asked for confirmation if you want to make the change to the new study type. Choose Yes.
6. At this point, we are going to make no other changes in the Settings screen, you can press the OK button to dismiss it.
7. On the Browser, if you expand the Study Materials, you can see that the materials have carried from the nonlinear static stress analysis to the event simulation, as both analysis types are capable of having nonlinear materials.
8. Immediately above the Study Materials is the option for Rigid Bodies. This option is one of the differences between nonlinear static and event simulation. It would be good to note that the smallest step size that gets used in an event simulation during the explicit solution is influenced by a few different factors (including mesh size and material). We can help speed up the solution if there are parts in our assembly that we don't care to have the program consider for stress calculation and we consider them to be rigid parts. In this simulation, we are really only concerned with the behavior of the bellows part. The steel end plates are not of much interest and could be considered relatively stiff in comparison, so we will make them rigid bodies.
9. Click the edit button on the Rigid Bodies field in the browser.
10. Select one of the steel plates. A message will appear indicating that there are constraints on the part that are not compatible with the rigid body attribute and will be deleted. Press the OK button to acknowledge this. The Rigid Body pop up dialogue should now indicate you have 1 selected Target.
11. Select the other steel plate and again say OK to the message about the constraints to dismiss it.
12. Confirm that you now have 2 selected Targets for rigid bodies and both steel plates should be highlighted in a blue color. Press the OK button on the rigid bodies dialogue to confirm the change.
13. The constraints have been removed now that we have made the steel plates rigid bodies and we will need to reassign them in the event simulation. The reason they were no longer applicable is because, with an elastic body, it might make sense to constrain a particular point or edge or face in a particular direction, however, since a rigid body is rigid, adding any boundary condition for any direction anywhere on the body, constrains that degree of freedom for the entire part.
14. Select the Constraint icon in the ribbon and then select the rigid body at the base (-z direction) of the model and fully constrain the body (Ux Uy Uz Rx Ry Rz) will all be constrained.

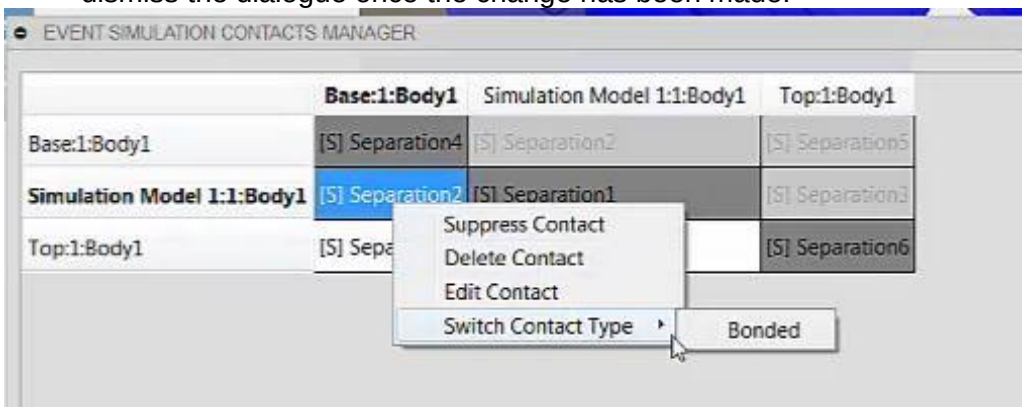
15. Add a Uy constraint to the flat surface of the bellows that is parallel to the XZ plane of the model. This is the symmetry constraint (constraining out of plane translation) that is appropriate for a symmetry face in the XZ plane.
16. Add a Ux constraint to the flat surface of the bellows that is parallel to the YZ plane of the model.
17. On the plate that is at the top of the bellows (+Z) add a fixed constraint to that body, but then uncheck Uz. We will need the z direction of that body free so that it can respond to the prescribed displacement that we are going to apply to that body.



Constraints for Event Simulation Model

18. From the Constraints pull down menu in the ribbon bar, access Prescribed Translation.
19. In the Prescribed Translation menu, uncheck the Components Ux and Uy. These directions are already controlled by the constraints that we added earlier to this top plate, so the only direction we want to specify/control is the Uz direction.
20. Select the top plate.
21. In the Magnitude Uz field, type in the displacement value of -0.5 inches (negative ½ inch in the z direction).
22. Now we come to one of the other big benefits of event simulation. In event simulation, the element of time is added, which is something that is not part of the nonlinear static stress analysis. So, in event simulation, your loads and displacement can be ramped or varied with respect to time, giving you more control over how loads are applied and in what span of time.
23. In the Prescribed Translation pop up menu, select the graph or curve icon to the right of the text of Multiplier curve.
24. You should now see a Multiplier Curve graph as well as input fields for Time and Multiplier on the right hand side. By default, the first row indicates that a time of 0 and a multiplier of 0. So, 0 times our 0.5 displacement at time 0 means no movement at time 0. Then, at row 2, we have a time of 0.001 seconds and a multiplier of 1. The time of 0.001 seconds is the default duration in event simulation, and 1 times the 0.5 inch of displacement means that it will move the full 0.5 inch over that span of time. For our analysis, we are going to make the duration of our event a little longer, so we need to adjust the time here so that the prescribed translation happens over the entire event span.

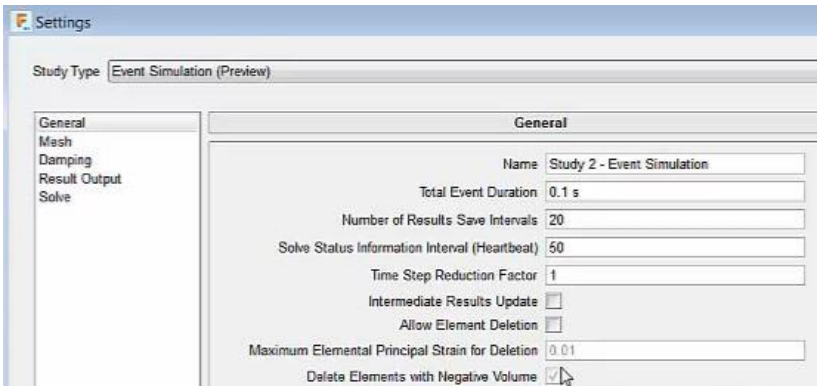
25. Double click in the time field of the second row and change the time from 0.001 to 0.1 second. Press the OK button to dismiss the Multiplier Curve.
26. Press the OK button to accept the setup for the Prescribed Translation window and close it.
27. The next thing we will adjust is the contact. Contacts setup is slightly different between the nonlinear static stress simulation and event simulation, so it is worthwhile to run through that process and confirm it is set up appropriately for our simulation.
28. Press the Global Contacts icon in the ribbon to open the Global Contacts dialogue window.
29. On the Global Contacts dialogue window, press the Generate button to let the program find and generate the contact pairs.
30. Once the pairs are generated, we now will go to the Browser, locate Contacts in the list (below Load Case) and choose the Edit icon on the right of the Contacts.
31. You will then see the Event Simulation Contacts Manager window. This is a matrix of the parts listed vertically down the left hand side and horizontally across the top. Where the parts of interest intersect in the matrix, you can set the specific type of contact between those parts. At the intersection of the Simulation Model 1:1:Body1 and Base:1:Body1 is where we have the contact between the bellows and the base metal plate. All the contacts were set to separation by default, right click on this particular one, choose Switch Contact Type and set it to Bonded. We will leave the others as separation contact. Press the OK button to dismiss the dialogue once the change has been made.



Editing Contact for Event Simulation

32. Locate the Mesh entry in the Browser and edit it. It should still be set to Absolute Size and with a size of 0.0625 inch from our nonlinear static stress run. If it is not, set it to use that absolute size and press the OK button.
33. Right mouse button click on the Mesh entry in the Browser and choose Generate Mesh in order to have the mesh generated.
34. Before we run the analysis, we will make our final changes to the analysis settings. From the ribbon bar, go to the Manage pull down menu and access Settings. Once the Settings window opens, we are going to make 3 changes.
35. Change the Total Event Duration from 0.001 s default to 0.1 s. This will make the displacement happen much more slowly than default. The reasoning behind this will be explained at the end of these analysis setup instructions.

36. Increase the Number of Results Save Intervals from the default of 10 to 20. The analysis should solve fine with the default of 10, but making it 20 will keep that part of the setup consistent with what we requested of the nonlinear static stress analysis.
37. The final change we are going to make on this screen is to uncheck Allow Element Deletion. Event simulation can delete elements during the solution when certain criteria is met (such as when the strain reaches a certain percentage) to simulate parts breaking away or tearing. In this analysis, we don't want that to happen. Press the OK button to close the dialogue.

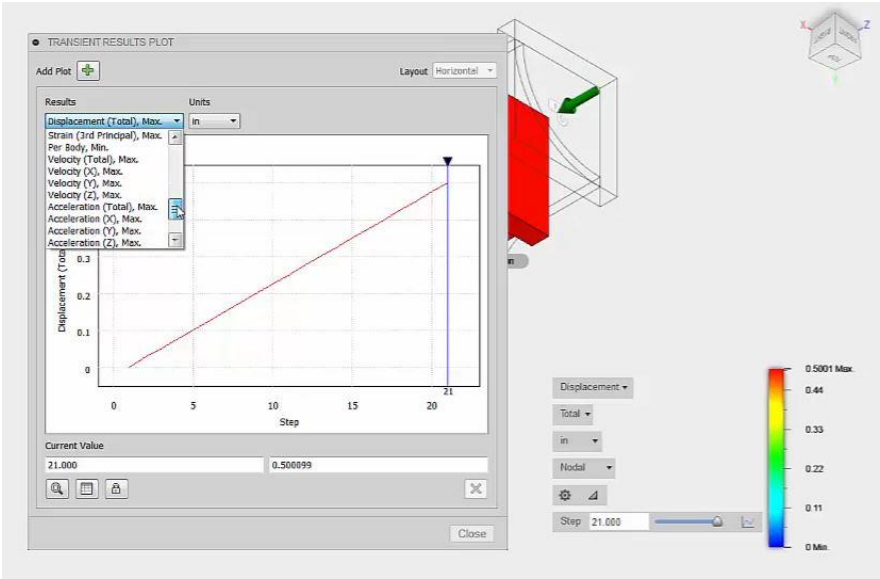


Analysis Settings for the Event Simulation

38. In the ribbon bar, press the Solve button.
39. Once the Solve window opens, ensure that the appropriate study is checked to be solved and then press the Solve 1 Study button to begin the solution of the event simulation analysis. It might be worth noting here, if you have done the nonlinear static stress analysis already, and followed all the instructions to this point, that you can see both studies listed in the solve window. It is certainly possible, if you desire, to set up multiple different studies that can be different analysis types, different materials, different meshes or loads and so forth and then after you have them all set up, then you send them to the cloud together. As the results for any particular study complete, they will be transmitted back to your computer and you can review them as they are received. It is difficult to predict exactly how long the analysis will take to solve as there are a variety of factors including upload and download speeds on your network, how many jobs are in queue on the cloud and the speed of the machine on the cloud. As a ballpark, the test analysis that I have run just now while compiling these instructions has taken approximately 1hr and 45 minutes.

Once the analysis has completed, you will be automatically moved to a results view. Use the pull-down menus to review various results, such as the displacement and strains. Note the slider that is available near the legend. Because we chose to solve the model with 20 steps, we can move the slider back and forth and review results over those steps. There is a small graph icon to the right of the slider. If you click that graph like icon, it will open a Transient Results Plot window where you can plot the various results versus solution step. Because the event simulation contains the element of time, we have additional results types available in the Transient Results Plot (in addition to the displacement, stress and strain we saw with the

nonlinear static stress analysis) you should also see velocity and acceleration results are available. The event simulation analysis is now complete!



Result Plot for Event Simulation

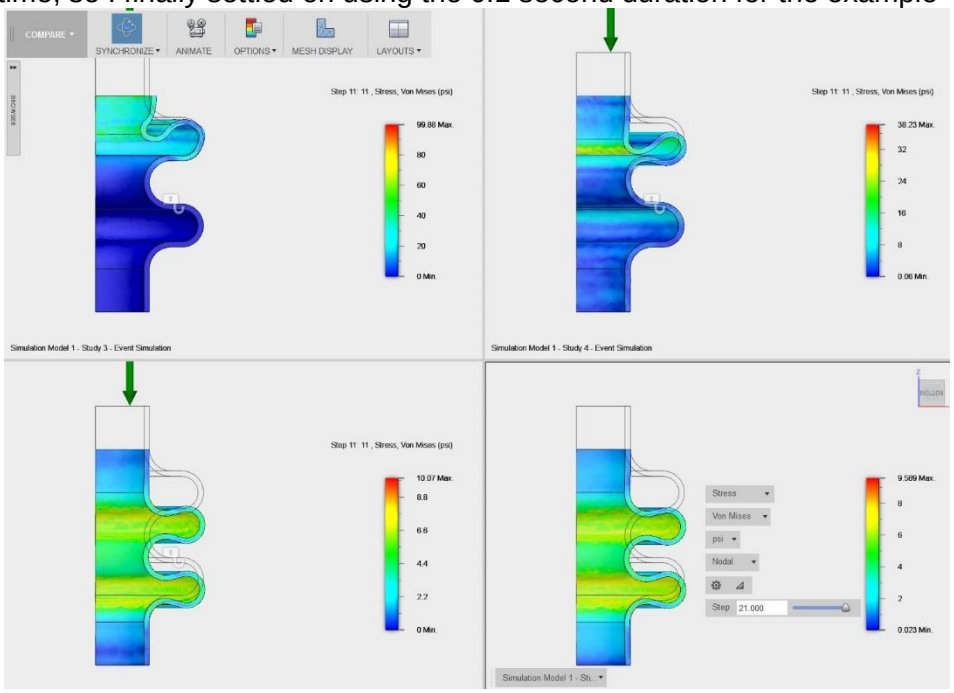
Final Comments on Event Simulation

The event simulation, as we have seen in these exercises – in comparison to nonlinear static stress – contains some additional features, including rigid bodies, the element of time, load curves and element deletion, all of which we have discussed to some degree. I indicated earlier, when the time aspect was introduced, that I would add some additional detail to that. Some comments regarding time;

1. The default duration that is utilized in the program is .001 s. The explicit solver will determine its own critical time step size (increment) based on the mesh size and the material. The time step size is going to be some small time increment of the duration of the analysis. The longer the duration of the analysis, the more time steps required and the longer the simulation solve will be. You can help the analysis by removing unnecessary details so that the mesh size is not too small, the material is what it needs to be (but leverage rigid bodies when possible), and try not to utilize a longer duration than necessary.
2. That being said, you must also consider the dynamics of the problem. When I began to review the bellows model for the exercise here, I reviewed four different analysis durations. Using the compare function in Fusion 360, I created the image below to look at a comparison of the displaced shape of the bellows. Each of the four windows you see there all use the same constraints, displacement, and material, with the only change being the duration of the analysis.
3. In the upper left hand corner, this used a duration of 0.002 seconds. While the analysis is able to solve, the one end of the bellows gets pushed down the ½ of an inch, it happens so rapidly that the lower half of the bellows has not responded to compression that is happening in the top half of the bellows within this snapshot of

time. You can see a pretty severe folding in where the radius is between the collar or neck where the displacement was applied and the first convolute.

4. In the upper right hand corner, I have increased the duration of the analysis to 0.01 seconds and this looks better. We can see the first convolute is compressing, but still, the second (lower convolute) is pretty much in its original form, experiencing very little strain. It would seem this is still not enough time for the displacement to propagate through the bellows.
5. In the lower left hand corner we have the duration of 0.1 seconds, which is the duration that we ultimately utilized for our exercise. You can see here, in comparison to the images from the two other shorter durations that now the compression in the two convolutes of the bellows looks rather fairly balanced.
6. Finally, in the lower right hand corner, I made a fairly large jump in time and set the duration to 0.25 seconds. Even though the duration is more than twice as long as the prior duration, we don't see too much difference in the results for the longer run time, so I finally settled on using the 0.1 second duration for the example



Comparison of ½ in. Displacement Over Various Analysis Durations
From top left, clockwise; 0.002 seconds, 0.01 seconds, 0.25 seconds and 0.1 seconds duration

In conclusion of this discussion, note that time is important in an event simulation analysis. Try to balance not having an unnecessarily long duration with one that is not artificially short and does not represent the real world conditions.

I hope this document proves to offer some help in your getting familiar with Autodesk Fusion 360's nonlinear capabilities and look forward to speaking with you at Autodesk Las Vegas. Don't hesitate to ask questions you might have and if you have the opportunity, I would love to hear your feedback on this handout. At the live session we will hopefully clear up any remaining questions, show demonstrations of these workflows and hopefully can give you a bit of insight in to what is to come with the software as well.

References leveraged:

1. Autodesk Simulation Mechanical – Part 1, available from Ascent. This resource was the source of information on the limitations of linear static stress. https://www.ascented.com/courseware-solutions/autodesk/courseware/simulation/2017/autodesk-simulation-mechanical-2017-part-1
2. Autodesk Nastran In-CAD 2018 help files for various elements of information about nonlinear analysis. http://help.autodesk.com/view/NINCAD/2018/ENU/
3. Autodesk Fusion 360 help files for various elements of information about nonlinear analysis and Fusion specific information. http://help.autodesk.com/view/fusion360/ENU/?guid=GUID-8D78CFF3-8F35-424B-8C46-2D5C887F9993
4. Autodesk Fusion 360 help files direct link to hyperelastic materials; http://help.autodesk.com/view/fusion360/ENU/?guid=GUID-471E915A-FAEC-45F0-8046-9C5EC14A521E