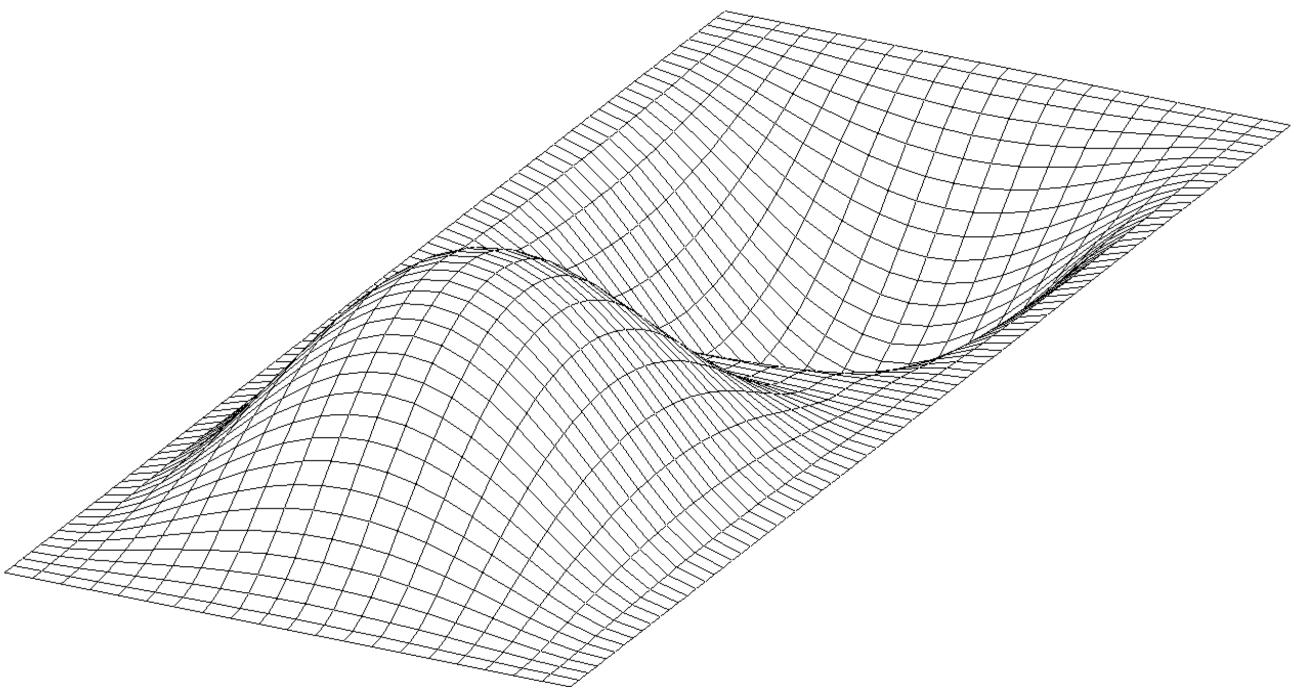
Autodesk Nastran 2024 Nonlinear Analysis Handbook

Version 11.1



AUTODESK

Autodesk Nastran 2024

Printed and bound in the United States of America.

TABLE OF CONTENTS

1. 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.

Unlike other solutions, subcase loads and results are additive. This allows different loads and boundary conditions to be applied in a specific sequence to the structure. Additionally, different nonlinear iteration parameters (NLPARM) may be specified for each subcase allowing further control. To initialize each subcase to zero, set PARAM,NLSUBCREINIT to ON. This setting allows multiple subcases with each having the same zero starting point.

2. 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 suchas 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(θ) ≈ (θ).

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 (LGDISP).

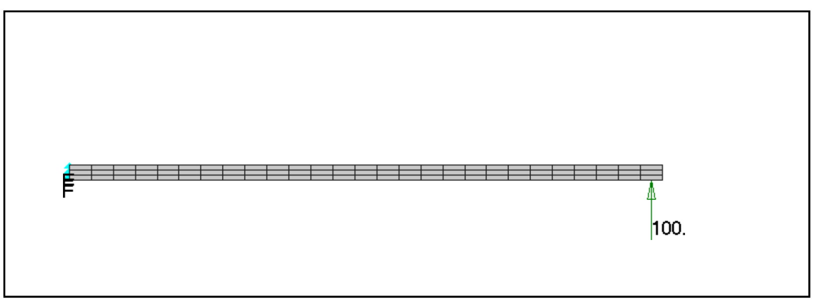


Figure 1. Cantilevered Rectangular Beam Loaded with 100psi Pressure Load.

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 (LGDISP=OFF). The second image shows the results of large displacements turned on, but follower forces turned off (LGDISP=2). The final image uses large displacement effects with follower forces and is the most accurate (LGDISP=ON).

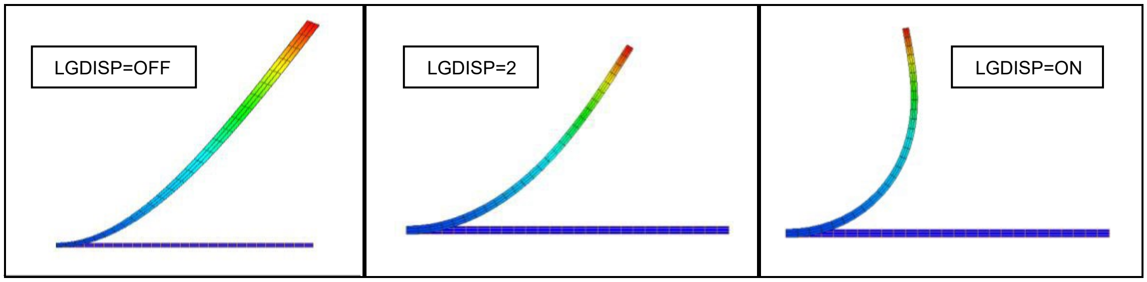


Figure 2. Deflection Results Using Three Different Nonlinear Settings.

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.

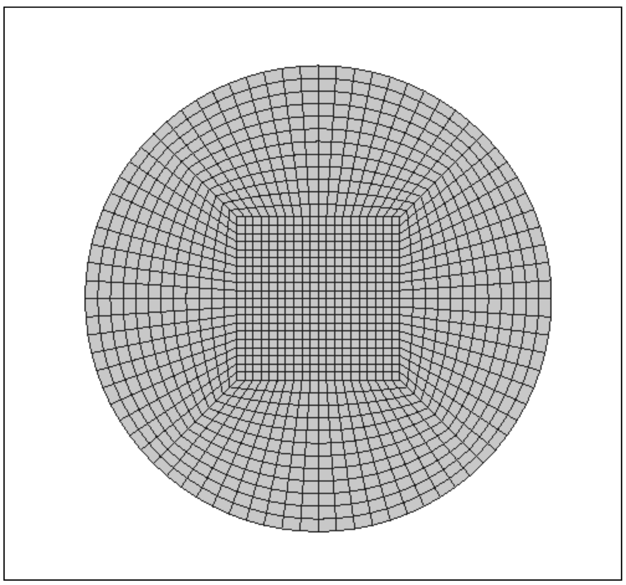


Figure 3. Pre-stiffened Drum Membrane Subjected to a Uniform Pressure Load.

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.

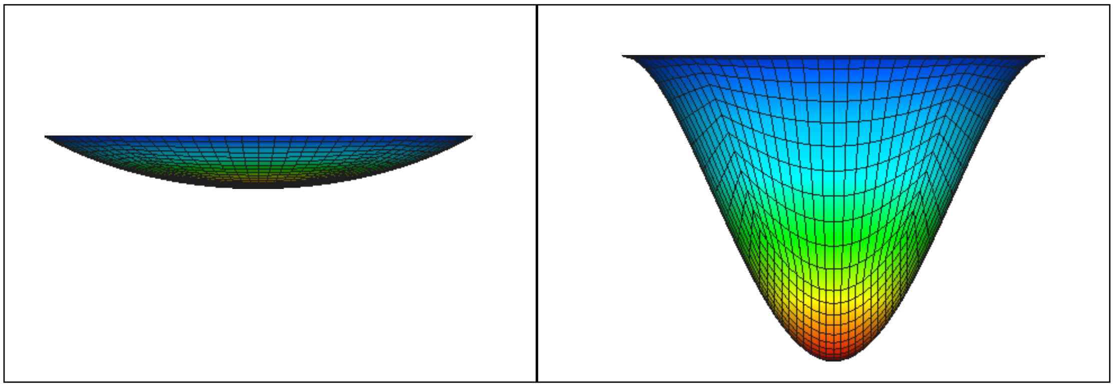


Figure 4. Displacements from a Nonlinear Analysis.

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.

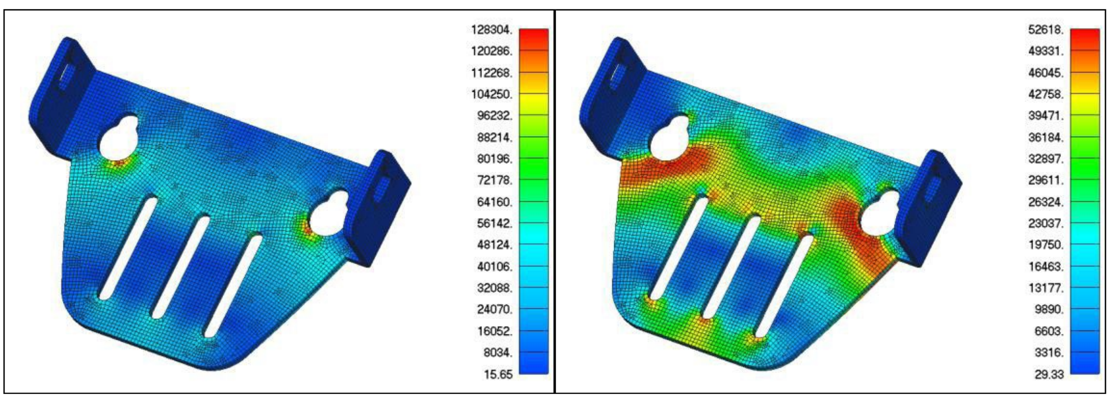


Figure 5. Stress Contour of a Metal Bracket with and without Nonlinear Materials.

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 thesequestions:

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 aresmall. 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.

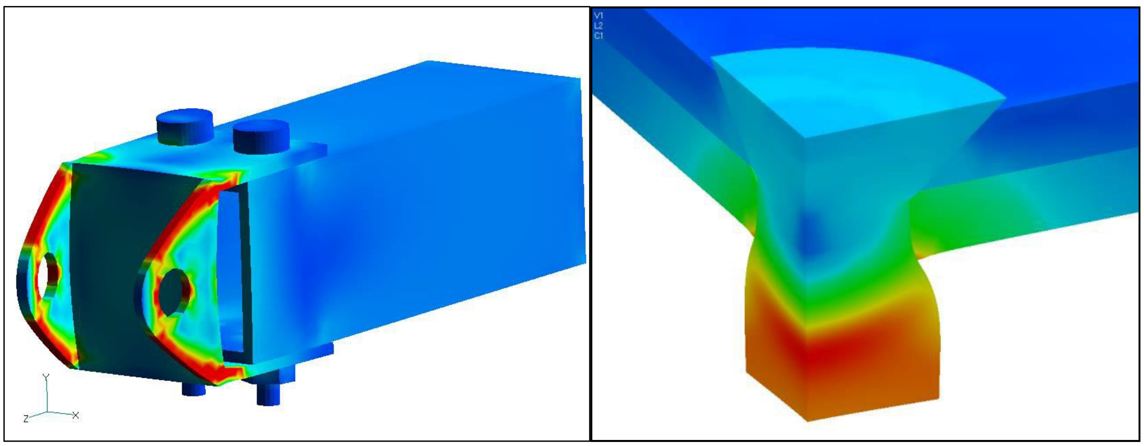


Figure 6. Trailer Hitch with Linear Contact Analysis and Rivet with Nonlinear Contact Analysis.

3. Nonlinear Static Analysis

3.1 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.

3.2 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 (uncheck the Large Displacements box in the Femap analysis window, or change LGDISP=OFF in the Autodesk Nastran Editor). If PARAM,LGDISP,1 or ON appears in the Model Input File, change this to OFF.

3. Turn off any nonlinear materials (either in Femap or in the Autodesk Nastran Editor by commenting out the MATS1 card).

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:

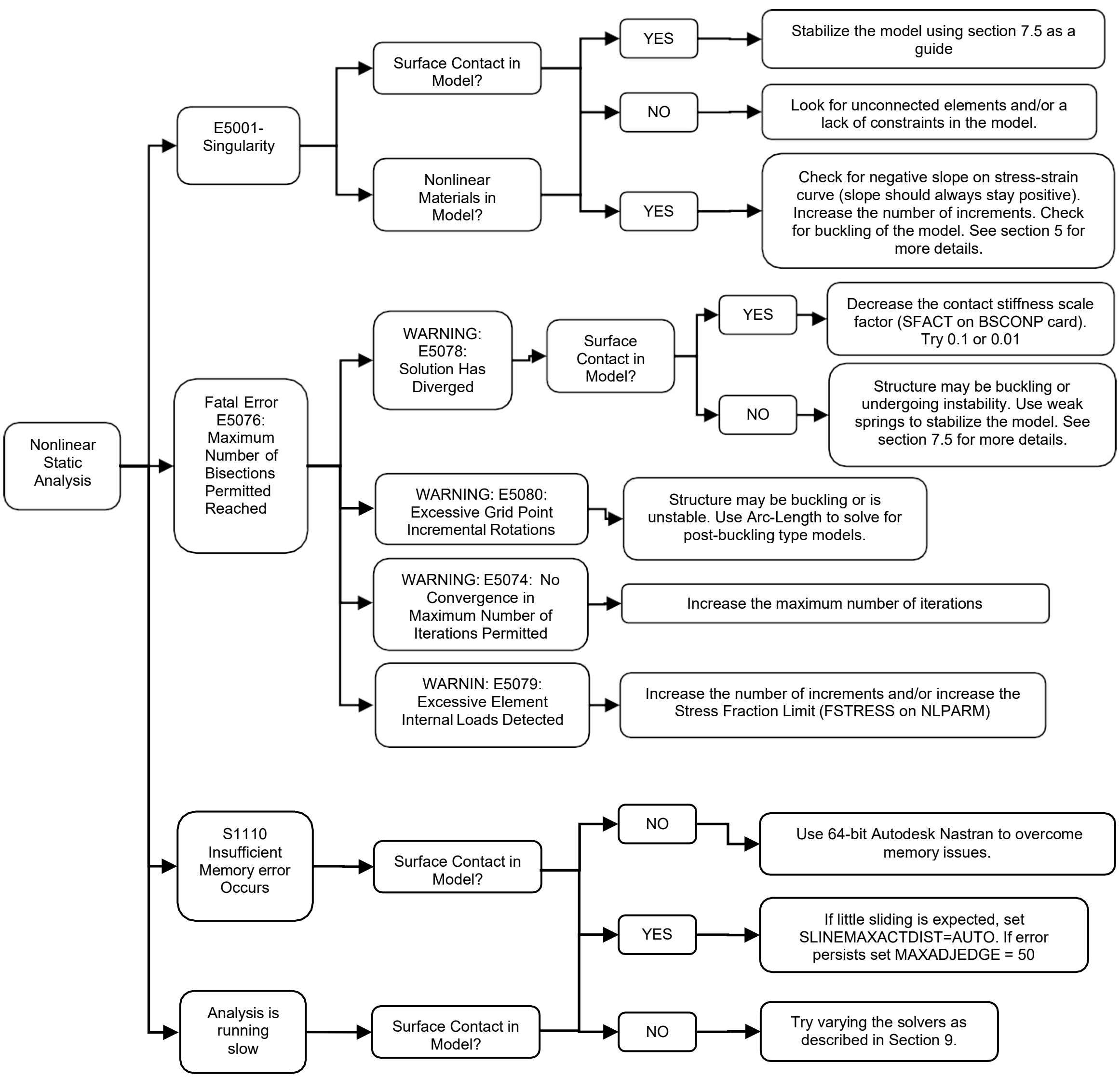
1. Run your model in a linear static solution and make sure it completes and that the results appear correct.

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 Autodesk Nastran Editor options (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.

3.3 Flowchart for Troubleshooting Nonlinear Static Analysis



4. 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 (see the Autodesk Nastran User’s Manual, Section 5 for more detailed information on 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).

4.1 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.

4.1.1 Understanding of 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 ball park 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.

4.1.2 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.

4.1.3 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. This strategy is used and explained in the Autodesk Nastran Tutorial Problem 8.

4.2 Automated Impact Analysis (AIA)

Autodesk Nastran V10 and above features a new automated impact analysis solution type that automatically does the steps mentioned in Section 4.1. The AIA solution type is activated via the IMPACTGENERATE Case Control card (see Reference Manual for more info). The solver will automatically perform the following steps to perform the AIA analysis:

- Drop Distance and Final Velocity – Autodesk Nastran AIA will calculate the distance the object will travel up to the point where contact is made, automatically repositioning the projectile.
- Automated Surface Contact Generation – Autodesk Nastran AIA seeks contacting mesh surfaces and creates contact between the two bodies.
- Vibration Characteristics – The natural frequencies of both the dropped object and the target are assessed at the point where they are just in contact and the dominant frequencies are identified.
- Impact Duration and Time Steps – Both the impact duration and the time step can be assessed from the characteristic mode being excited in the impact.
- User Input – Define the starting position of the projectile or drop object, the initial velocity, and the acceleration on the IMPACTGENERATE card.
- The analysis is solved as a NLT solution using all the data gathered above.

5. Nonlinear Materials

5.1 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 NLMATSFACT 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.

5.2 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 areon 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.

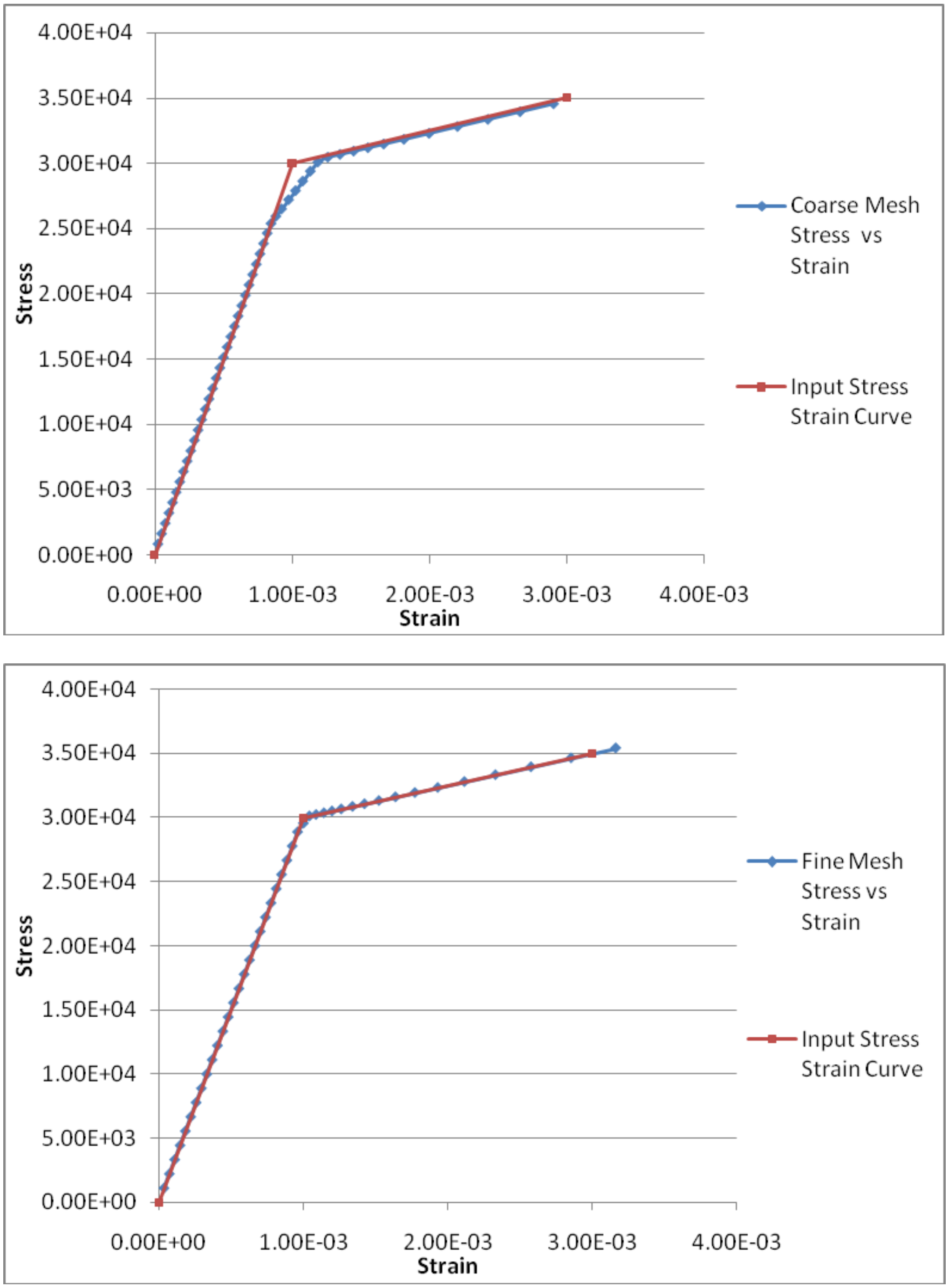


Figure 7. Stress-Strain Curves for Two Different Mesh Densities.

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.

5.3 Advanced Parameters

NLMATSFACT is a stabilization technique used in nonlinear material analysis when abrupt stiffness changes occur in the material. It controls how the solver follows the stress-strain curve during iterations. The default AUTO setting automatically lowers the scale factor when it encounters divergence. A small value of NLMATSFACT (for instance .01) is more stable but requires more iterations per increment.

6. Nonlinear Element Types

6.1 Tension Only Cable Element

The tension only cable element is defined using a PCABLE/CCABLE entry. On the PCABLE entry a material is referenced, along with a cross-sectional area. Either an initial cable tension or an initial cable slack can be specified. For an initial cable tension, the preload type can be set with the PTYPE option on the PCABLE entry. The PTYPE=INIT setting will treat the initial tension value as a starting preload. This value will be continuously added to the element internal axial load generated from the displacement of the end nodes. The PTYPE=CONT setting will force the cable internal load to always be the initial tension regardless of the element nodal displacements. Use of the CONT setting may result in slower than normal nonlinear iteration convergence.

The cable element must reference a linear isotropic material, but may be temperature dependent. Both thermal and inertial element loads are supported. The cable element, when subjected to lateral loading, requires a small amount of bending stiffness. The default bending stiffness is based on the square of the area of a circular cross-section.

6.2 Nonlinear Spring Element

The nonlinear spring element is defined using the PBUSHT entry in conjunction with the CBUSH and PBUSH entries. The PBUSHT entry references a table that defines the force versus deflection relationship. The stiffness value on the PBUSH entry is used initially on the first iteration of the analysis, so it is recommended to set the initial stiffness to match the initial stiffness on the force-deflection curve.

6.3 Gap Contact Elements

The settings for the Initial Gap (-1) instruct Autodesk Nastran to begin the analysis with the various physical spacing between the nodes that are specified in the model geometry. The value for the Compression Stiffness should be greater than the stiffness of the neighboring solid elements. The units are the same as the Young's Modulus entry. The optimal stiffness is determined iteratively. If the value is too stiff, the solution time may be excessive and decreasing the value too much may cause inaccurate results. With the Adaptive setting, the Max Penetration is a length parameter that allows Autodesk Nastran to adapt the gap properties for convergence. Its value should be much smaller than the solid element dimensions around it. The best value is determined iteratively, when the solver converges quickly to a good result. Small, non-zero values are chosen for the gap's tension and transverse stiffness (weak springs).

6.4 Slide Line Contact Elements

Autodesk Nastran adaptively adjusts the slide line stiffness. Entering the default value of zero in the Stiffness Scale Factor field allows Autodesk Nastran to determine the initial stiffness on its own. The Width values are used to calculate the stress. The Symmetrical Penetration button determines how Autodesk Nastran checks for penetration and is the most accurate setting that will result in the least amount of penetration.

Tip: For models that contain a large amount of slide line contact elements that will not undertake any sliding, we recommend setting the parameter SLINEMAXACTDIST=AUTO. This will result in a much faster solution time.

7. Surface-to-Surface Contact

7.1 Types of Surface Contact

General Contact - 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. For V9.1 and above, 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 areignored.

Welded Contact - 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.

Bi-Directional Slide - 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).

Rough Contact - 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 Weld Contact - 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.

7.2 Basic Steps for Setting up Surface-to-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 contactsegments 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 primary as the contact segment that has the least amount of curvature, or if it is cylindrical contact choose the primary 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.

7.3 Surface Contact Normals

When modeling surface contact between concentric cylinders (and other undulating surfaces), care must be taken when choosing which part is the secondary and which part is the primary. As a general rule the primary contact segment should be the outer surface, and the secondary the inner surface. In Figure 8, the primary 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 secondary 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.

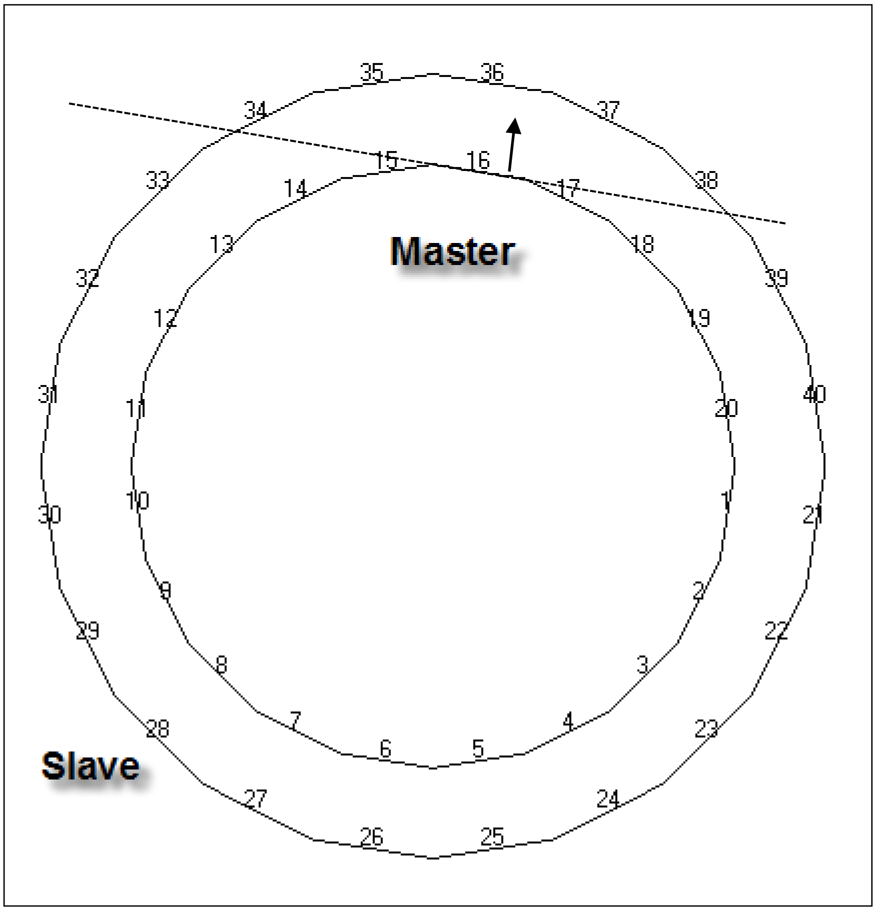


Figure 8. Incorrect Primary-Secondary for Concentric Cylinders.

In Figure 9 the correct primary-secondary setup is shown; and element 36 is shown as an example. A new plane is drawn for element 36. All secondary elements should be positive (below) the plane, and wesee that indeed all the secondary elements are below the contact plane.

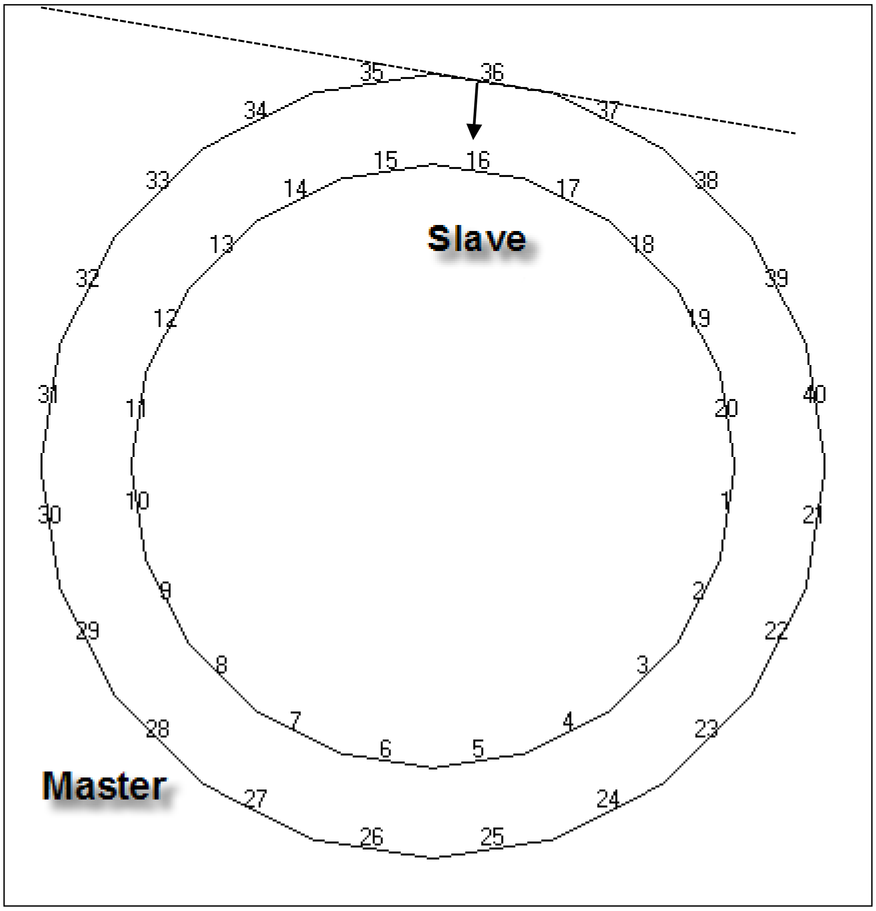


Figure 9. Correct Primary-Secondary for Concentric Cylinders.

If little sliding between the two parts is expected, the parameter, SLINEMAXACTDIST (located under Nonlinear Solution Processor Parameters in the Editor) 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.

7.4 BSCONP Settings

The explanations below expand upon the full list of definitions in the Autodesk Nastran Reference Manual under **BSCONP**.

1	2	3	4	5	6	7	8	9	10
BSCONP	ID	SECONDARY	PRIMARY	SFACT	FSTIF	MU	PTYPE	MAXAD	
	W0	TMAX	MAR	TRMIN	MAXRAD	MAXNAD	SMAX		

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

FSTIF – Frictional Stiffness for Stick. 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 x 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 using SMAX.

MAXAD – Maximum 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.

W0 – Surface Contact 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.

TMAX – Allowable Penetration. There are two methods for adaptive stiffness updates: proximity stiffness based and displacement based. If TMAX > 0.0, the displacement based update method is selected. When TMAX = 0.0 (default), the proximity stiffness based update method is selected. The recommended allowable penetration TMAX 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.

7.5 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 V10 and beyond 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, NLKDIAGAFACT, and NLKDIAGMINAFACT 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 byspecifying 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, NLKDIAGAFACT 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 NLKDIAGAFACT and NLKDIAGMINAFACT 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 allows models that are either being under or over stiffened to be easily adjusted.

For completeness, the alternate manual method of defining physical stabilizing springs is described below. This method can provide more control of exactly how the model is stabilized.

The figures below show a typical surface contact model consisting of 2 parts. The gold colors represent surfaces for contact. The lug back surface is held fixed in all directions. The stud/nut part initially has no constraints, since in the real-world friction and contact will keep it stable. To make this a stable model for an FEA solution, 6 springs are added (3 springs would be sufficient, but 6 allow for more symmetric stability). Each spring is a 1 degree of freedom spring with a small stiffness in the respective direction (x,y,z). The other ends of the springs are held fixed in all directions. These springs allow the contact to "seat" or activate during the initial iterations, and once the contact is activated, the weak springs will no longer influence the results. A good estimation of the spring stiffness to use is:

Kspring = 0.001 * Kmodel

where Kmodel = Applied Force / Est. Peak Deflection

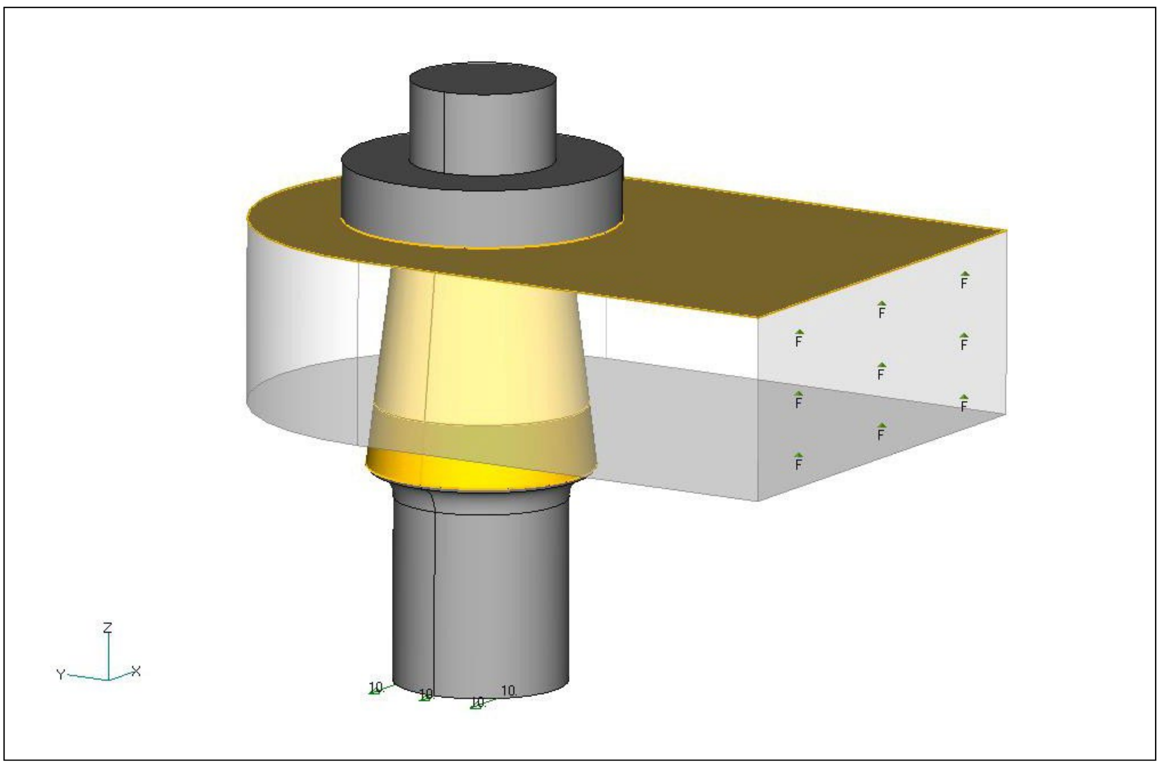


Figure 10. Two Part Surface Contact Model.

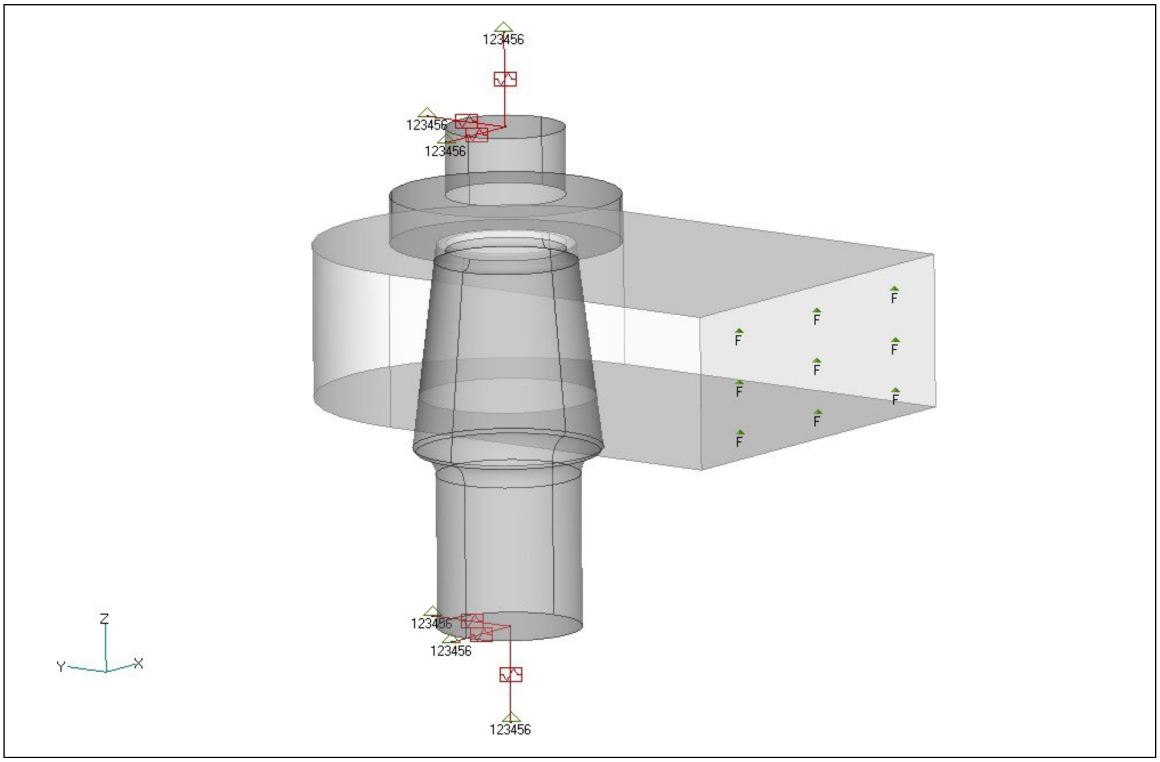


Figure 11. Six Grounded Weak Springs Added to Stabilize the Model.

7.6 Contact Output Vectors

Autodesk Nastran writes out several different types of contact output described below:

- Contact normal stresses
- Contact shear stresses

It is important to note that all contact output vectors are nodal based output. Because they are grid based data, contouring onto elements should only be used to get a qualitative feel for the results. To look at specific values, a nodal vector plot is recommended. The contact stresses are named SSHL CONTACT <type>. For instance, for the normal stresses, they will be named SSHL CONTACT NORMAL STRESS.

A positive contact stress indicates a compressive stress. For general sliding contact, the stresses should be primarily positive. There may be very small numerical negative stresses, but they should be several orders of magnitude smaller than the areas in contact. For welded contact regions it is possible to get large negative contact stresses. This simply means the weld has gone into tension.

The shear stresses listed are in the SURFACE coordinate system defined by the SURFACE Case Control command as shown in Figure 12. The default SURFACE coordinate system for grid point resultsis the basic system. More specifically the SURFACE x-direction is projected onto the SQUAD or STRI element surface and these stresses are in this direction. The RSLT quantities are the resultants of the x and y components.

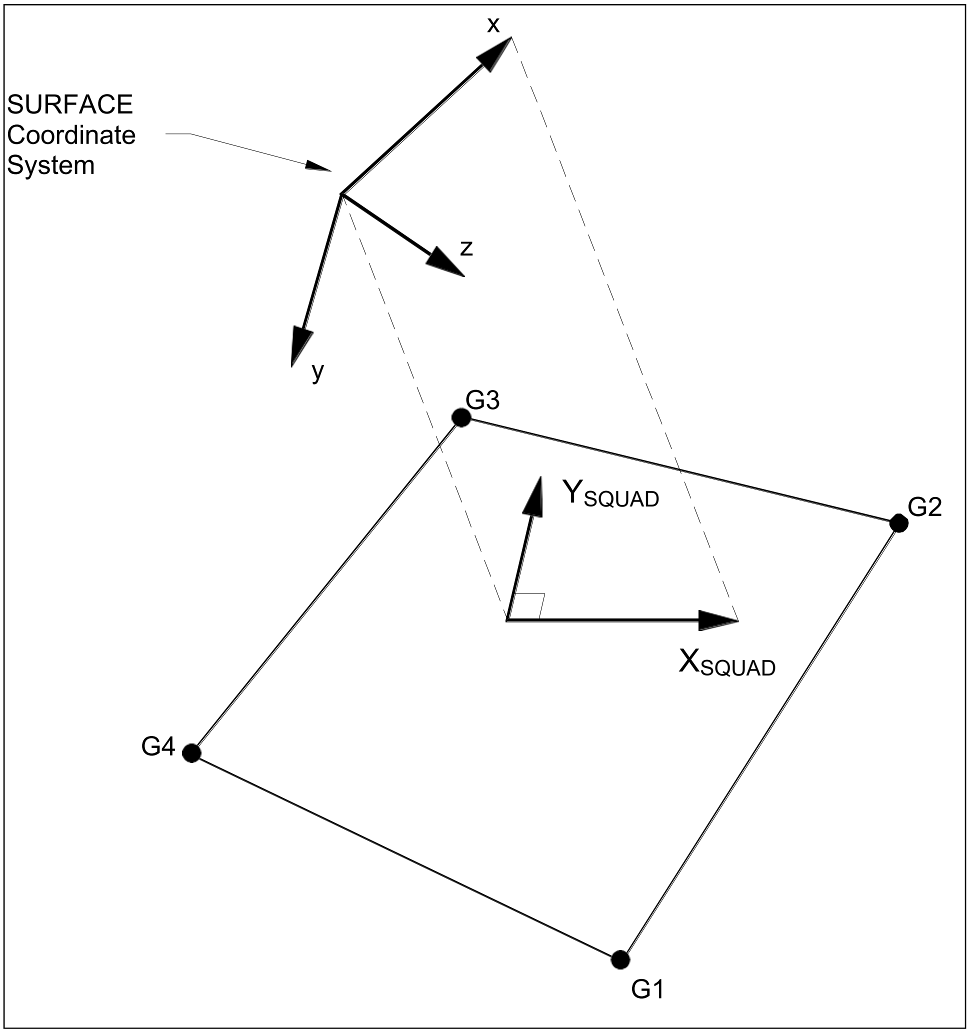


Figure 12. SSHL Shear Directions.

7.7 Troubleshooting Contact Analysis

There are many reasons a contact analysis may not run to completion. The following are some tips at getting the analysis to complete:

Problem: Your analysis is going very slow.

Solution: If you expect no sliding in the contact, set the parameter SLINEMAXACTDIST=AUTO. If you expect some sliding set the parameter to a value slightly larger than the maximum amount of sliding you would expect.

Problem: You are getting an **E5072: WARNING: CHECK NORMAL FOR CONTACT ELEMENT *id* CONTACT SEGMENT *id*** warning message during the analysis.

Solution: See Section 7.3 for additional information. This warning means your normals for the contact segments are pointing the wrong way. If they are shell elements, check the normal direction and reverseif necessary.

Problem: Your solution bisects until you reach maximum bisections and receive a fatal error of max bisections reached.

Solution 1: This is the most common problem in contact analysis. The Autodesk Nastran solver is having a hard time converging during the analysis. The best trick in getting this to converge is to set SFACT = 0.1 or 0.01. This will reduce the stiffness of the contact and make it easier for Autodesk Nastran to converge.

Note: Reducing SFACT will result in more penetration between contact surfaces, so after the analysis completes be sure to check the amount of penetration to see if it is reasonable.

Solution 2: If your model has little (less than one element length) or no sliding, set the TMAX value on the BSCONP to an accepted value of penetration (in your modeling units) that you are willing to accept.A recommended value is 1-5% of the associated thickness of the parts in contact. Note that the larger the value you choose, the easier it will be for the Autodesk Nastran solver to converge, but the less accurate the results may be.

Problem: You tried the above steps but your model still does not converge.

Solution 1: If your contact segment mesh is very coarse, consider re-meshing the area with a finer mesh.

Solution 2: Set the number of increments to a larger value (i.e., from 10 to 40).

Solution 3: If your model is almost converging, try reducing the work, load, or displacement tolerances.

Problem: You are modeling gears with contact or rotating geometry and the analysis is not working.

Solution: You may need to set SLINEMAXPENDIST to a value that is approximately the size of one of your element lengths. This will tell Autodesk Nastran to ignore any contact pairs that appear to be penetrating at a distance larger than this value.

Problem: You get an out of memory error at the very beginning of the analysis.

Solution: Make sure you are running the 64-bit version of Autodesk Nastran. If you have large contact segments with a lot of elements undergoing sliding and you receive this error, set the parameter MAXADJEDGE=50.

Problem: You are getting non-positive definite or singularity errors, yet you know your model is setup correctly.

Solution: See Section 7.5 on adding stabilizing springs. If only a checkout run is needed, a faster method is to use the parameter NLKDIAGAFACT which specifies the stiffness to be added to diagonal terms of the global stiffness matrix (in units of the model). Specifying a small positive value is useful in stabilizing a solution and preventing a non-positive definite or singularity error. In nonlinear static solutions the added stiffness is decreased at the completion of each increment so to reach the value defined by NLKDIAGMINAFACT at the completion of the last increment. If the parameter NLKDIAGMINAFACT is left at zero (default), the augmented stiffness will be set to zero on the last increment.

Problem: Your stress results appear spotty with stress concentrations where there should not be any.

Solution: The reason for this problem is due to irregular meshes. When two pieces are initially in contact certain nodes on the contact surface may be closer than other nodes solely due to the geometry of the mesh. The nodes that are closer will therefore have larger stress than other regions. The best way to get more even results is to increase the mesh density at the contact interface.

7.8 Simulating Interference/Press Fit Using Autodesk Nastran

7.8.1 Introduction

In Autodesk Nastran there are 3 approaches to simulating an interference fit:

Using thermal loads with an appropriate thermal expansion coefficient to get the desired prestress.
Modeling the parts with the interference and using surface contact to remove the interference.
Using Autodesk Nastran's initial strain approach.

All three methods have their advantages and disadvantages. For example, the first method is best used on 1D elements, such as bars/beams, as a quick way to simulate preloaded bolts. The second method is best when the geometry already contains the interference and it would be difficult to create new geometry. The third approach is best when the parts do not have the interference modeled in. The following two sections describe the 2nd and 3rd approaches mentioned above.

7.8.2 Interference Contact

Interference contact is the easiest method to setup as it only requires a few recommended parameters for best convergence. The general workflow is as follows:

Existing CAD geometry with interference is used without modification.
Surface contact is setup between the bodies and the model is setup for nonlinear analysis.
Displacement based penetration is used for contact with an initial soft contact stiffness allowing the interference to slowly be removed over several increments.
The solution iterates until convergence.

The recommended settings for interference contact models are as follows:

Set PARAM,NCONTACTGEOMITER,0 to prevent repositioning of penetrated nodes during initial processing.
On the contact property use displacement based penetration (TMAX) with an appropriate value based on your model dimensions. Note that the max penetration value is in the units of your model.
Set the stiffness scale factor on the contact property to a low value (0.01 recommended). This allows gradual transition to the equilibrium contact position.
If the geometry does not contain initial interference, a surface contact offset can be defined on the contact property (W0) to simulate any amount of interference.

In the image below, the cylinder was modeled with a height slightly larger than the inner dimensions of the box. Surface contact was setup between to two solids using the recommended settings above. The final equilibrium condition is determined by the solver with no penetration between the solids.

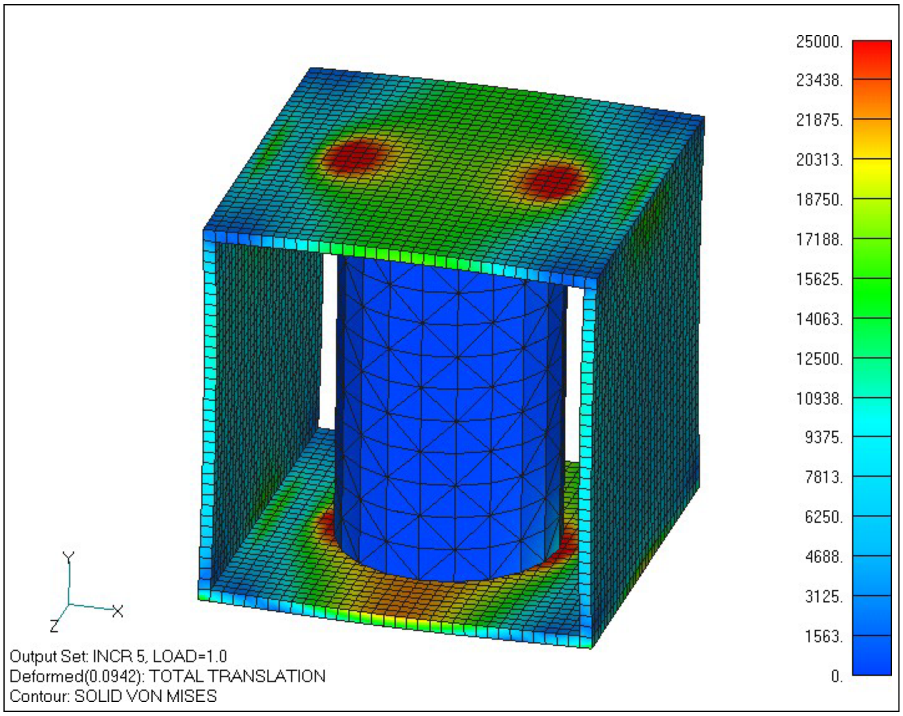


Figure 13. Stress Results from Interference Contact Model.

7.8.3 Initial Strain Model

The following section describes a technique to simulate an interference fit problem on a simple two cylinder assembly model. The initial strain approach is used to compress the inner cylinder. The strains from the initial strain model are reapplied to the inner cylinder of the interference fit model. The strain is resolved to the original state. Surface contact in the model opposes the initial strain resolution thus achieving the interference fit result. This technique has unique advantages over the thermal shrinking method. The user can specify the exact amount of shrinking in the radial direction without having to worry about a trial and error process to determine the right thermal load for the desired interference. Thefollowing figure shows the initial strain model set up (Units: English):

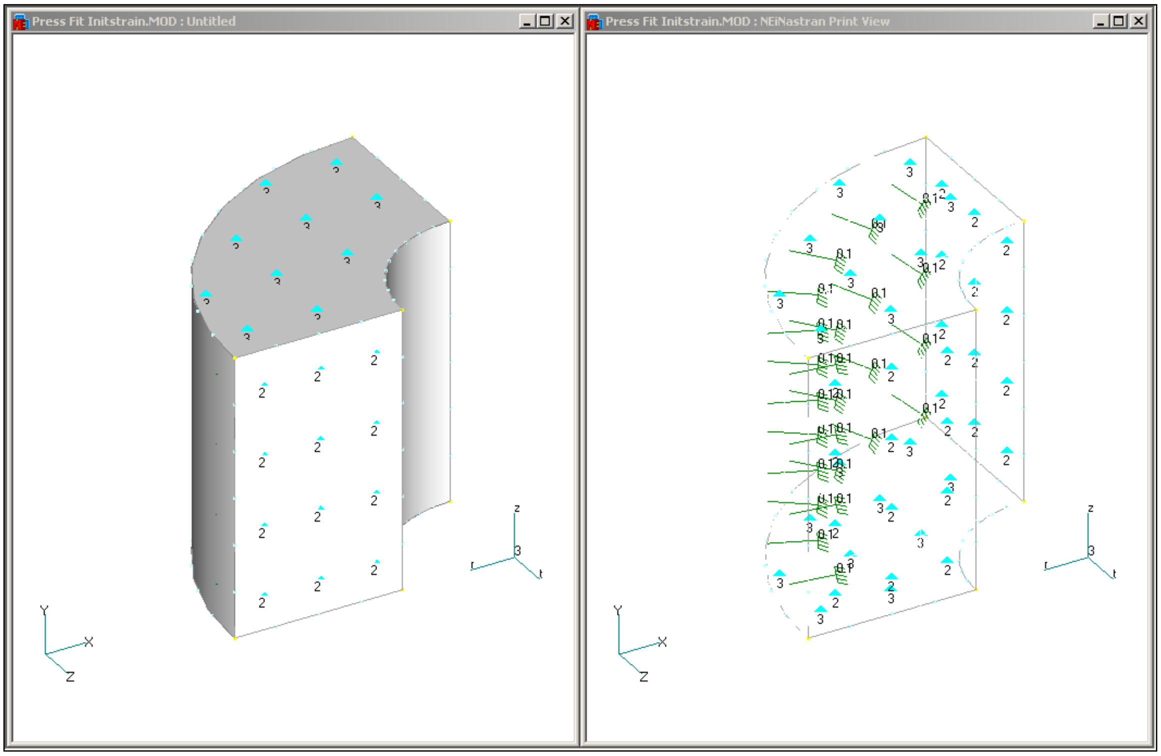


Figure 14. The Inner Cylinder in the Part Model with Boundary Conditions.

7.8.4 Boundary Conditions

Figure 14 shows the inner cylinder model. The bottom and top surfaces of the cylinder are constrained in the axial direction (z or 3 direction of the radial coordinate system seen in the bottom right areas of the figure). The symmetry faces are constrained in the theta direction (y or 2 direction of the radial coordinate system).

The enforced displacement load equal to the amount of interference desired is applied inward in the radial direction on the outer surface of the cylinder. The definition and output coordinate systems of all the grids of the model are changed to the radial coordinate system for convenience. Linear static analysis is performed.

7.8.5 Mesh

The final meshed model with linear hex elements, aluminum material and solid property, is shown below:

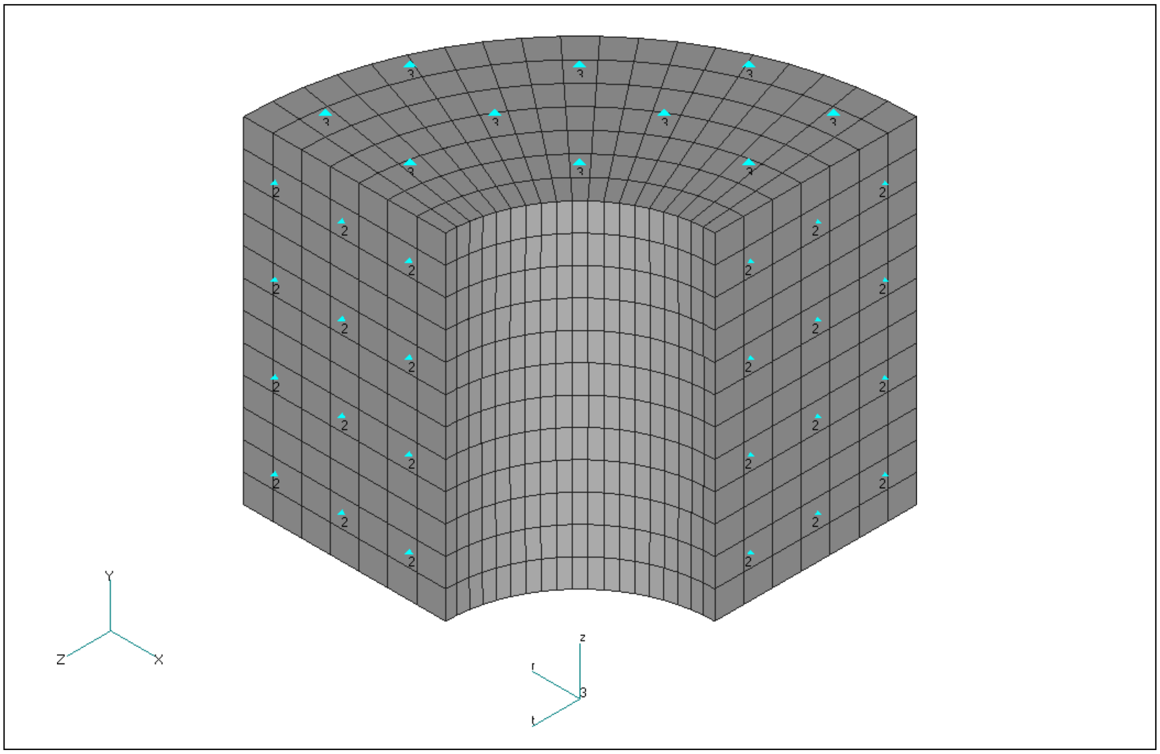


Figure 15. The Inner Cylinder Mesh.

Inner cylinder dimensions:

Inner radius: 2.5”
Outer Radius: 6”

Aluminum (common material):

E = 10700KSI
ν = 0.33

7.8.6 Model Parameters

Set the parameter TRSLSTRNDATA to ON under the Output Control Directives in Autodesk Nastran Editor. This parameter instructs Autodesk Nastran to generate a .bdf file of the strain data from the initial strain analysis. The strain data is later included in the interference fit analysis.

7.8.7 Analysis and Results

Radial displacement results of the initial strain model are presented in Figure 16.

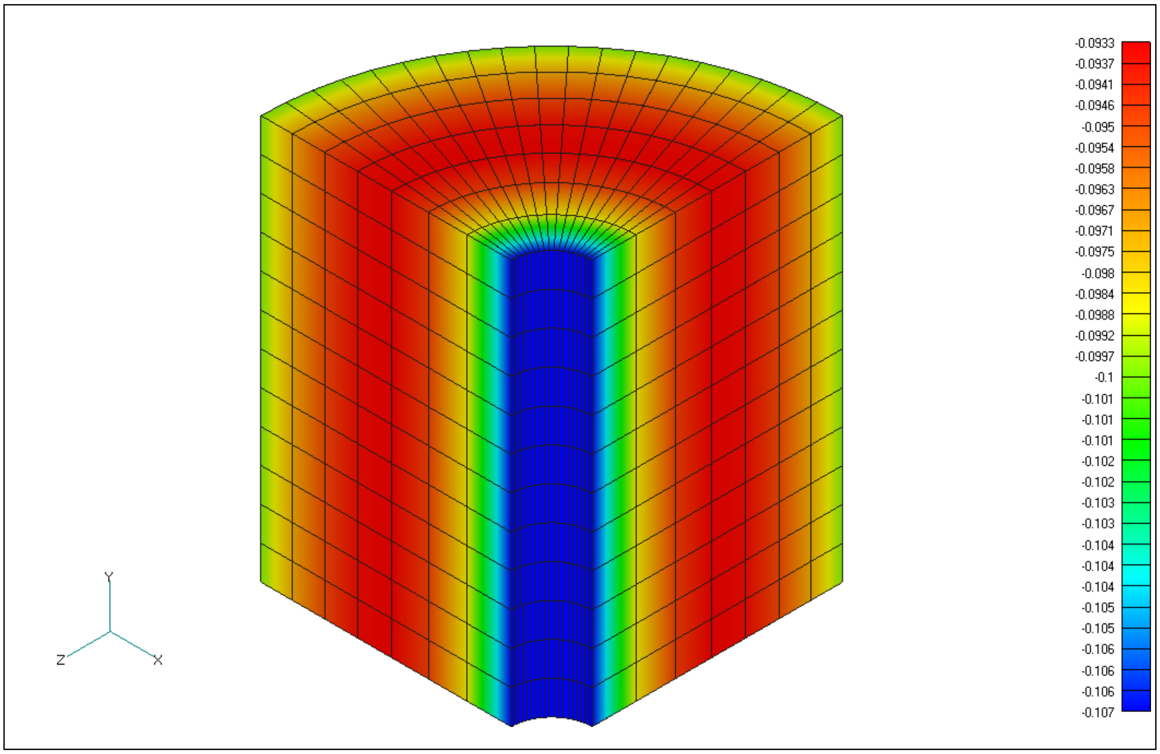


Figure 16. Radial Displacement Plot of the Inner Cylinder.

7.8.8 Interference/Press Fit Model

Figure 17 shows the set up of the assembly model with surface contact, linear hex elements with the same material and properties.

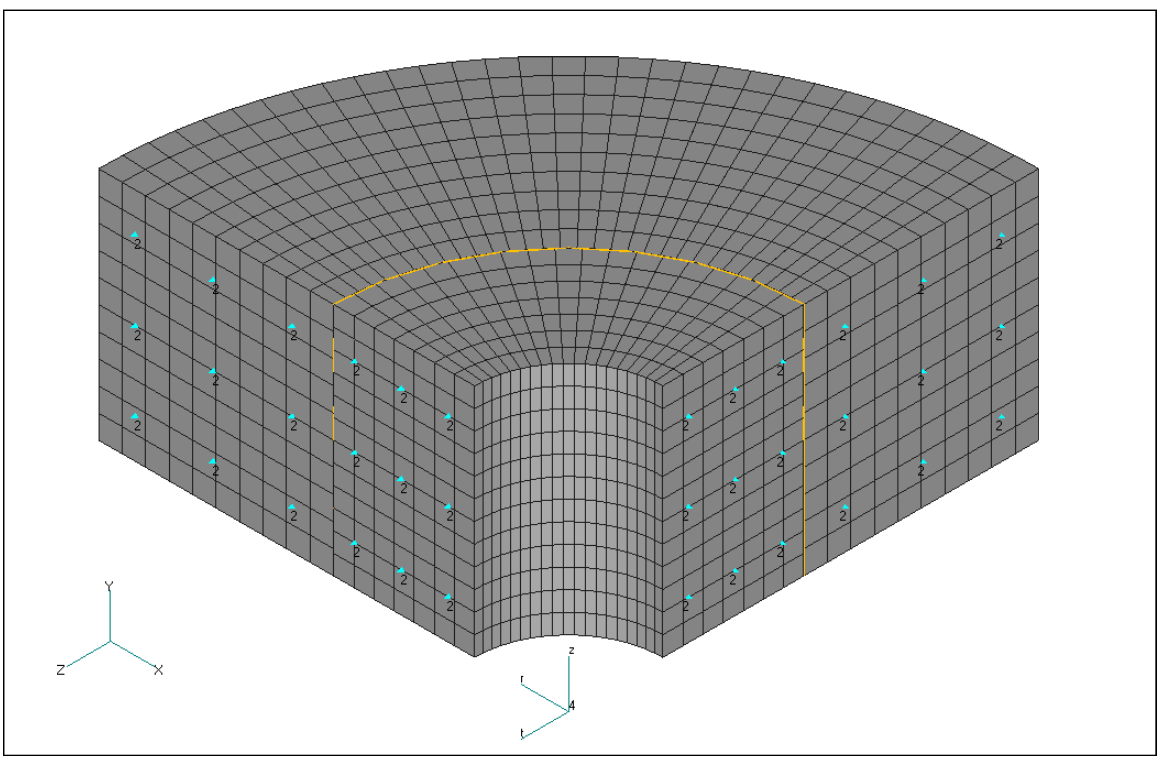


Figure 17. Assembly Model with Surface Contact.

7.8.9 Boundary Conditions

Figure 17 shows the model set up. The same mesh is used for the inner cylinder; this is important as the strain data has to be applied to the same nodes for the analysis to run properly. The outer cylinder is added to the model and meshed separately using hex elements. The symmetry faces are constrained in theta in a similar fashion using the radial coordinate system. The bottom faces (not seen in the figure) are constrained along the axial direction similar to the initial strain model. Surface contact is generated between the contacting surfaces (orange color in the figure).

A nonlinear static analysis is set up and the model is written out with a new name to be opened in the Editor.

7.8.10 Data Manipulation in the Editor

The following figure shows the Bulk Data in the Nastran tab inside the Autodesk Nastran Editor. The red boxes show the changes.

```
ID C,NEiNa
SOL NLSTATIC
CEND
  TITLE = Rebuilt Press Fit Model
  ECHO = NONE
  DISPLACEMENT(PLOT) = ALL
  SPCFORCE(PLOT) = ALL
  NLSTRESS(PLOT) = ALL
  STRESS(PLOT,CORNER) = ALL
  INITSTRAIN=100
  SPC = 1
  LOAD = 1
  NLPARM = 1
BEGIN BULK
$ ***************************************************************************
$   Written by : NEiNastran for Windows
$   Version    : 9.10
$   Translator : NEiNastran
$   From Model : C:\Sunny\Press Fit\Press Fit Full.MOD
$   Date       : Wed Feb 15 07:57:32 2006
$ ***************************************************************************
$
NLPARM         1       5                                       W      YES+
+                                                                        +
+
include, 'press fit initstrain.bdf'
PARAM,LGDISP,1
PARAM,OGEOM,NO
PARAM,AUTOSPC,YES
PARAM,GRDPNT,0
CORD2C         1       0      0.      0.      0.      0.      0.      1.+NEINAC1
+NEINAC1      1.      0.      1.
CORD2S         2       0      0.      0.      0.      0.      0.      1.+NEINAC2
+NEINAC2      1.      0.      1.
$ NEiNastran for Windows Coordinate System 3 : Radial
CORD2C         3       0     12.      0.2.15E-15     12.      1.2.09E-15+CS     3
+CS     3    11.      0.2.15E-15
$ NEiNastran for Windows Load Set 1 : Dummy
FORCE          1     750       0      1.   1.E-8   1.E-8   1.E-8
$ NEiNastran for Windows Constraint Set 1 : COnstraints
SPC            1       1      23      0.
SPC            1       3       3      0.
SPC            1       5       3      0.
```

Figure 18. Autodesk Nastran Editor with the Autodesk Nastran Input File.

Case Control:

The following Case Control command should be included in the interference fit model:

INITSTRAIN = 100

This instructs Nastran to look for Set **100** for initial strain values in the .bdf file from the initial strain analysis.

Bulk Data:

The following Bulk Data entry should be added to the input file:

INCLUDE, 'PRESS FIT INITISTRAIN.BDF'
This instructs Nastran to include the strain data in the .bdf file from the initial strain analysis.

Model Parameters:

Set the parameter, **SLINEMAXDIST** to **AUTO** under the Nonlinear Solution Control Parameters in the Editor. This minimizes the number of unnecessary contact elements generated during analysis.

7.8.11 Results

After the analysis is finished, the initial radial compression of the inner part is resolved back to the original state. The surface contact in the fit model opposes the resolution and thus the outer cylinder's inner surface is also pushed out radially as shown in the following figure.

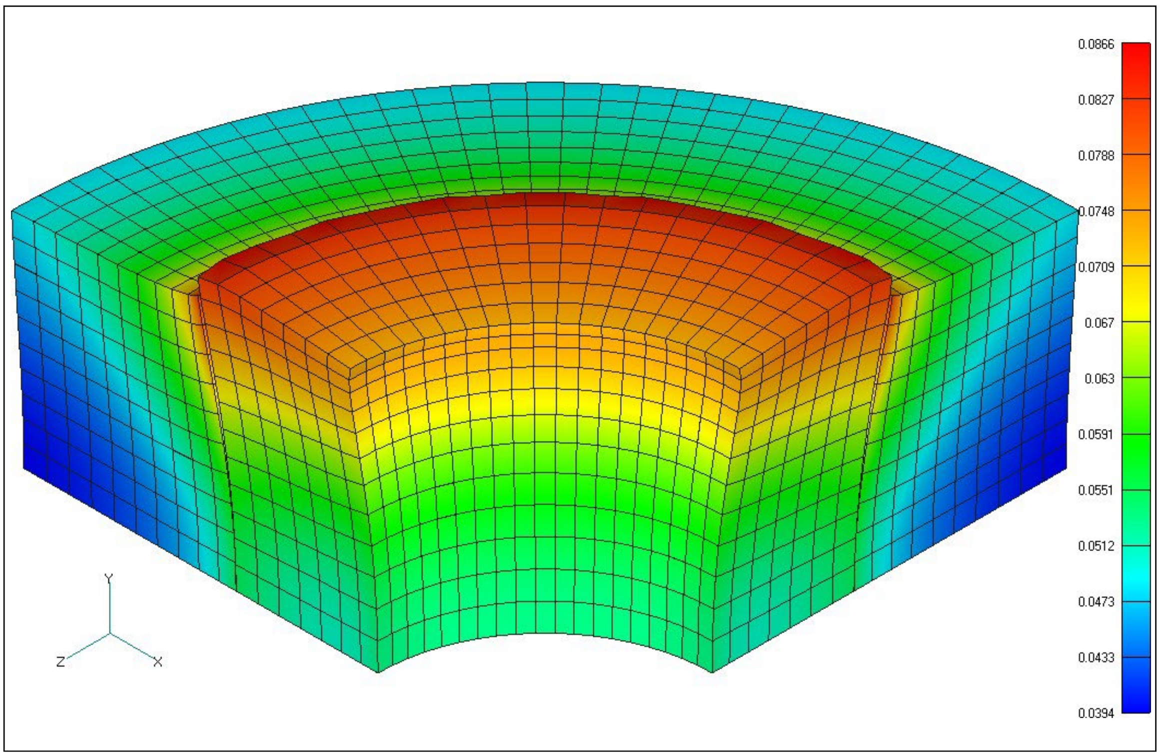


Figure 19. Radial Displacement Plot of the Assembly Model.

The inner cylinder's material tends to flower out over the top edge of the outer cylinder which is the expected flow of the material. The stresses in this region tend to be higher due to stress concentration.

7.9 Advanced Contact Settings

Autodesk Nastran V10 and above contains a new parameter, SLINESLIDETYPE, to aid in the convergence of surface contact models. The parameter can be set to DYNAMIC (default), STATIC, or DISABLE. Those 3 settings are described below:

- **DISABLE** - V9.2X behavior is used. No adjustment of contact stiffness.

- **DYNAMIC** - Proximity based stiffness using a softer initial penalty stiffness, SFACT, which is automatically increased from 0.01 to 1.0 depending on penetration, convergence, and divergence.
- **STATIC** - Displacement based stiffness where the maximum penetration (TMAX) is calculated automatically based on contact area and SLINEMAXDISPTOL (default = 1E-4).

The main advantage to the new parameter is that SFACT no longer has to be manually adjusted to find a good compromise between convergence and contact penetration. SLINESLIDETYPE=DYNAMIC is best for models that undergo sliding contact. SLINESLIDETYPE=STATIC is best for models with little sliding between contact.

8. Using Enforced Displacements in Nonlinear Analysis

Enforced displacements and rotations in Autodesk Nastran are referenced by an SPCD Bulk Data entry. Some confusion can arise when using enforced displacements because in most pre- and post-post processors they are created as a “load”, when in fact they are treated as a special type of constraint in Autodesk Nastran. In addition, an SPCD entry is referenced by a LOAD card in the Case Control.

8.1. Proper Method of Applying Enforced Displacements

For either linear or nonlinear static analysis, the following method will work correctly and not cause any errors in Autodesk Nastran:

1. Apply the enforced displacement in the appropriate direction.

2. Constrain the same nodes/surfaces/curves in the direction of the enforced displacement. It is important that only the nodes that have the enforced displacement are constrained in the direction of the displacement. All other constraints will be treated as actual boundary conditions.

8.2. Tricks for Linear and Nonlinear Analysis

For ease of use, Autodesk Nastran will automatically create a constraint for the SPCD entry in certain situations listed below:

1. Linear Static Analysis – A constraint will automatically be created for an SPCD entry for each subcase. However, if the model has varying constraint sets then the method listed in step 2 above must be used.

2. Nonlinear Analysis – If you use the same load set ID as the SPC ID then you do not need to constrain the enforced displacement. For example:

   ```
   SUBCASE 1
   SPC = 1
   LOAD = 1 (WHICH REFERS TO AN SPCD OR FORCE)
   SUBCASE 2
   SPC = 2
   LOAD = 2 (WHICH REFERS TO AN SPCD OR FORCE)
   ```

   This would work as is. No additional constraints would be required.

   However, the following would require that in SPC set 1 the DOF that the SPCD refers to be constrained.

   ```
   SUBCASE 1
   SPC = 1
   LOAD = 1 (WHICH REFERS TO AN SPCD OR FORCE)
   SUBCASE 2
   SPC = 1
   LOAD = 2 (WHICH REFERS TO AN SPCD OR FORCE)
   ```

9. Performance

Nonlinear analysis will generally take at least an order of magnitude longer to run than a linear analysis. Because of this fact, models should be simple and relatively small initially to gain insight into behavior and verify the approach taken. A linear static solution should be run first to verify boundary conditions and loading.

For nonlinear static solutions, several solvers are available, each with their own strengths. The list below describes the advantages of each:

1. **PSS** – Parallel sparse direct solver available in all solutions. This is the default solver for models below 100,000 DOF. This solver will generally be faster than the VSS solver especially on multiple CPU machines, but may require more memory. It is highly scalable for multiple CPUs or multi-core processors. Due to the large memory usage it can only be used for small to medium sized models when used in the 32-bit version of Autodesk Nastran. The 64-bit Autodesk Nastran is able to take advantage of large quantities of memory making the PSS solver an excellent choice.
2. **VSS** – The VSS solver is a fast solver when the entire model will fit into memory. For a typical desktop system with 2 GB of RAM, model sizes up to 100,000 DOF will run fastest using this solver.
3. **PCGLSS Direct** – This solver is best when running large models, and/or when there is not a lot of physical memory on the system. The PCGLSS Direct solver will generally use less memory than the VSS solver.
4. **PCGLSS Iterative** – This solver uses the least amount of memory and is best for very large models. Since this solver uses an iterative approach, element quality will directly affect how many iterations are needed. The PCGLSS Iterative solver is often faster than the PCGLSS Directfor good quality meshes.
5. **VIS Solver** – Unlike all other solvers, the VIS will not produce a fatal error for an ill-conditioned stiffness matrix. This solver is not generally recommended as it takes a significant number of iterations to converge. It is best used for performing checkout runs when problems exist with the model.

To change the solver, set DECOMPMETHOD=PSS/VSS/PCGLSS/VIS (located under Program Control Directives on the left side of the Editor). The default setting is AUTO and allows the program to pick the best method based on the RAM directive setting, material properties, model size, and solution selected in the model. To switch between PCGLSS Direct and Iterative, set the Parameter SPARSEITERMETHOD=ITERATIVE or DIRECT (located under Solution Processor Parameters).

9.1. Contact Models

By default Autodesk Nastran will generate a contact element for every combination between the primary contact elements and secondary contact elements. For instance, if a model contained a primary contact segment with 2000 elements in it, and a secondary segment with 500 elements in it, the total number of contact elements generated would be 1,000,000 (2000 x 500). This many generated contact elements requires a large amount of memory, and can often lead to a **S1110 Insufficient Memory** error. The parameter SLINEMAXACTDIST (located under Nonlinear Solution Processor Parameters) sets the distance (in the units of the model) that the solver will look for and generate contact elements. The AUTO setting will use a distance that is approximately 1 element width away. If we consider that for every primary contact element there will be 9 secondary elements within 1 element width, the amount of contact elements generated will only be 18,000 (2000 x 9) which is a reduction of 55x.

MAXADJEDGE helps to more accurately predict memory storage needed for contact. It represents how many other primary segments a secondary node can attach to. When SLINEMAXACT is set to AUTO and MAXADJEDGE is set to AUTO we assume small movement (i.e. only an element length of sliding max). If SLINEMAXACTDIST is not set to AUTO or zero, then you need to set MAXADJEDGE to something which represents how many primary segments a secondary node will slide over. Note that if you set MAXADJEDGE too small, it will simply return a T2260 FATAL ERROR indicating that not enoughelement storage space was allocated for contact.

10. Special Topics

10.1. Advanced NLPARM Features

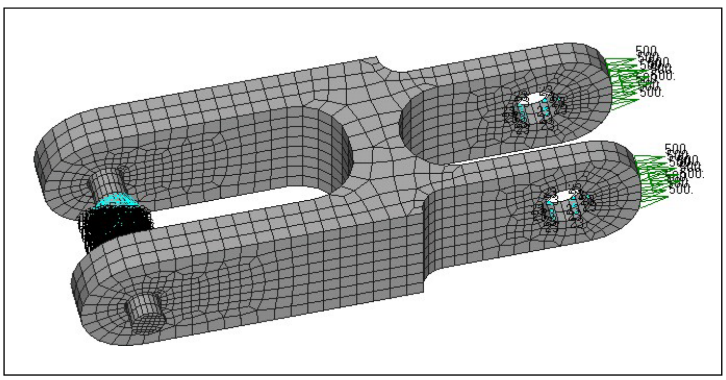
Autodesk Nastran V10 and later contains a new adaptive load increment method for nonlinear static analysis. It is controlled via the INITINC, MININC, and, MAXINC fields on the NLPARM cards. These are defined as:

- **INITINC** - Initial load increment (replaces the value determined using MININC and MAXINC)
- **MINNINC** - Minimum load increment
- **MAXINC** - Maximum load increment
- **STATIC** - Displacement

The advantages of this technique are as follows:

- No need to set a number of increments
- Can start the solution at a very small load for better stability
- Minimizes the number of increments based on convergence

In the example below, a small initial load is needed to seat the contact, so INITINC=1.E-4. MININC is set to 1.E-4, and MAXINC is set to 0.3. The graph below illustrates the exponential ramp-up of load as convergenceis achieved.



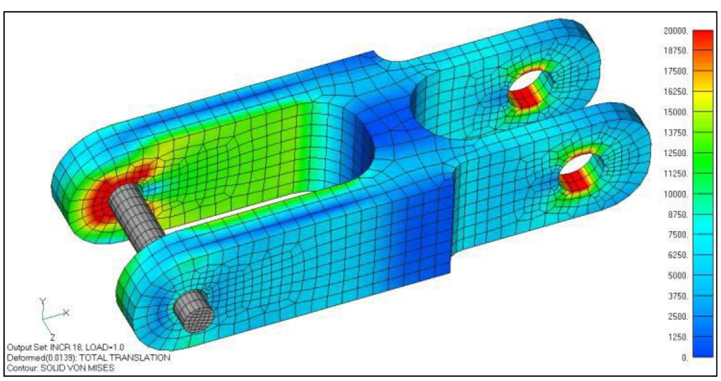


Figure 20. Unstable Model Needs Small Initial Seating Load.

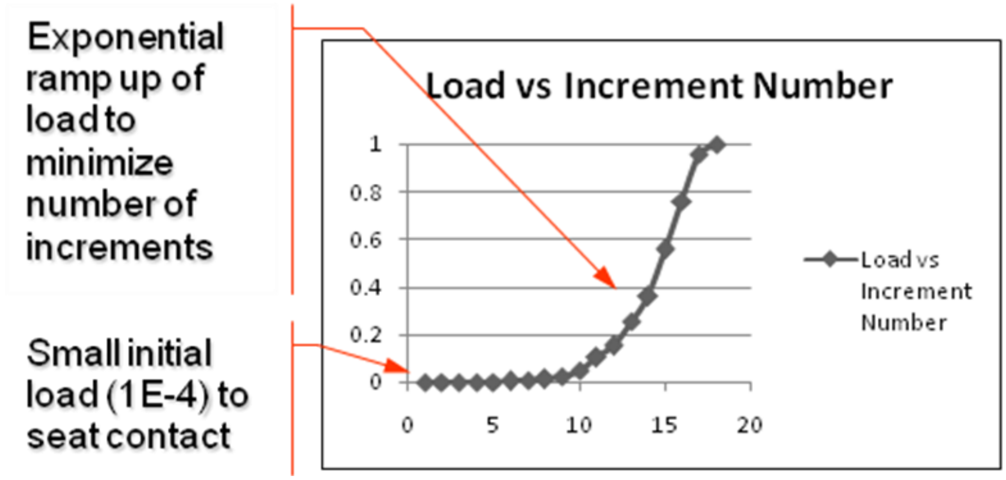


Figure 21. Load Ramp-up Using Adaptive Load Method.

10.2. Progressive Ply Failure Analysis

Composites are not able to absorb energy during peak loading the way a metallic material would due to composites' inherent brittleness. In a metallic material, yielding can absorb a huge amount of energy through plasticity thus often preventing catastrophic failure. In addition, composite materials have large variability in their material properties often forcing analysts to over-design their parts.

Traditional composite analysis consists of a linear solution that predicts first ply failure. This is a perfectly acceptable method, and with new failure theories such as Puck, LaRC02, and MCT can provide very accurate results. However, in many situations it is important to know what happens after first ply failure. Does the component catastrophically fail or is it simply a local failure? Understanding the behavior of a composite part beyond the initial ply failure is important to designing optimized and safe parts.

Progressive Ply Failure Analysis (PPFA) is a nonlinear solution where failure criteria is set for each ply of a composite model. During the nonlinear iterations, as the stresses or strains exceed the allowables in any particular ply that ply's stiffness is reduced (either to zero, or by a knockdown factor). This ultimately provides a much better understanding of what happens after first ply failure. Deciding whether to do a PPFA analysis (and hence a nonlinear analysis) ultimately comes down to the analyst, but needing to know the answers to any of the following questions would be a good indication that a PPFA analysis should be performed:

Does a first ply failure cause an abrupt failure of the component?
After first ply failure, does the load redistribute, thus maintaining the load carrying capability of the part?
How much additional load can the component take after first ply failure?

To perform a PPFA analysis, you must fill in the Reduction scale factors on the material card(s) (MAT1/MAT8, see Reference Manual for additional info). In addition, you must turn on the parameter, PARAM,NLCOMPPLYFAIL,ON to activate PPFA.