

ES125249

# Autodesk Nastran for Inventor: Unlocking Nonlinear Analysis

Tony Abbey  
FETraining

Mitch Muncy  
Autodesk

## Learning Objectives

- Learn how to spot the three key types of nonlinearities in product performance
- Discover how nonlinearities can impact physical test results
- Discover the terms and processes involved in including nonlinearities in simulation
- Discover nonlinear analysis so that these concepts can be employed in upfront design decision making

## Description

Real-world problems are rarely linear or static. Nonlinearities often impact the prediction of a product's behavior in unpredictable ways. These add uncertainty and risk to decisions made from either virtual or physical test results for engineers and designers who don't understand them. We'll summarize simply and clearly the process of identifying and planning for the 3 key types of nonlinearity in product performance so that you will be able to spot them in both physical as well as virtual testing. We'll teach you how to account for nonlinearities in testing, but the remaining focus will be on nonlinear simulation covering processes, terms, and troubleshooting. While we'll use Simulation Solutions for illustration, the concepts covered will be valid for any simulation software attendees.

## Speakers

Tony Abbey

Tony Abbey has created and taught a wealth of FEA training material over the past 20 years. Thousands of engineers across the world have benefited from his live and e-learning based classes. He has developed a reputation for providing the student with an experience that is full of insight gained from his extensive experience, but which also challenges and motivates. Tony has been working with FEA for nearly 40 years, both in Industry and for leading FEA software providers in the UK and the US. His informal and interactive presentation style allows the key concepts to be taught in a manner which involves participants fully in the course material. Tony presents the popular NAFEMS live e-Learning classes, covering a wide FEA subject area. He writes a series of authoritative articles on FEA for Digital Engineering magazine each month.



Mitch Muncy

Mitch is a Product Manager responsible for the Mechanical FEA strategy at Autodesk. This includes overseeing products such as Autodesk Simulation Mechanical, Autodesk Nastran, and Autodesk Nastran In-CAD, as well as the Mechanical FEA roadmap on Forge.

## Introduction

There are many types of behavior that may be referred to as nonlinear. Some examples of nonlinear behavior include materials that change properties as they are loaded, displacements which cause loads to alter their distribution or magnitude, gaps which may open or close. The degree of nonlinearity may be mild or severe.

In linear static analysis, we assume that displacements and rotations are small, supports do not settle, stress is directly proportional to strain, and loads maintain their original directions as the structure deforms. Most problems can usually be considered linear because they are loaded in their linear elastic, small deflection range. For these types of problems, the slight nonlinearity does not affect the results and the difference between a linear and nonlinear solution is negligible.

While many practical problems can be solved using linear analysis, some or all of its inherent assumptions may not be valid. Adjacent parts may make or break contact with the contact area changing as the loads change. Elastic materials may become plastic, or the material may not have a linear stress-strain relation at any stress level. Part of the structure may lose stiffness because of buckling or material failure. Displacements and rotations may become large enough that equilibrium equations must be written for the deformed rather than the original configuration. Large rotations cause pressure loads to change in direction, and also to change in magnitude if there is a change in area to which they are applied.

## Deciding Between Linear and Nonlinear Analysis

The defining line between linear and nonlinear is gray at best. Traditionally, in finite element analysis, there has been a set of criteria that determines if nonlinear effects are important to a particular model. If any of these criteria are present, a nonlinear analysis is needed to accurately simulate real-world behavior. While this criteria still holds true, new capability such as linear contact and new materials such as composites further blur the line on when it is necessary to carry out a full nonlinear analysis.

Before delving into advanced material types, a look at the traditional nonlinear criteria is an important first step. A nonlinear effect can be broken down into several sub categories:

- Large displacement effects
- Nonlinear materials (plasticity, nonlinear stress-strain curves)
- Nonlinear boundary conditions such as contact or nonlinear springs/dampers

**Large displacement effects** are a collection of different nonlinear properties.

The first type is simply **large deflections**, movements or rotations of a part. For instance, if you expect a part to rotate or deflect 45 degrees, then a nonlinear analysis is required. In fact, any rotation more than about 10 deg will start to have increasing error in a linear analysis. This is because linear analysis assumes small displacement theory in which  $\sin(\theta) \approx (\theta)$ .

The second nonlinear effect is **follower forces**. Follower forces simply mean that the direction of the forces move with the deformations or movement of the part. Pressure loads are a perfect example of follower forces since they always act normal to a surface. As a part deforms, follower forces will adjust the direction of the loads to ensure they stay normal to the surface.

The beam in Figure 1 is loaded with a tip pressure load of 100psi, and three analyses are performed with difference large displacement settings.

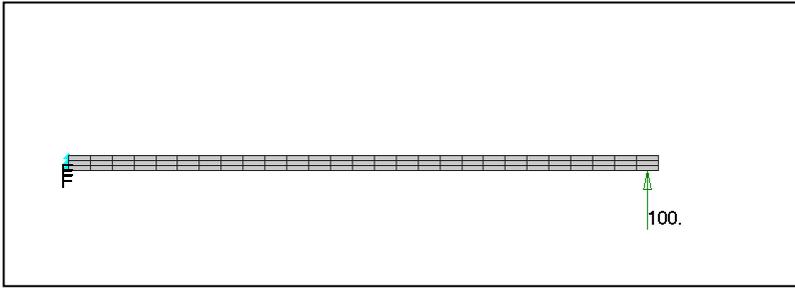


FIGURE 1: CANTILEVERED RECTANGULAR BEAM LOADED WITH 100 PSI PRESSURE LOAD

In Figure 2 you can see the results of the three runs. The first image shows the unrealistic "growth" that occurs when large displacement effects are turned off. The second image shows the results of large displacements turned on, but follower forces turned off. The final image uses large displacement effects with follower forces and is the most accurate.

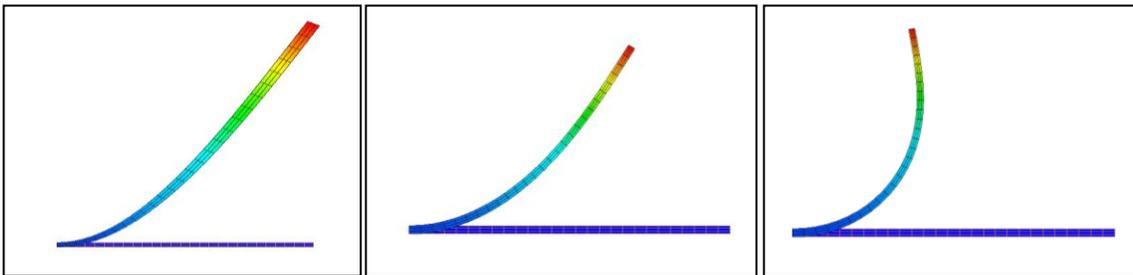


FIGURE 2: DEFLECTION RESULTS USING DIFFERENT NONLINEAR SETTINGS

The third large displacement effect is called **stress stiffening** (sometimes referred to as geometric stiffening). This effect is most pronounced in thin structures where the bending stiffness is very small compared to the axial stiffness. For instance, consider a pre-stiffened drum membrane (Figure 3) subjected to a uniform pressure load. The structure is fixed around the perimeter. This thin walled structure will undergo significant stress stiffening as the part transitions from reacting the load in bending, to reacting the load in-plane.

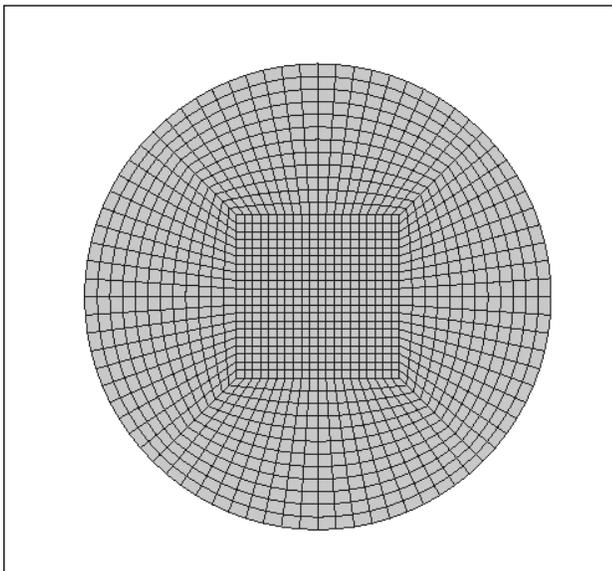


FIGURE 3: PRE-STIFFENED DRUM MEMBRANE SUBJECTED TO A UNIFORM PRESSURE LOAD

Figure 4 shows two results of the pre-stiffened drum membrane. The first image is an actual deflection with large displacement effects turned on (peak deflection is 0.8 inches). The second image is the deformed shape with large displacements turned off. Note in the second image the deformation is scaled down, as the peak deflection is over 5,000 inches.

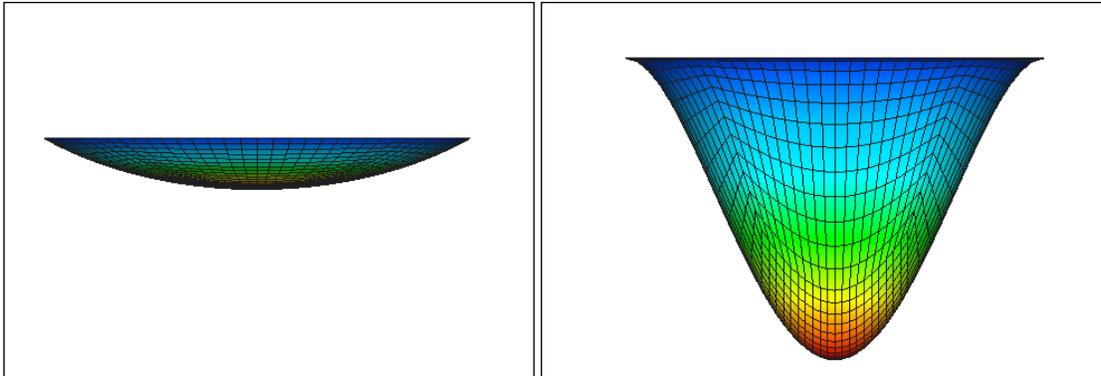


FIGURE 4: DISPLACEMENTS FROM A NONLINEAR ANALYSIS

**Nonlinear material effects** can be important when you want to find out what happens past initial yield of the material. Alternatively, non-metal materials like rubber and plastic can show a highly nonlinear stress strain curve even at low strain values, so getting a more accurate picture of the stiffness of the material through its strain range is important to accurately predicting the stiffness of the overall model. Brittle materials such as cast iron have little inelastic deformation before failure, so a linear analysis approach for these types of materials is generally okay. However, the majority of materials and even metals have some amount of ductility. This ductility allows hot-spots to locally yield thus reducing the stresses compared to what a linear analysis would predict.

The bracket in Figure 5 shows the very different stress distribution between linear and nonlinear materials. The metal has a yield stress of 50ksi. The left image contains the results of a linear material analysis and show peak stresses well above yield. The nonlinear material analysis on the right shows a much different contour due to the stress redistribution. Peak plastic strain was 1% in the nonlinear material analysis.

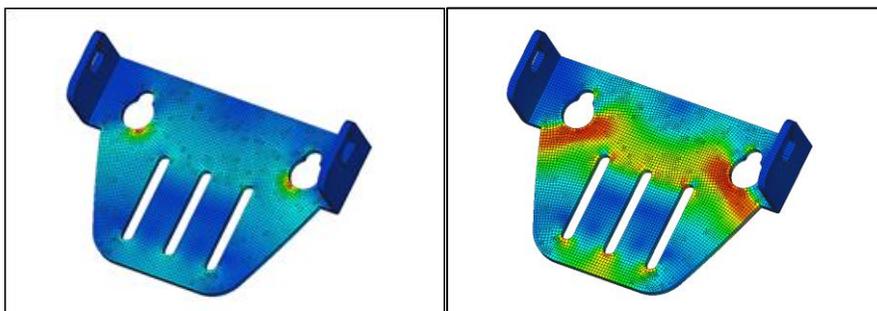


FIGURE 5: STRESS CONTOUR OF A METAL BRACKET WITH AND WITHOUT NONLINEAR MATERIALS

**Boundary conditions** such as surface contact are generally regarded as nonlinear. However, a new trend has emerged lately that allows a contact analysis to run in a linear solution in some FEA applications. In deciding between a linear and nonlinear contact analysis it is best to ask these questions:

1. Are there large movements in the model or any of the other nonlinear effects mentioned above?
2. Is there significant sliding between contact bodies in the model? Is the contact solution path dependent (for instance a snap-fit)?
3. Are detailed contact stresses needed in the model?

If the answer is yes to any of the three questions above, it is generally recommended to run a nonlinear solution to get the best accuracy. The two models in Figure 6 show two examples of when to use linear contact versus nonlinear. The trailer hitch model on the left, while consisting of 6 parts in an assembly, can be run as a linear contact solution since all the parts are initially in contact and the displacements are small. The rivet model on the right, however, needs a nonlinear solution due to the large displacements involved and the need for a nonlinear plastic material model.

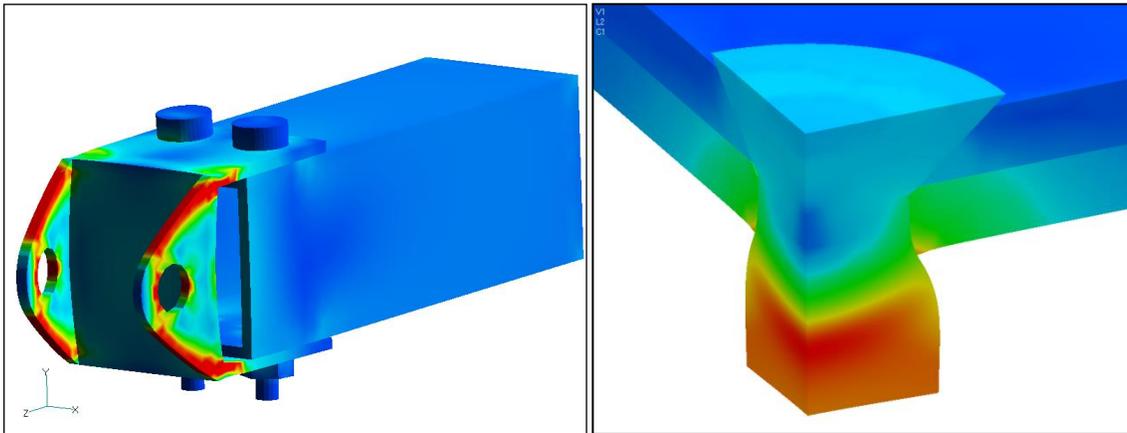


FIGURE 6: TRAILER HITCH WITH LINEAR CONTACT ANALYSIS AND RIVET WITH NONLINEAR CONTACT ANALYSIS

## Nonlinear Static Analysis

### General Guidelines

The following guidelines should be followed when building a nonlinear finite element analysis model:

1. Run the analysis as a linear static solution first and make sure the results are as expected.
2. Keep the model size small. Simplify the geometry as much as possible before meshing (unnecessary fillets, holes, etc. should be removed). Identify areas of symmetry and cut the model at these planes and apply symmetry boundary conditions. Using symmetry will not only reduce the model size considerably, but the symmetry constraints will help to stabilize the model from rigid body movement.
3. Ensure a good quality mesh. The convergence of a nonlinear analysis can be affected by poor quality elements. If the geometry is simple consider using a mapped plate mesh, or hex mesh for solid elements. Perform distortion checks to make sure there are no severely distorted elements.
4. Only apply nonlinear materials in the areas of the model where you expect nonlinear or plastic behavior. This will help to speed up the analysis and can improve the convergence rate.

5. If surface contact is being used, split up the contact areas into specific regions where you expect contact to occur. Using broad or general surfaces will cause a large number of contact elements to be generated, resulting in an increase in the analysis time.

## Troubleshooting

The following steps should be used to diagnose problems when running any nonlinear static analysis:

1. Run your model in a linear static solution. Check that the run completed normally and that the results appear correct. Also, check that the Epsilon value is small ( $< 1.0E-7$ ) and review any warning messages.
2. Setup a nonlinear analysis with large displacement effects turned off. If `PARAM, LGDISP, 1` or `ON` appears in the Model Input File, change this to `OFF`.
3. Turn off any nonlinear materials.
4. If the model still does not run after performing steps 2 and 3, go to the nonlinear analysis settings, set the Number of Increments to 1, Output Control to YES, and Maximum Bisections (under the Advanced button) to 1. You should then get 1 output set of results that you can examine to help diagnose the problem in your model.
5. If you are able to get the model running as detailed in step 2, turn ON large displacement effects and see if the model still converges. If it does, turn ON the nonlinear material and see if the model converges.
6. If, after turning on the nonlinear material your model no longer converges, see Section 5 “Nonlinear Materials” for additional info.

If you are getting an **E5001: NON-POSITIVE DEFINITE DETECTED AT GRID id COMPONENT n**, we recommend you follow the steps listed below to diagnose what is causing the problem:

1. Run your model in a linear static solution and make sure it completes and that the results appear correct.
2. If it runs okay in linear static, run the model as a nonlinear analysis and use the VIS solver. The VIS solver is chosen in the Parameters (on the left side) under Program Control Directives – `DECOMPMETHOD` (double click to change). Also change `MAXSPARSEITER` to 500 iterations to shorten the analysis time. The goal is to force a solution to help diagnose the problem. Unlike all other solvers the VIS will not produce a fatal error for an ill-conditioned stiffness matrix.
3. On the `NLPARM` card, change the number of iterations to 1 (field 3) and change the maximum number of iterations to 1 (field 7).
4. Run this analysis and look at the results. Please keep in mind this is only a diagnostic run and should not be used as actual results. If you see an area or node with a large amount of displacement/stress, this is the probable cause of the fatal error. This may be caused by badly distorted elements.

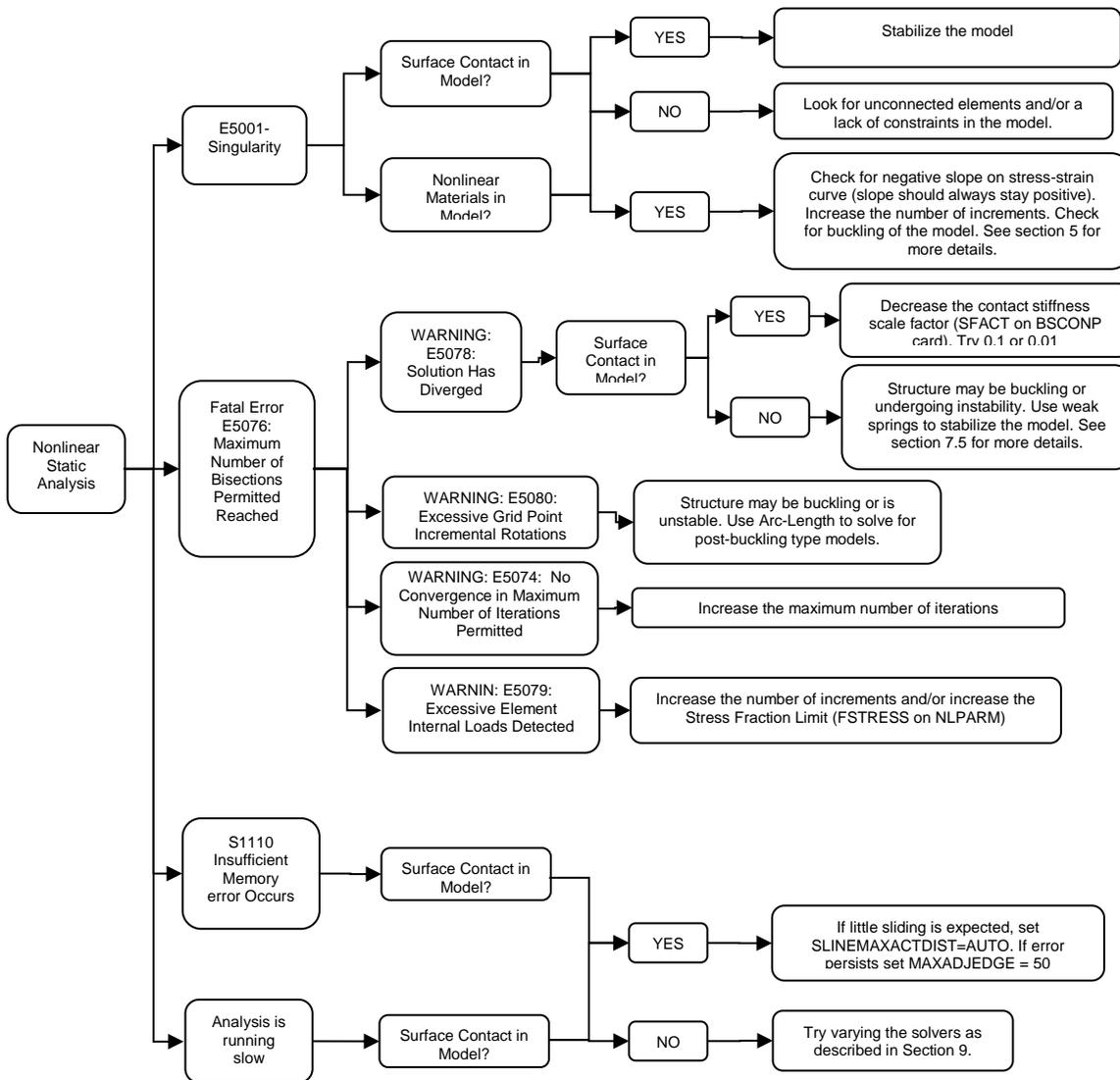


FIGURE 7: TROUBLESHOOTING

## Nonlinear Transient Analysis

If the effects of inertia, damping, and transient loading are significant, then a nonlinear transient analysis should be used. Additionally, “quasi-static” models that undergo buckling or other instable loading conditions will often converge better in a nonlinear transient analysis due to the inertia effects keeping the model stable.

An important element to having a stable NLT (nonlinear transient) solution is to provide damping in the model. There are two types of damping that can be applied in NLT solutions. The first type is a global damping value specified using a PARAM,G followed by a PARAM,W3 which defines the frequency at which to apply the damping. The second type is material based damping and is defined on each material card directly. PARAM,W4 is needed to define the frequency at which to apply the material based damping. Note that the units of W3 and W4 are radians per unit time. The increased flexibility of material based damping (i.e., different damping values can be applied to different areas/materials of the model) makes it the logical choice for NLT analysis.

A note of caution when using damping in a NLT solution is that for models where the velocity/inertia is the main driver of the analysis such as in an impact solution, damping can have a significant effect on the acceleration/velocity/displacement of the model. This is because the solver cannot make a distinction between rigid body motion/velocity, and flexible motion/velocity, so the damping is applied to any part of the structure that has a velocity. For impact analysis it is recommended to use no damping or a small “stability” damping value (i.e., 1.E-6).

## Impact Analysis

There are a few guidelines to follow when performing an impact analysis that will have a large impact on solution time and quality of the results.

### Understanding the Normal Modes of the Structure

This is a very important and often overlooked stage. We need to know the linear response characteristics of the structure to get some idea of what the actual nonlinear frequencies and mode shapes are going to be. It can never be an exact representation, but it gets us in the right ballpark for several key input parameters:

- Frequency range of interest
- Size of time step
- Duration of analysis

Constrain (fully fixed) the area of the model that you expect to make contact with the ground (or other impactor) and run a normal modes solution with ~20 modes. Look at the mode shapes and find the mode you would consider to be the “dominate” response of the structure during/after impact. A look at the modal effective mass table in the \*.OUT file may also help determine the critical mode. The frequency of the mode can be used to calculate the key input parameters above:

- Frequency range of interest – This would be the frequency of the dominant mode.
- Size of time step – This can be calculated using  $1/f$ , and then assuming 100 data points per cycle would net:  $dt = 1 / (100*f)$ .
- Duration of analysis – This largely depends upon the velocity of the impact, the size of the model, and the flexibility of the model. A good estimate is to run the analysis for 2-5 cycles.

### Positioning of the Model

In most situations it is best to perform a hand calculation to find the velocity at impact and then start the two models near each other. This approach will net shorter analysis times, and better fidelity than starting the two bodies at a physical distance (i.e., as in a drop test). A good method for calculating the small separation distance is to use the equation,  $d = v * (2*dt)$ , where  $d$  = separation distance,  $v$  = velocity, and  $dt$  = time increment. This separation distance will allow for the solution of 2 time steps before impact.

### Multiple Subcases

When pre/post-impact behavior is desired, using multiple subcases is a good way to fine-tune the analysis such that detailed time stepping can be used during impact, and a much coarser time-step can be used after impact.

## Nonlinear Materials

### General Guidelines

The following guidelines should be followed when building a nonlinear finite element analysis model with nonlinear materials:

1. When performing an analysis that will have large amounts of strain it is recommended to use a nonlinear elastic material.
2. Check that the slope of your stress strain curve (in the linear region) matches the Young's Modulus of the material.
3. Stress-strain data should be taken from a true stress-strain curve, not an engineering stress-strain curve. A true stress-strain curve takes into account the "necking" or reduction of cross-sectional area during a standard axial pull test. If the slope of the stress-strain curve is very "flat" (or negative) you may get a non-positive definite error during the analysis. If acceptable, remove the flat area on the stress-strain curve and re-run the analysis. If you must use a flat or negative slope stress-strain curve you can try and force a solution by turning SOLUTIONERROR=ON and FACTDIAG=0. Also, try setting NLMATSFAC to 0.1 – 0.5.
4. When using a nonlinear plastic material, it is recommended to use a stress-strain curve instead of the Bi-Linear method. The Bi-Linear method essentially creates a stress-strain curve with two different slopes. This abrupt change in the elastic modulus makes it more difficult to converge on a solution.

### Stress-Strain Curves

When post-processing results of a nonlinear material model, it is important to understand how the material yields on an element basis. Internal to each element are Gauss points where stresses/strains are calculated. For a nonlinear material, Autodesk Nastran does a table lookup on the stress-strain curve for each Gauss point stress, then it reports an average of all Gauss points (which is the center stress) and extrapolates the Gauss point values to the corners.

If an element undergoes a pure axial load, all Gauss points will yield at the same time, so the center stress will match the input stress-strain curve exactly. In the bending case, the outer Gauss points yield first then the inner points. This means the average center stress may not match the input stress-strain curve exactly during abrupt changes of slope on the input curve. This is because some Gauss points are on one point of the curve and the others are on another. If there is a large difference between the input stress-strain curve and the output stress results, a refined mesh is recommended.

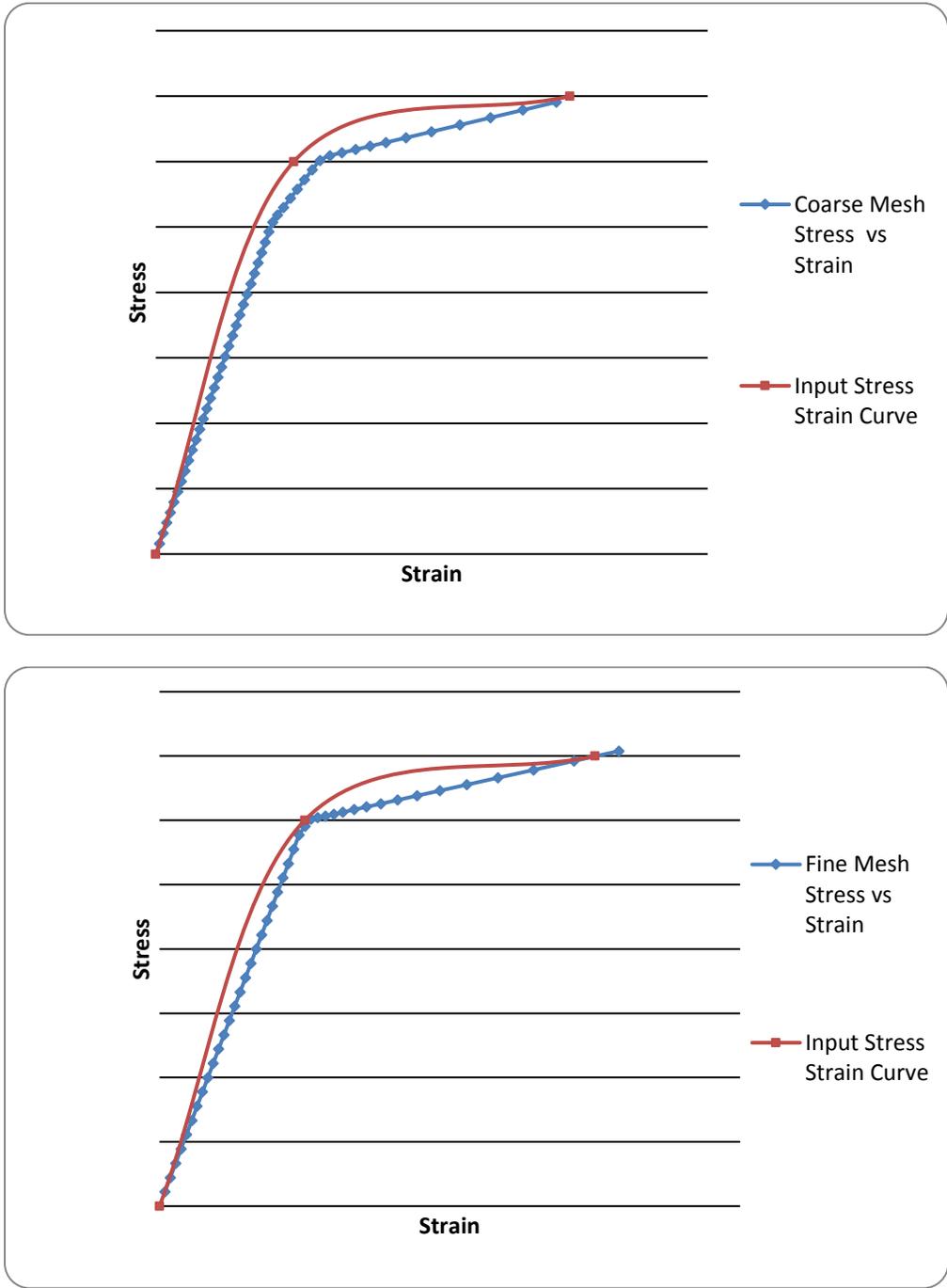


FIGURE 8: STRESS-STRAIN CURVES FOR TWO DIFFERENT MESH DENSITIES

It is also important to note that since the corner stresses are extrapolated from the Gauss points, their stress values can often exceed the stress values on the stress-strain curve. If there is a large difference between the center stress values and the corner stresses, it is

recommended to refine the mesh to get a more accurate representation of the yielding that is occurring.

## Surface-to-Surface Contact

### Types of Surface Contact

**Separation** - This is true surface to surface contact. It will include friction effects. Parts are allowed to slide relative to one another and separate. This is available in a nonlinear analysis. In a linear static analysis, it uses the linear contact solution. This will iteratively solve the contact region so that only areas which are making contact are being used. The areas trying to lift off are ignored.

**Bonded** - This essentially glues the parts together. They will not be allowed to slide relative to one another and will not separate. This is available in linear and nonlinear analysis.

**Sliding / No Separation** - This will allow for sliding along the contact face, but the parts will not be allowed to separate. This is available for linear and nonlinear solutions. This works best for planar surfaces since curved surfaces require a little bit of lifting to accomplish sliding relative to one another (due to faceting of the mesh).

**Separation / No Sliding** - This will allow parts to separate, but not slide relative to one another. This is essentially infinite friction. This is only available in a nonlinear solution.

**Offset Bonded** - Like welded contact type, it is available in linear and nonlinear solutions. It will also glue the parts together. This contact type works better for large gaps between the parts. This contact type also works for edge to surface contact, typical of what you would see with shell models or midsurface models.

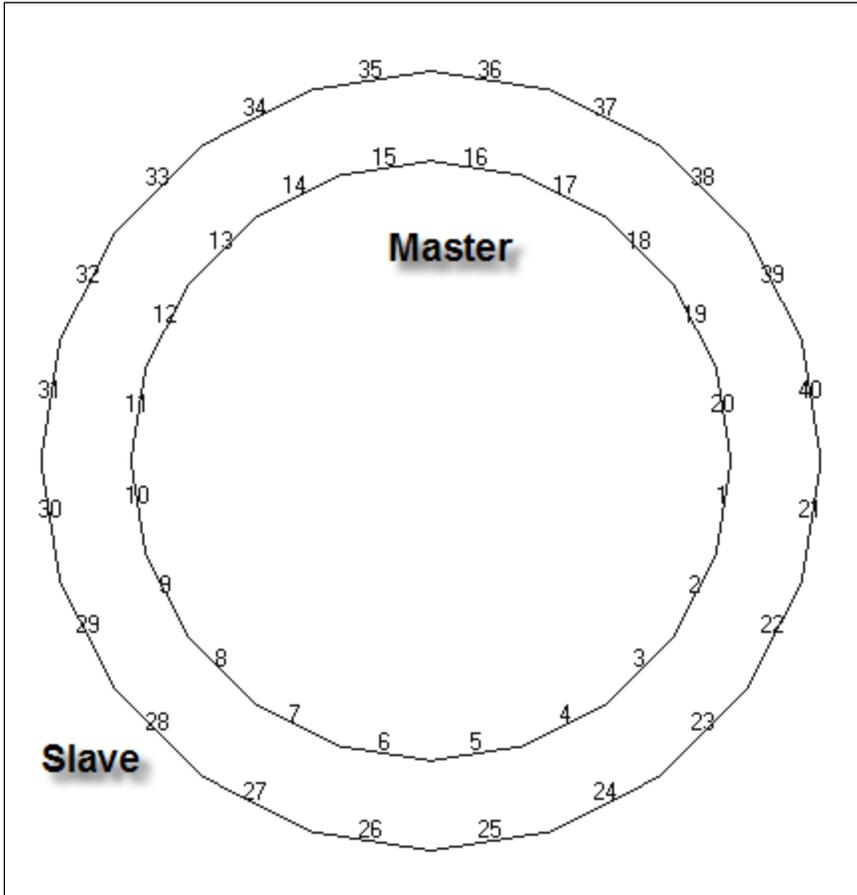
### Basic Tips for Setting up Surface Contact

1. If you are setting up plate elements for contact you should first check that the plate element normals are pointing towards each other.
2. Define your contact segments by either defining them by surfaces or by element faces. It is always recommended to use groups when setting up contact, as it will be much easier to view/select your contact segments.
3. With the contact segments defined, uncheck all other element types in the pre- and post-processor. This will allow you to see only the contact segments. Make sure the contact segments are defined as intended.
4. Next, define the contact property. Fill in any friction values if you expect sliding and friction is important.

5. Define the contact pair. In general, choose the master as the contact segment that has the least amount of curvature, or if it is cylindrical contact choose the master as the outside segment (and use unsymmetrical penetration).
6. Setup the nonlinear analysis settings. In general use 5 increments for contact with no sliding, 10 increments for contact with sliding, and 20 increments for contact with nonlinear materials.
7. First, run your model in a static/modal analysis and make sure it runs fine, and that the results look good. Note that the contact elements will behave as weld elements during a static/modal analysis. If the results are as expected proceed to a nonlinear analysis.

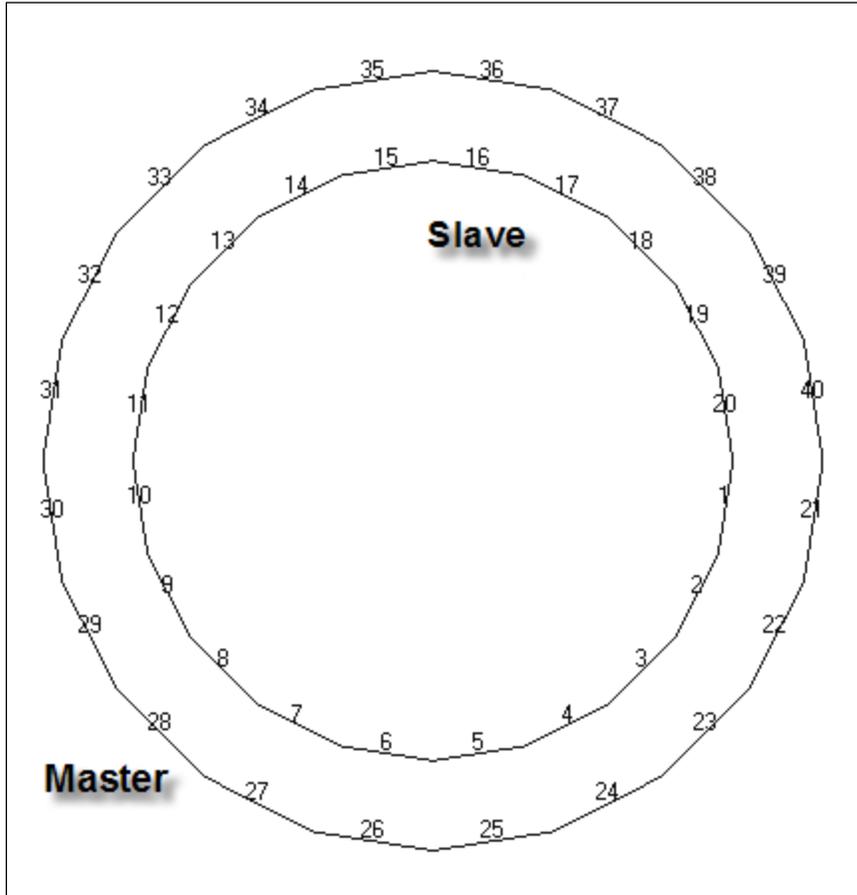
### Surface Contact Normals

When modeling surface contact between concentric cylinders (and other undulating surfaces), care must be taken when choosing which part is the slave and which part is the master. As a general rule the master contact segment should be the outer surface, and the slave the inner surface. In Figure 9, the master surface is incorrectly chosen to be the inner surface. Element 16 is shown as an example of checking the normals. The dotted line represents the surface contact plane for element 16, in which all the slave elements should be positive (above) this plane. You can see that elements 21-33 and 39-40 are below element 16, which will result in a **CHECK NORMALS** warning when run in Autodesk Nastran.



*FIGURE 9: INCORRECT MASTER/SLAVE FOR CONCENTRIC CYLINDERS*

In Figure 10 the correct master-slave setup is shown; and element 36 is shown as an example. A new plane is drawn for element 36. All slave elements should be positive (below) the plane, and we see that indeed all the slave elements are below the contact plane.



*FIGURE 10: CORRECT MASTER/SLAVE FOR CONCENTRIC CYLINDERS*

If little sliding between the two parts is expected, the parameter, SLINEMAXACTDIST (located under Nonlinear Solution Processor Parameters under Parameters) can be set to AUTO. With this setting only nearby elements will be considered for contact, which will not only speed up the solution, but will eliminate the problem of surface normals as in the above example.

### Contact Settings

The explanations below describe the various effects of the contact settings in Autodesk Nastran In-CAD.

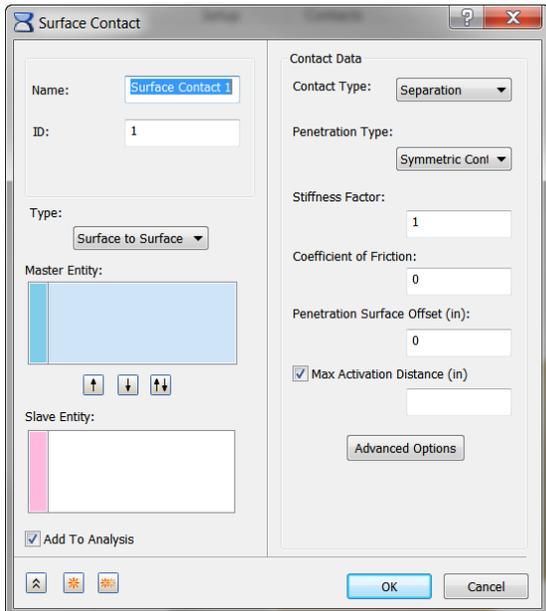


FIGURE 11: AUTODESK NASTRAN IN-CAD CONTACT SETTINGS

**Stiffness Factor** - By default Autodesk Nastran will use the material stiffness of the master entity. Sometimes this can be too stiff and cause convergence problems. Setting the stiffness factor to 0.1 (10%) or 0.01 (1%) will often help with convergence. Be careful though, lowering the stiffness factor too much may cause penetration with the model. Usually 5% penetration (of an element width) is acceptable for a contact model.

**Frictional Stiffness for Stick (Advanced)** – Calculated internally by the solver, so it is recommended to leave this entry blank. A method of choosing a value is to divide the expected frictional strength ( $\mu \times$  expected normal force) by reasonable value of the relative displacement before slip occurs. A large stiffness value may cause poor convergence, while too small a value may result in reduced accuracy. An alternative method is to specify the value of relative displacement.

**Max Activation Distance** - This is the distance for which elements will check to see if they are in contact or not. For models with little or no sliding, setting this to length of sliding will drastically cut down the number of contact elements generated. When no sliding is expected, the AUTO setting is recommended. This will use the element size as the distance. The maximum activation distance can be set for each contact pair here or globally by using the SLINEMAXACTDIST parameter.

**Penetration Surface Offset** - Surface contact is defined using the nodes. So if you are modeling a 2D plate, by default the contact will take place at the centerline. To account for the thickness, you can use this offset.

**Max Allowable Penetration (Advanced)** - There are two methods for adaptive stiffness updates: proximity stiffness based and displacement based. If this is set  $> 0.0$ , the displacement based update method is selected. When this is set  $= 0.0$  (default), the proximity stiffness based update method is selected. The recommended allowable penetration for this

value is between 1% and 10% of the element thickness for plates or the equivalent thickness for other elements that are connected to the contact element.

## Model Stability

Model stability is an important factor in getting a nonlinear static surface contact model running properly. As a general rule, surface contact should never be used to satisfy a stability constraint. To put it another way, the model must be stable even if the surface contact was taken out. Numerically there will generally be a small gap between the contacting surfaces. This gap means (initially) there will be no stiffness transferred between the parts. If one of the parts is unconstrained it will cause a singularity in the solver on the first increment.

Autodesk Nastran has an automated method of stabilizing surface contact models via the parameter CONTACTSTAB. In the Reference Manual it is defined as follows:

When set to ON, will generate stabilization spring stiffness via the model parameters NLKDIAGSET, NLKDIAGAFAC, and NLKDIAGMINAFAC on the contact boundary. The default AUTO setting will automatically detect and stabilize all surface contact in the model with a significant initial gap (i.e., model reference dimension multiplied by 1.0E-04). The stabilization stiffness used can be controlled by specifying a scale factor which is a multiplier to the stabilization stiffness calculated automatically.

In other words, CONTACTSTAB can be set to ON, AUTO, or a real number that specifies a scale factor to the automatically calculated value. If the ON setting is used, NLKDIAGAFAC must be set to a value to provide a stiffness. CONTACTSTAB has many advantages such as the ones listed below:

- Automatically stabilizes parts in contact by generating stabilization spring stiffness via the model parameters NLKDIAGAFAC and NLKDIAGMINAFAC on the contact boundary.
- Default AUTO setting will stabilize parts with an initial gap opening greater than 1E-04 x model reference dimension.
- No longer have to use stabilizing springs or other techniques to prevent singularities.
- Parts can now have initial gaps between contact without convergence issues.
- Can also improve convergence rate for models with friction.
- Stabilization limited to contact boundary which minimizes any errors in the solution due to presence of stabilization stiffness.
- CONTACTSTAB may be set to a multiplier to the stabilization stiffness calculated automatically allowing models that are either being under or over stiffened to be easily adjusted.