

Date	October 14, 2007	Memo Number	STI:07/05
Author	Sheldon Imaoka	ANSYS Revision	11.0
Subject	Sheldon's ANSYS.NET Tips and Tricks: Buckling in WB Simulation		
Keywords	Eigenvalue Buckling, Nonlinear Buckling		

1. Introduction:

Buckling may be a concern for some situations, where a user may need to modify the design to prevent buckling within a given load range. Conversely, failure analysis may need to be performed when a structure has collapsed.

ANSYS and Workbench Simulation have many tools to aid users in solving geometric instability problems, ranging from linear (eigenvalue) buckling to nonlinear, post-buckling analyses.

2. Eigenvalue Buckling as a Preliminary Step:

Similar to dynamic analyses, where performing a modal analysis to understand the frequency content of the system is a necessary first step, one should always perform a linear buckling analysis prior to solving a nonlinear buckling problem.

Eigenvalue or linear buckling provides a 'classical' solution to a buckling problem. While the critical load determined by linear buckling is *unconservative*, linear buckling is essential for the following reasons:

- Eigenvalue buckling gives the user an estimate of the critical load to induce buckling. While the reported value may be higher than the actual critical load, it provides a good starting point to understand how high of a load to apply in the nonlinear buckling analysis.
- Linear buckling can solve for many buckling modes. This is helpful to determine if there are more than one possible modes of buckling.
- The buckling mode from a linear buckling analysis can be used to generate 'imperfections' for use in a nonlinear buckling analysis.
- The solution time for eigenvalue buckling is typically much, much faster than a nonlinear buckling analysis, so a great amount of useful information comes at a relatively cheap computational price.

Workbench Simulation allows users to easily set up linear buckling analyses. First, a user must set up the loads and boundary conditions under a "Static Structural" analysis branch. A second analysis branch, "Linear Buckling," is then added – the "Initial Conditions" branch will reference the "Static Structural" branch, so loads, boundary conditions, and the stress state of the system can be obtained from this analysis. Under the "Analysis Settings" branch, the user can request any number of buckling modes – while the default is to solve the first buckling mode, the author recommends solving for 3 or more buckling modes in order to verify whether or not there may be multiple buckling modes that could be triggered. After solution, the buckling 'mode shapes' and load multipliers can be reviewed – the magnitude of all of the loads defined in the "Static Structural" branch times the load multiplier reflects the estimate of the critical load.

As the name implies, linear buckling does not account for nonlinearities. Hence, contact must be of the types "Bonded" or "No Separation," all material properties must

be linear elastic, and no large deflection effects are considered – these are factors which make the estimated load factor unconservative. Moreover, the load multiplier calculated by ANSYS is multiplied to *all applied loads*, so if a user has both constant and ramped loads, the user can iteratively adjust the ‘ramped’ load until the calculated load multiplier is near 1.0. Of course, the results from a nonlinear buckling analysis are typically of most interest to the user, so the user can set up their model with any type of nonlinearity required after this preliminary study.

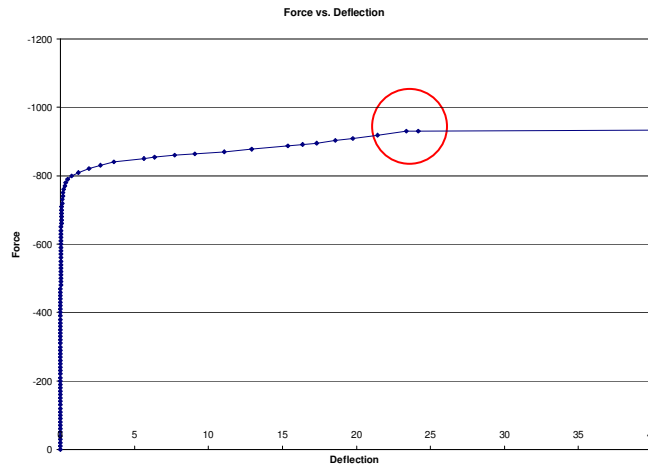
3. Nonlinear Buckling as a Design Tool:

The inherent difficulty in solving nonlinear buckling problems is due to the fact that a user may wish to solve these types of problems in a static framework, but at the point of geometric instability, the stiffness matrix may be singular (i.e., slope of zero on a force-deflection curve), thus preventing a solution from being obtained. In ANSYS, there are many ways to solve buckling problems, including obtaining post-buckling response.

If a user is only interested in the critical load and not interested in post-buckling response, a typical nonlinear static solution can be run.

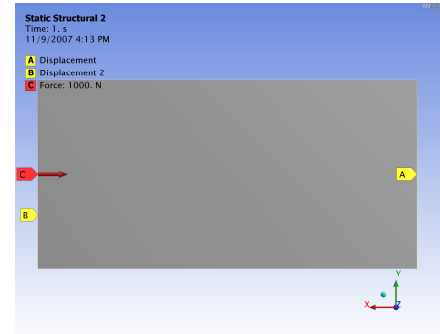
In Workbench Simulation, the user can duplicate the “Static Structural” branch used for the linear buckling analysis, and the load magnitudes should be adjusted by multiplying by the load factor reported in the linear buckling analysis. Large deflection effects should be activated under the “Analysis Settings” branch with Substeps manually entered: the initial and minimum substeps can range from 10 to 100, depending on the user’s preference, and the maximum number of substeps should be set to a high value, such as 1e5 or 1e6. Also, any sources of nonlinearities (e.g. contact, plasticity) should be included.

When the model is solved, near the point of geometric instability, ANSYS will bisect the solution *until the maximum number of substeps is reached and the solution does not converge*. The reason why such a high maximum number of substeps is used is because afterwards, using the “New Chart” feature, one can plot displacement vs. applied force to verify that non-convergence is due to geometric instability being encountered (e.g., force-deflection slope approaches zero). From this information, the user can determine the critical load that caused buckling, an example (plot in Excel) shown the figure below:



4. Including Initial Imperfections:

If one considers geometry that is symmetric, even a nonlinear buckling analysis may predict too high of a critical load. Consider a simple plate simply-supported at one end (A) and guided on the other (B) with a compressive load (C), as shown in the figure on the right.



Although one may assume that buckling should occur in the out-of-plane direction, if the geometry is modeled perfectly, this may not occur.

One can use a buckled mode shape calculated from a linear buckling analysis to create a small imperfection or perturbation in the mesh for use in nonlinear buckling analyses. (Similar techniques involve applying small point loads in specific areas to induce buckling.) This can be accomplished in Workbench Simulation using a “Commands” object, using the UPGEOM ANSYS command.

All ANSYS result files are contained in subdirectories in the “Simulation Files” folder, such as “Linear Buckling”. (These folders use the same name as the analysis branch in the Workbench Simulation database; when more than one analysis branch shares the same name, a number in parentheses will follow for newer analyses.)

To use a buckled mode shape to perturb the geometry, first determine the buckled mode shape to use as well as the maximum amplitude. In the nonlinear static analysis branch, insert a “Commands” object with the following APDL commands:

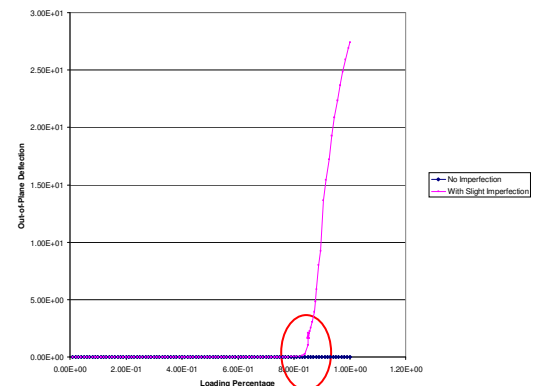
```
/PREP7
UPGEOM,factor,1,mode,'..\Linear Buckling\file',rst
/SOLU
```

Note that “factor” will be multiplied to the buckled mode shape “mode”, and the nodes will be moved to new locations. For example, a user may want to use the first buckled mode shape to perturb the mesh – the maximum amplitude may be 0.5. Using information such as manufacturing tolerances or a given percentage of the thickness of the part, the user may wish to include an imperfection with a maximum value of 0.002. The user could then use the following commands to include the first buckled mode shape:

```
/PREP7
UPGEOM,0.004,1,1,'..\Linear Buckling\file',rst
/SOLU
```

When using ANSYS commands in Workbench Simulation, note that the system of units should not be changed (the ANSYS result files will be based on the active units when the Linear Buckling analysis was performed). Also, because the mesh is being modified directly by the ANSYS solver, Workbench Simulation will not display the updated nodal position – assuming that this ‘imperfection’ is small, it should not pose a significant problem in post-processing.

The aforementioned simply-supported plate was loaded in-plane with and without an imperfection (based on the first buckled mode from the eigenvalue buckling analysis). The chart on right shows the plot of displacements in the out-of-plane direction – note that without any imperfection, no buckling occurs. With the small imperfection, buckling occurs around 85% of the applied load.



5. Capturing Post-Buckling Behavior:

In situations such as failure analysis, post-buckling behavior is sought. If the loading is displacement-controlled, calculating the response past the buckling point is typically not a problem. For force-controlled analyses, however, zero stiffness would result in a singular matrix.

For these situations, users can solve a quasi-static analysis by including inertial effects – by solving a transient analysis (with some damping to lessen inertial effects), one can solve past the point of buckling since the inclusion of the inertial and damping terms no longer make the matrix singular.

Another technique is using the arc-length method. For the full Newton-Raphson method, which is the default solution method in nonlinear problems, the applied loads are always increased incrementally, which causes problems at the point of buckling. The arc-length method, however, adjust the applied load based on a relationship between the calculated incremental displacement and the arc-length ‘radius’, thus enabling the applied load to *decrease*, if needed.¹ As a result, this can help capture post-buckling behavior beyond the point of instability.

An alternative method has been introduced in ANSYS 11.0, and this is the nonlinear stabilization technique, controlled with the `STABILIZE` command. Because it is a relatively new feature and easy to implement in Workbench Simulation, the subsequent discussion will focus on this method.

Conceptually, nonlinear stabilization in ANSYS can be thought of as adding artificial dampers to all of the nodes in the system. Before the critical load is reached, the system may typically have low displacements over a given timestep – this can be thought of as a low *pseudo velocity*. A low pseudo velocity would not generate much resistive force from the artificial dampers. On the other hand, when buckling occurs, larger displacements occur over a small timestep, so the pseudo velocity becomes large, and the artificial dampers would generate a large resistive force.

Nonlinear stabilization can be specified by either inputting a damping factor or energy dissipation ratio. The energy dissipation ratio typically ranges from 0 to 1, and it can be thought of the ratio of work done by the damping forces to the potential energy. When this method is used, in the “Solution Information” solver output, the effective *damping factor* will be printed for reference purposes:

```
*** DAMPING FACTOR FOR NONLINEAR STABILIZATION = 0.1840E-01
```

Because of the easier interpretation of the energy dissipation ratio value, the author recommends using this first, where a value closer to 0 reflects less damping.

Other controls with nonlinear stabilization include using constant values or ramping the stabilization forces to zero at the end of the load step, as well as determining when nonlinear stabilization is activated.²

¹ Refer to Section 15.13.6 “Arc-Length Method” in the *ANSYS 11.0 Theory Reference* for additional details.

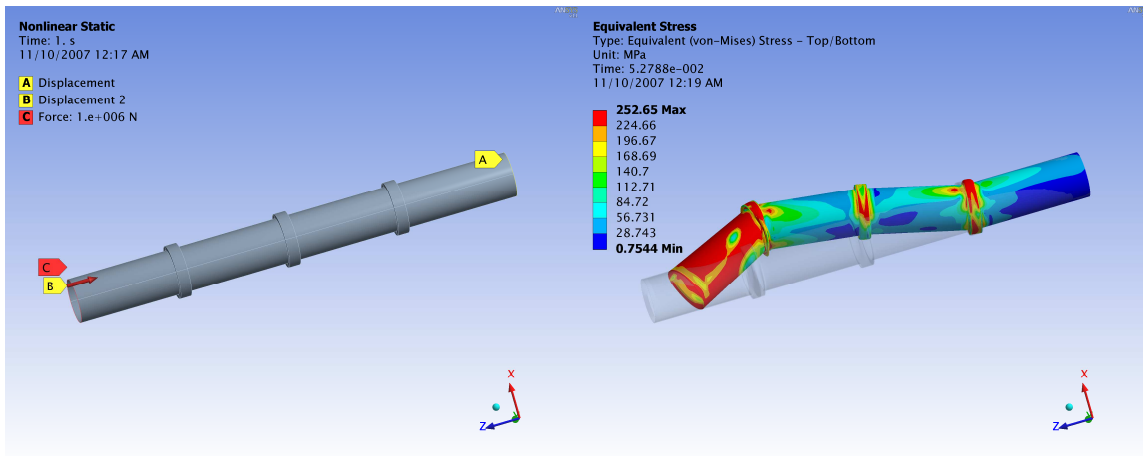
² Refer to the `STABILIZE` command in the *ANSYS 11.0 Commands Reference* for more information. §8.11 “Unstable Structures” of the *ANSYS 11.0 Structural Analysis Guide* also provides good guidelines on solving buckling problems.

To use nonlinear stabilization, a user simply needs to insert a “Commands” object under the “Static Structural” branch with the `STABILIZE` command and relevant arguments. For example, to use 0.01% constant energy dissipation ratio, one can use:

```
STABILIZE,CONSTANT,ENERGY,1e-4
```

Note that because nonlinear stabilization can be turned off or used only for certain load steps, a user may wish to separate the load history in multiple Steps via the “Analysis Settings” branch, then activate nonlinear stabilization only when needed through the Details view of the “Commands” object.

A tubular system was loaded in compression, as shown in the left-hand figure below. With nonlinear stabilization active, the post-buckling behavior was captured, as shown in the contour plot on the right.



6. Conclusion:

This memo provided an overview of linear and nonlinear buckling, as well as introduced the nonlinear stabilization feature, new at 11.0. Both buckling for design (determination of the critical load) and failure analysis (post-buckling response) can be investigated with tools available with ANSYS.