Autodesk® Moldflow® Insight 2012

AMI Stress Analysis

Revision 1, 22 March 2012.

This document contains Autodesk and third-party software license agreements/notices and/or additional terms and conditions for licensed third-party software components included within the product. These notices and/or additional terms and conditions are made a part of and incorporated by reference into the Autodesk Software License Agreement and/or the About included as part of the Help function within the software.

Contents

Stress analysis

1

Stress analyses are used to identify structural related problems, typically with the strength, stiffness, and life expectancy of plastic products.

The Stress analysis program performs isotropic and orthotropic stress analysis of normal or fiber-reinforced thermoplastic materials. Stress analysis for injection molding predicts actual molding stiffness. It analyzes a product for possible structural defects or failure points when the product is exposed to a load. In order to perform a Stress analysis, your study must contain:

- A selected material model
- Model constraints which prevent rigid body motion (global translations and rotations) of the model, in response to the applied loads, whilst not interfering with the shrinkage of the part.
- Loading and boundary conditions.

Constraints must be carefully applied so that they are representative of the physical situation being modeled. By default, the six degrees of freedom of all nodes in the model are free. To constrain the model, you specify which degrees of freedom are to be constrained for one or more nodes.

The study file (*.sdy) also contains any local coordinate system definitions that you have set.

Stress analysis types

2

Stress analyses are used to identify structure-related problems, typically with the strength, stiffness, and life expectancy of plastic products.

The following stress analysis types are available and can be selected from the Process Settings Wizard:

Small deflection	Select this analysis type if you expect the deformation behavior of the part to be stable. The small deflection analysis provides the final deformed shape of the part, assuming linear stress-strain behavior within the part.
Large deflection	Select this analysis type if you expect the deformation behavior of the part to be unstable, as determined from a previous buckling analysis, if the deformation behavior is borderline stable/unstable and/or you want the most accurate prediction of the shape of the part. The large deflection analysis provides the final deformed shape of the part, allowing for nonlinear stress-strain behavior within the part.
Buckling	A buckling analysis is used to determine whether the deformation of the part will be stable or unstable under the applied load(s). If the buckling analysis indicates the deformation behavior of the part is stable (critical load factor > 1), the deflection results obtained from the buckling analysis provide a good indication of the final deformed shape of the part. If the buckling analysis indicates the deformation behavior of the part is unstable (critical load factor < 1), you need to run a large deflection analysis to determine the final deformed shape of the part.
Modal frequency	Select this analysis type if you want to determine the natural, undamped frequency response of the part. Theoretically, this analysis type is similar to the buckling analysis, however the physical interpretation of the results is different.
Creep	Select this analysis type if you want to analyze the creep behavior of the part, that is, its time dependent deformation under the applied load(s).

Stress analysis types

Stress analyses are used to identify structure-related problems, typically with the strength, stiffness, and life expectancy of plastic products.

Setting up a Stress analysis

The following table summarizes the setup tasks required for a Stress analysis of a non fiber-filled, or fiber-filled thermoplastic material.

Setup task	Analysis technology
Setting up a Fill analysis	
Setting nodal constraints[1]	

Optional setup tasks

Setup task	Analysis technology
Defining a local coordinate system[2]	
Activating a local coordinate system	
Nodal loads on page 48[3]	
Elemental loads on page 50[4]	

Stress analysis types

Use this dialog to specify settings for a Stress analysis.

Process Settings Wizard dialog—Stress Settings

This page of the **Process Settings Wizard**, which can be accessed by clicking (**Home tab > Molding Process Setup panel > Process Settings**), is used to specify the stress analysis related process settings for the selected analysis sequence.

Stress analysis is available only for Midplane analysis technology. The **Process Settings Wizard—Stress Settings** page appears only if the selected analysis sequence includes Stress analysis. Depending on the analysis sequence, it may be necessary to click **Next** one or more times on the **Process Settings Wizard** dialog until the **Stress Settings** page appears.

NOTE: Some of the items listed below may not be available on the current dialog. This is dependent on the selected analysis sequence.

[1] Nodal constraints also include spring constraints.
[2] Local coordinate systems are useful for constraints or loads not acting in a global coordinate system direction.
[3] Nodal loads include point and edge loads.
[4] Elemental loads include pressure, surface, thermal and volume loads.

Stress analysis type	Select the type of Stress analysis to run.
Stress result(s) to output	Specify which laminate-based stress-related results the solver will output.
Consider gate surface and cold runners?	Specify whether cold runners and/or gate surface elements, if present in your model, are taken into consideration during Warp or Stress analysis.
Matrix solver	Select the equation solver to be used in the Warp analysis.
Molding material	Select and edit the material to analyze.

Creep Analysis Solver Parameters dialog

This dialog is used to specify the settings for the solver parameters of a creep Stress analysis.

To access this dialog, ensure that you have selected an analysis sequence that includes Stress, click  (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Stress Settings** page of the Wizard, set the **Stress analysis type** option to **Creep**, then click **Solver parameters**.

NOTE: All solver parameters have a default value that will be suitable for most analyses.

Small deflection Stress analysis

Small deflection analysis is the most common type of analysis and is the basis of both large deflection and buckling analysis. A thorough understanding of small deflection analysis is an important prerequisite for understanding the other analysis types.

Small deflection analysis is also frequently called *linear analysis*.

Assumptions

Small deflection analysis is based on the assumption that the displacements and corresponding stresses and strains, resulting from the application of loads to a flexible body, are a linear function of the magnitude of such loads.

To achieve this linearity, we must assume that the material from which the body is made is elastic and that the displacement gradients are small enough everywhere to neglect the effects of the associated changes in geometry. In other words, we must assume that the body behaves as if it is both materially and geometrically linear. That is, the resulting change in geometry does not significantly alter the stiffness of the structure. Under these conditions, the governing equilibrium equations take the familiar form:

$$K \cdot U = P$$

where:

- K is the assembled stiffness matrix of the idealized structure,
- U is the unknown nodal displacement vector, and
- P is the vector of equivalent nodal forces.

Constitutive Model

Within the restriction of a linear elastic material (a material which obeys Hooke's law), it is nevertheless possible to include both homogeneous and non-homogeneous materials. In the present case, for shell elements (LMT3, LBT3), we will however assume that the material is homogeneous with respect to the thickness direction. With this restriction, the material properties can vary from one point in the structure to another, but at each point there will be two orthogonal directions with respect to which the elastic moduli take on principal (that is, maximum and minimum) values. These directions are called directions of orthotropy and the associated material model is called orthotropic.

Two alternative elastic material models are available. The first is an isotropic model (material is non-directed and homogeneous). The second is a special case of the orthotropic model described above, in which any point-to-point variation in the values of the principal moduli is ignored (material is multi-directed but homogeneous). The latter model is applicable to molded plastics since it accounts for the molecular orientation effect that occurs during the molding process.

Boundary Conditions

Prior to the application of boundary conditions, the equilibrium equations will be singular, that is, the stiffness matrix K will not be positive-definite and its inverse cannot be found. However, the stiffness can be rendered positive-definite by application of a suitable set of displacement boundary conditions. The resulting "reduced" set of equations will be determinate (that is K will be positive-definite) provided all possible rigid-body displacement modes have been removed. In practice, provided the finite element model is free from internal releases (for example pins) and the elements themselves do not contain any spurious zero-energy modes, then an admissible set of displacement constraints is any set that provides a finite (greater than zero) resistance to each of the six possible rigid-body movements of the structure.

To categorize available forms of displacement constraint, it is useful to imagine that the movements corresponding to each degree of freedom at a constrained node are resisted by external (that is linked to ground) springs. The three most common forms of displacement boundary condition are then:

- Rigid (displacement is zero, spring stiffness is infinite).
- Semi-rigid (displacement non-zero, spring stiffness finite).
- Prescribed (displacement takes a non-zero prescribed value, no spring).

There are two alternative ways of dealing with a rigid constraint. The first method is simply to remove the corresponding equation from the overall set. The second method is to replace the requisite leading diagonal coefficient of K , namely ki⊠, by a coefficient, say ⊠⊠, that is large enough to uncouple the constraint equation from the remaining set. At the same time, the ith component of the force vector P should be set to zero. The latter technique is easily extended to the prescribed displacement case by simply setting the ith component of P to:

pi=⊠⊠ui

where ui is the required displacement. The advantage of retaining the full (original) equation set is that the information needed to calculate reactions at constrained nodes is directly available during the back-substitution phase.

Applied Loads

The generalized load term P is called an equivalent load vector because it accounts for the effects of distributed loads such as pressures, tractions, body forces, initial strains or initial stresses, as well as concentrated loads applied directly to the nodes. In the former case, a system of nodal forces can always be found that is in equilibrium with the specified distributed loading. For simple elements, these forces will be statically equivalent to the applied load (that is they may be found from statical equilibrium conditions alone), but in other cases a detailed knowledge of the element formulation is required to determine their values. The contribution of any distributed loads to P is evaluated automatically by the analysis.

Small Deflection Analysis Solution

Once the boundary conditions and applied loads are known, the nodal displacements U are found from:

KU = P

In practice, it is unnecessary and inefficient to invert K when solving the above equation. Instead the above equations are solved by Gaussian elimination.

(a) Bandwidth, **(b)** Semi Bandwidth, **(c)** Profile, **(d)** Skyline, **(e)** All Zeros Outside Band.

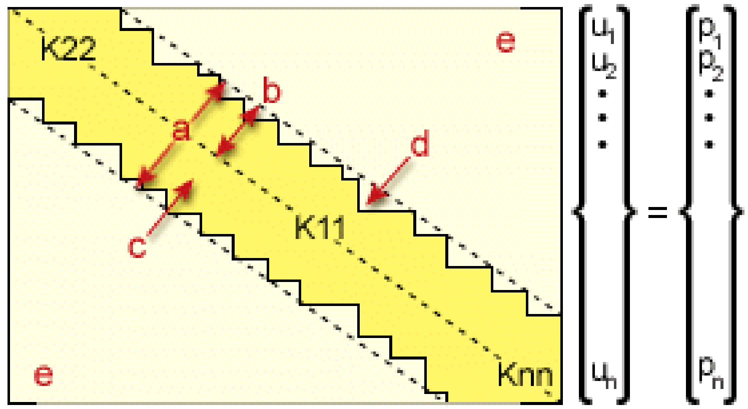


Figure 1: Banded Stiffness Matrix

The stiffness matrix K, has three properties that make Gaussian elimination very efficient:

- The matrix is symmetric (that is, the entries on one side of the main diagonal are "mirror images" of those on the other. The main diagonal is the imaginary line from the top left to the bottom right entries.
- Most terms in the matrix are zero.
- In the matrix all the non-zero terms lie within a narrow band centered about the main diagonal.

Figure 1: Banded Stiffness Matrix on page 6 illustrates the appearance of the stiffness matrix. The boundary between the (mostly) non-zero terms and the all zero terms is termed the skyline. The set of coefficients between the main diagonal and the skyline is referred to as the profile. Since the matrix K is banded and sparse, processing time is decreased by only performing elimination within the profile. The symmetry of K further reduces the amount of calculation since the part of K yet to be eliminated remains symmetrical any stage in the elimination, and so calculation of the known symmetric terms is not required.

For maximum efficiency, the number of coefficients within the profile is first minimized using a nodal re-numbering scheme (the re-numbering is entirely transparent to the user). The bandwidth optimization algorithm adopted is due to Gibbs, Pool and Stockmeyer [1].

Often, the number of equations involved in an analysis is too large to fit all the coefficients in virtual memory (for historical reasons we say “core” for virtual memory). To solve such large systems of equations, an “out-of-core” solver is used in which the full matrix is kept on secondary storage (usually a hard disk drive). Thus, if necessary, the full matrix is divided into blocks and the Gaussian elimination carried out with only two blocks resident in “core” at any one time (one block is being reduced while the second block contains the coupling coefficients). The minimum block size is equal to the maximum semi-bandwidth of K. The maximum block size is machine dependent, but a value that is appropriate to achieving optimum efficiency will automatically be assigned by the program (the highest efficiency usually coincides with the use of a minimum number of large blocks).

Multiple Load Cases

Stress and Warp analysis give the option to specify more than one load case for the analysis, that is different shrinkage strains or external load cases. These multiple load cases can be solved very efficiently because the structure stiffness matrix K is formulated once and then used to determine the displacements U for each load case. Each load case appears as a row in the Job Progress table written to the result file of the analysis.

Analysis Output

Once the displacement field is known, the strain-displacement and stress-strain relations of each element are used to determine strain and stress levels at various positions within the element. In FENAS, these positions will automatically be selected so that they coincide with the numerical integration stations used when integrating over the element volume to find the element stiffness matrix. The use of numerical quadrature stations as stress/strain recovery points is optimum or near-optimum.

Apart from echoing the input data, the analysis output file includes reactions at constrained and initially displaced nodes, nodal displacements and element stresses.

By definition, all these quantities are a linear function of the applied loading—thus, for example, if the loading is halved, the displacements, and stresses will also be halved. This allows you the freedom to choose any nominal intensity of loading without affecting the qualitative validity of the results. Care must however be exercised when extending this concept to include the influence of material constants. If the structure is externally loaded, the strains and displacements will be inversely proportional to the material constants (for an orthotropic material, to maintain proportionality it will be necessary to hold the ratio of the material moduli constant), but stresses will remain constant. On the other hand, if an unconstrained structure is subject to “internal” loading, such as that caused by temperature changes or free shrinkage, then the stresses will be proportional to the material's mechanical properties. The displacements will remain constant. Finally, for a problem involving both internal and external loading, both displacements and stresses become dependent on the material moduli.

Results interpretation

The recommended procedure to use once the first small deflection solution has been obtained is to check that the results are sensible. The best way to do this is by looking in detail at the displacement field. This is easily done using the graphical post-processing facilities.

The questions to ask are:

- Does the pattern look sensible?

- Are the magnitudes roughly what you expect?
- Are any expected symmetries being reproduced? In general, such symmetries can only be reproduced if the geometry, loading and boundary conditions are all symmetric. Note that if the finite element mesh is itself non-symmetric, then any expected symmetry in the results will rarely be reproduced exactly.

Once you are satisfied that a consistent displacement field has been obtained and that the average stress levels make sense in relation to specified material data and other parameters, it will be necessary to decide whether or not a second analysis using a finer mesh should be undertaken. The reason why, in general, the solution for two different mesh densities is required is simply that the finite element method always leads to an approximation to exact or fully converged solutions. By implication, an objective measure of accuracy can only be arrived at by looking at the changes that occur in the results for the two meshes.

To expand on this important point, in a stress analysis suppose you wish to achieve an accuracy of at least 95% in the predicted peak level of von-Mises stress, and have obtained two values of this stress corresponding to the two different meshes. If the change in stress for the two meshes is less than 5% then, providing convergence from this point is assumed to be monotonic (which will generally be the case if the target accuracy lies roughly in the range 95%-100%), then the target accuracy has been exceeded for the finer mesh. On the other hand, if the change is greater than 5%, then further finer meshes must be used and the process repeated until the target is achieved. Note that when the solutions for only two meshes are known and the finer mesh passes the test, it is impossible to say whether or not the coarser mesh would also pass (to answer this question it is necessary to analyze a third mesh that is coarser than either of the original pair).

Against this background it should be clear that there is really only one situation in which a second finer mesh solution might be deemed unnecessary. This is when you have sufficient experience based on previous finite element analyses of similar problems to be confident that the mesh you used is adequate from the point of view of the precision required.

Relevance of Small Deflection Analysis

There are three main reasons why small deflection (as opposed to large deflection) analysis is a valuable tool in its own right:

- Small deflection analysis offers the fastest and most effective way in which useful and consistent information about the mechanical behavior of a structure can be obtained.
- Small deflection analysis provides an excellent tool for building up an understanding of and insight into a wide range of “structural mechanics” problems. Moreover, until some understanding of the small deflection response has been gained, you will have no chance at all of understanding the corresponding large deflection response.

- Small deflection analysis provides the only cost-effective way in which to decide on mesh density and mesh distribution. Because non-linear analysis can be viewed as a series of linearized steps each of which is governed by a different set of starting conditions, it is generally accepted that accuracy checks made on the basis of a small deflection convergence study are approximately valid in the large deflection range as well.

References

1. Gibbs, N.E., Poole, W.G., and Stockmeyer, P.K., “An algorithm for reducing the bandwidth and profile of a sparse matrix”, S.I.A.M. Journal on Numerical Analysis, Vol. 13, No. 2, 1976, pp. 236-250.

Large deflection Stress analysis

In practice, most structures will exhibit a linear or approximately linear response only over a restricted range of load intensities. At higher loads the stiffness of the structure can alter significantly, leading to a non-linear response.

As a simple, but nevertheless realistic illustration of non-linearity, consider a slender column of length *L* subject to a horizontal force *H* and a downward vertical force *V* at the top (bottom is clamped, top is free). Ignoring the small downward movement of the tip, the primary effect of such a load system will be to deflect the top of the column laterally by an amount, say u. If we now consider the structure and loads in this displaced configuration, it is immediately apparent that the bending moment *M* at the column base is now:M=H⊠+V⊠

The fact that *M* is not simply a function of the external loads but depends on u as well, shows us immediately that the problem is non-linear. Evidently, the effect of the vertical load acting on the bent column will be to further increase the lateral displacement, so that direct solution of the problem is not possible. We can however get close to the true solution by dividing the applied loads *H* and *V* into small increments and progressively building up the load to the required level.

As each new increment of load is applied, we can calculate the lateral stiffness of the column tip in its current displaced configuration and then solve to find the corresponding increment in *u* using⊠i=1ki⊠Hi where:

- *i* is the increment number and
- ki is the current lateral stiffness of the column tip.

If we now draw a load-deflection graph taking Hi as the load and ui as the displacement, a curve will be obtained whose slope at the origin is equal to the initial linear stiffness, k0, of the tip, that is,k0=3⊠⊠L3 where *E* is the elastic modulus and *I* is the second moment of area. As the load increases, the slope of the load-deflection graph decreases. This is referred to as a softening response (*Figure 2: Load-Displacement Plots for Non-linear Response* on page 11, left). Usually, if the loading creates compressive stress (as in this example), the non-linear response will be of the softening type, whereas, if it creates tensile stress (for example, a laterally loaded beam whose ends are not free to move inwards), a stiffening response (*Figure 2: Load-Displacement Plots for Non-linear Response* on page 11, right) will generally ensue.

Left softening response, **Right** stiffening response, **P** Nonlinear Load, $\mathbf{u_i}$ Displacement Plots

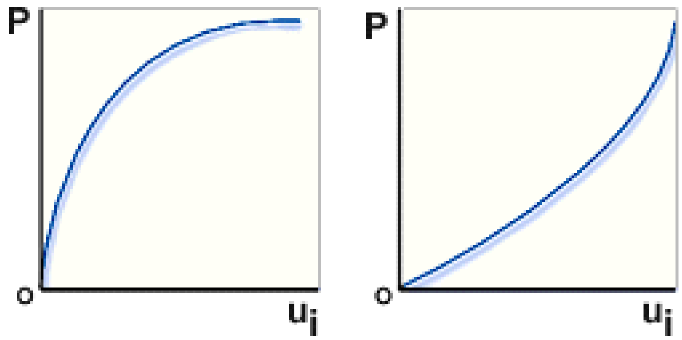


Figure 2: Load-Displacement Plots for Non-linear Response

In the above example, we used an approximate method of following a non-linear load-deflection path—in this method, the true or exact response could only be found by taking infinitely small steps. To understand the basis of a more exact model of non-linear response, it is necessary to recognize that the structure must be in equilibrium at each point on a true load-deflection path. In fact, what is really happening is that as the load is increased, the structure takes on successive configurations that are uniquely determined by equilibrium alone. For this reason, any curve drawn in load-displacement space is more correctly referred to as an equilibrium path. In a structure with many degrees of freedom, the various coefficients of the stiffness matrix will vary at differing rates as the intensity of load varies. Consequently, the equilibrium paths obtained by plotting different displacement components against the applied load may look quite different. The seemingly arbitrary selection of suitable components to plot is forced on us because the full (generalized) non-linear response actually leads to a surface in an *(N+1)* -dimensional load-displacement space (here, *N* is the number of free displacement degrees of freedom of the structure).

Equilibrium Equations

The basis of the large deflection analysis model used by the analysis is a set of incremental equilibrium equations spanning the full set of nodal degrees of freedom of the structure. These equations can be written:Kt+K□U=Rext–Rint orKt□U=Rres where:

- Kl is the linear stiffness matrix for the current configuration.
- K□s the initial stress or geometric stiffness matrix for the current configuration.
- U is the vector of incremental nodal displacement.
- Rext is the vector of nodal forces equivalent to the current level of externally applied loads.

- Rint is the vector of nodal forces equivalent to the internal stress field.
- Kt is the tangent stiffness matrix for the current configuration.
- Rres is the vector of residual (unbalanced) nodal forces in the current configuration.

These equations are non-linear and cannot be solved directly. Instead we can employ any of the classical predictor-corrector iterative techniques (for example, Newton-Raphson or quasi-Newton) where the left-hand side is used as the predictor and the right-hand side as the corrector. During a normal (successful) set of iterative cycles, the configuration of the structure will converge towards the true equilibrium configuration and simultaneously the residual out-of-balance forces, Rres, will become arbitrarily small. In practice, the iterations are terminated when a target accuracy (defined by the program or by you) is achieved. Note that during equilibrium iterations the load level is normally held constant.

Solution Methods

For each solution step, the non-linear equilibrium equations are linearized around the current configuration of the structure. The associated linearization errors result in residual unbalanced forces at each node of the finite element mesh. Iterations of the Newton-Raphson (NR) type are therefore required to reduce these unbalanced forces to acceptably small values. This process is called equilibrium iteration. The full Newton-Raphson algorithm involves reformation and factorization of the global stiffness matrix at the beginning of each iterative cycle. Although this generally results in optimum convergence rates, it is nevertheless very costly. To alleviate this problem, a number of more cost-effective strategies can be used. A total of six such strategies (including NR) are available in the analysis. They are:

KSTRA=0: Initial stiffness method

In this very cheap but rather crude method, the initial linear elastic stiffness K is used for all loading increments and each iterative cycle within an increment (*Figure 3: Initial stiffness method* on page 13). Because the prediction of displacement increments is always based on linearization of the stiffness matrix around the initial geometry, convergence can be very slow and usually the algorithm will fail (that is, diverge or fail to converge within a specified number of iterations) as soon as any significant non-linearity in the equilibrium path is encountered.

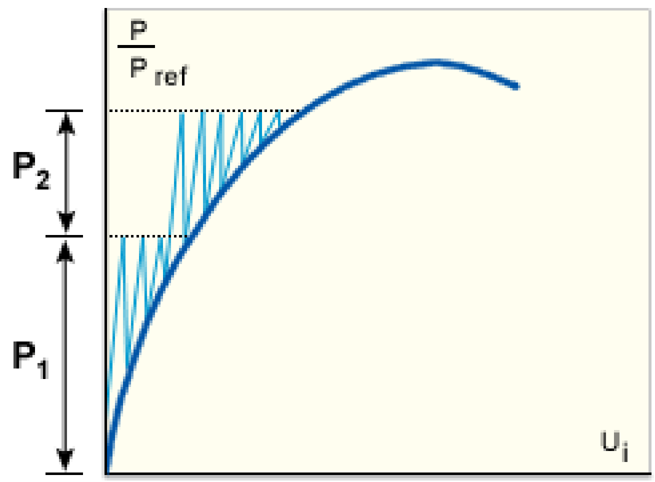


Figure 3: Initial stiffness method

KSTRA=1: Modified Newton-Raphson (MNR)

Here the structure stiffness matrix is updated and factorized at the beginning of each loading increment, that is, at the equilibrium configuration obtained at the end of the previous step (*Figure 4: Modified NR method for softening structure* on page 13 and *Figure 5: Modified NR method for stiffening structure* on page 14). Iterations are then carried out without reforming the stiffness. This strategy is well suited to those parts of the equilibrium path that are mildly non-linear.

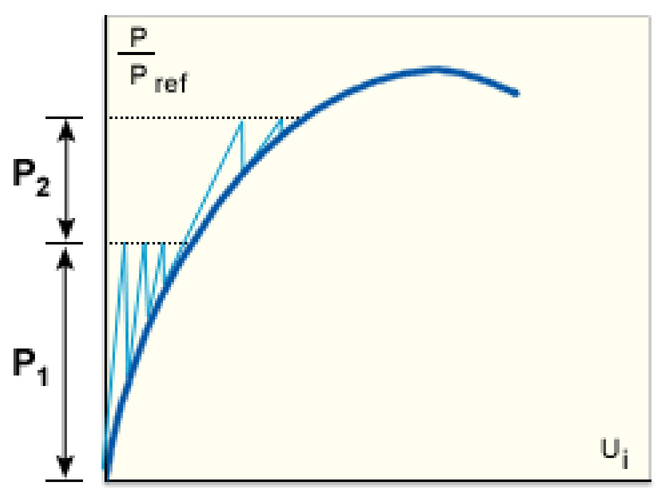


Figure 4: Modified NR method for softening structure

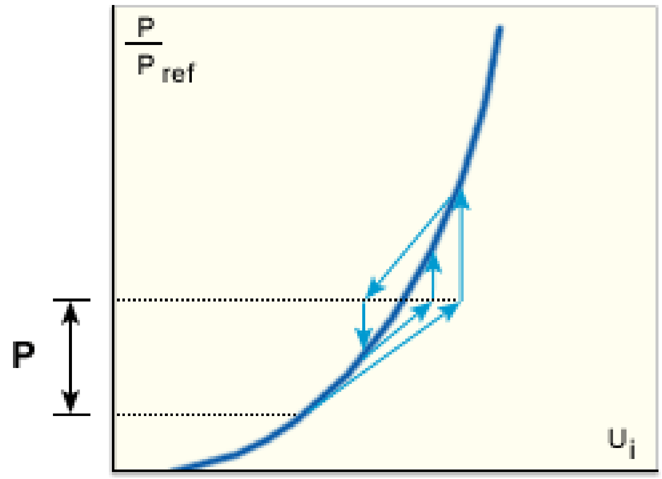


Figure 5: Modified NR method for stiffening structure

KSTRA=2: Combination of NR and MNR

This is a modification of the MNR method in which the stiffness is reformed and factorized at the beginning of the step and again after the first iteration (*Figure 6: Combined method for stiffening structure* on page 14). The algorithm can cope with moderate non-linearities.

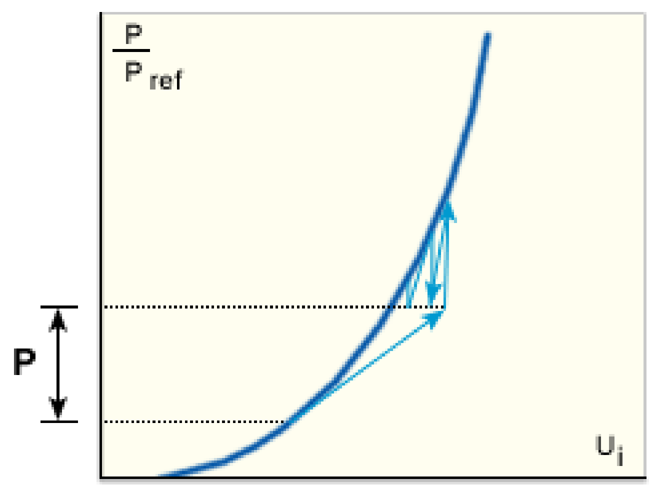


Figure 6: Combined method for stiffening structure

KSTRA=3: Newton-Raphson (NR)

In this case, the structure stiffness matrix is updated at the beginning of each iterative cycle (*Figure 7: Full NR approach* on page 15). The method exhibits rapid (that is approaching quadratic) convergence and is suitable for dealing with strong non-linearities and bifurcations in the equilibrium path. However, for a given number of iterations, it is clearly the most time-consuming of the alternatives considered so far.

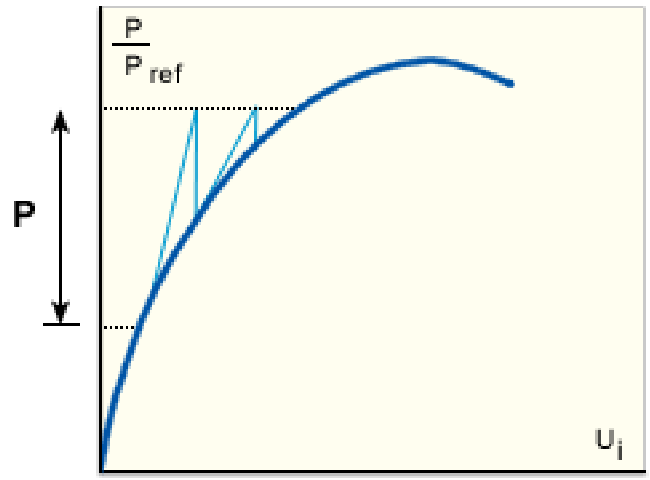


Figure 7: Full NR approach

KSTRA=4: Newton-Raphson (NR) with step reduction

This strategy is identical to KSTRA=3 but is introduced so that when convergence difficulties are encountered using KSTRA 3, the step will be retaken using a quarter of the step size while maintaining pure NR iterations.

KSTRA=5: Load stepping

In this method, equilibrium iterations are temporarily suppressed. Thus, for each load increment, only one reformation and reduction of the structure stiffness is required and no iterations are used (*Figure 8: Straight forward load stepping* on page 16). To minimize drift from the true equilibrium path it is essential that the load increments are much smaller those used in MNR or NR and that the unbalanced forces are carried forward (as opposed to being discarded) into the next step. The method is only used as a last resort when all other strategies have failed.

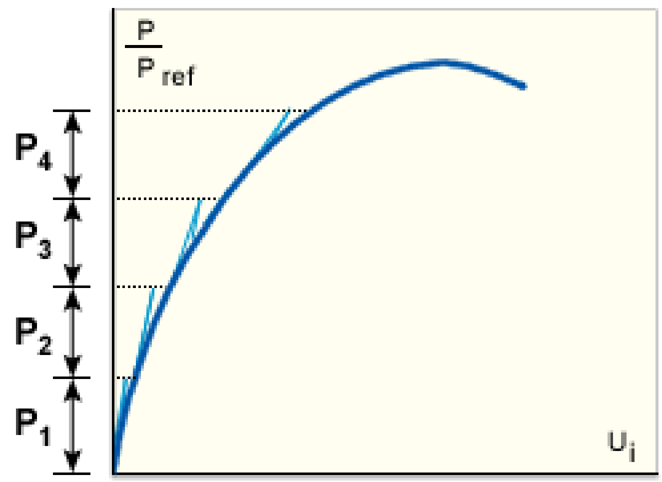


Figure 8: Straight forward load stepping

Equilibrium paths can contain limit points and bifurcations. In the vicinity of such points, the structure stiffness matrix can become ill-conditioned causing even the NR algorithm to fail. However, in such a case it is sometimes possible to successively traverse the difficult section by using load stepping before reverting back to NR. An important feature of the implementation of all the above techniques is that any residual out-of-balance forces remaining at the end of a given step (these will never be identically zero because you must set a target tolerance that is greater than zero) will always be carried forward to the beginning of the next step. In this sense the methods are self-correcting. Compared to methods that discard any residual out-of-balance forces on the basis that they are negligible, the self-correcting scheme leads to higher precision and ensures that there is no accumulative drift away from the true solution path (note that discarding outstanding residuals leads to errors that are tolerance dependent and also accumulate step-by-step). As explained in the section Equilibrium Equations on Page 23, during a successful set of iterative cycles, both the out-of-balance forces and the iterative change in the nodal displacements will tend to zero as the structure settles into its equilibrium configuration. Consequently, the iterative process can be terminated by applying a convergence tolerance to either the residual forces, or to the iterative changes in displacement, or to a combined (energy) measure. The analysis bases its convergence on the following displacement criterion:▯iU<RTOL where ▯i is the Euclidean norm of the change in nodal displacements between two successive iterations, that is:▯i=Ui–Ui–1 and U is the Euclidean norm of the current total nodal displacements. The appropriate value of RTOL to use is somewhat problem-dependent, but values that lie in the range 0.01 to 0.0001 can be expected to provide adequate precision.

Automatic Control Techniques

One of the main difficulties with non-linear finite element analysis is that no single iterative method is best suited to the entire solution path. When the non-linearity of the path becomes more severe, the selected strategy may fail to converge, making further progress impossible. Clearly what is needed is an automatic control system that is capable of retaking a step that has failed using either a different step size and/or a different strategy. Such a scheme is available in the analysis and has been found to be very successful in allowing solution paths to be traced automatically without your intervention. The features of the scheme are as follows:

- You select an initial minimum strategy (normally KSTRA=1 or 2) and an initial step size. This strategy will be maintained until convergence difficulties (if any) are encountered.
- Convergence difficulties are deemed to occur when:
    - convergence has not been achieved in the maximum permitted number of iterations Imax (the default value or specified by you), or
    - the solution is diverging (a divergent solution is assumed if, for iteration 4 or higher, the norm of the current displacement increment, ▯Ui , is greater than the norm of the first displacement increment of the step ▯U0 . Euclidean norm of the out-of-balance forces is greater than the Euclidean norm of the applied loads).
- The solution methods are arranged in order of increasing values of KSTRA. When convergence difficulties are encountered, the program will retake the step using the next higher strategy. Simultaneously the step size is reduced to a quarter of its previous value. The new strategy is now maintained until further convergence difficulties occur in which case the next higher strategy is selected and the step size is again reduced.
- The process will continue until either the maximum number of steps is reached or the load exceeds an allowable level (both these parameters can be adjusted by you).
- Once any strategy that is higher than that originally selected is in use an attempt will be made to return to the next lower strategy provided:
    - four steps of the current strategy (KSTRA = 1, 2, 3 or 4) did not require more than half of the permitted number of iterations, or
    - the current strategy (KSTRA = 5) has been used four times.

If the selected lower strategy fails, then the previous method will be used for a further four increments but without additionally reducing the step size.

Another important feature of the control system is that the step size, ▯is adjusted dynamically to reflect the degree of difficulty that was experienced in the previous step. The idea is to ensure that the step size decreases in regions of increasing non-linearity and increases in regions of decreasing non-linearity. To achieve this we use the relations:

☒new=☒pre☒0.4☒maxIpre≤☒max where Ipre is the number of iterations used in the previous step, Imax is the maximum number of iterations, and ☒max is the step size limit (which can be adjusted by you). As an example, assume that ☒pre = 1, Imax = 20, Ipre = 4, and ☒max = 2. Then, from the formula we find that the current step size will be 2.

This also shows that, following a convergence failure, it will take four steps with an average of 4 iterations in each before the original step size (that is, the step size that was in use in the increment that failed) is recovered. Note that when Ipre exceeds 40% of Imax , the step size will contract, albeit relatively slowly.

Large deflection Stress analysis

Use this dialog to specify the Large Deflection settings for a Warp or Stress analysis.

Large Deflection Analysis Solver Parameters dialog

This dialog is used to specify the settings for the solver parameters of a large deflection Stress/Warp analysis.

To access this dialog, ensure that you have selected an analysis sequence that includes Warp or Stress, click (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Warp Settings** or **Stress Settings** page of the Wizard, set the **Warpage analysis type** or **Stress analysis type** option to **Large deflection**, then click **Solver parameters**.

NOTE: All solver parameters have a default value that will be suitable for most analyses.

Apply imperfection to model to assist convergence	If you have previously run a large deflection analysis that failed due to convergence problems, then applying an imperfection to the model may resolve the problem.
Relative convergence tolerance	
Maximum number of load incrementation steps	Enter the upper limit of loading increments (steps) that the program can take in the nonlinear analysis.
Maximum load factor	
Maximum load factor increase per step	
Maximum number of iterations per step	This option specifies a limit on the number of equilibrium iterations that the solver will perform in each step of the nonlinear analysis.
Output stress results at	

Load incrementation method	
Reconstruct stiffness matrix	Specifies the initial solution strategy to be used in the nonlinear analysis.

Manual Load Control Settings dialog

This dialog is used to enter load increment data for a large deflection analysis using the manual loading control method of load incrementation.

To access this dialog, ensure that you have selected an analysis sequence that includes Warp or Stress, click (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Warp Settings** or **Stress Settings** page of the Wizard, set the **Warpage analysis type** or **Stress analysis type** option to **Large deflection**, click **Solver parameters**, set the **Load incrementation method** option to **Manual loading control**, then click **Edit load increments**.

Load factor increment at each step	Use this table to specify the load factor increments that the solver will apply when using the manual loading control method in the nonlinear analysis. A load factor of 1 corresponds to the total load applied to the part.

Manual Displacement Control Settings dialog

This dialog is used to enter displacement data for a large deflection analysis using the manual displacement control method of load incrementation.

To access this dialog, ensure that you have selected an analysis sequence that includes Warp or Stress, click (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Warp Settings** or **Stress Settings** page of the Wizard, set the **Warpage analysis type** or **Stress analysis type** option to **Large deflection**, click **Solver parameters**, set the **Load incrementation method** option to **Manual displacement control**, then click **Edit displacement increments**.

Model Imperfection for Large Deflection Analysis dialog

This dialog is used to specify a model imperfection to be applied to the model to assist the convergence of a large deflection analysis.

To access this dialog, ensure that you have selected an analysis sequence that includes Warp or Stress. Click (**Home tab > Molding Process Setup panel > Process Settings**), and if necessary click **Next** one or more times to navigate to the **Warp Settings** or **Stress Settings** page of the Wizard. Set the **Warpage analysis type** or **Stress analysis type** option to **Large**

deflection, click **Solver parameters**, select the **Apply imperfection to model to assist convergence** option, and then click **Edit settings**.

NOTE: Only apply a tiny imperfection to the model, otherwise the change in geometry will affect the warpage prediction. The recommended scaling factor to use is one of the following:

- 1/1000 of the part length expressed in mm, or
- 1/10 of the part thickness expressed in mm.

Create imperfection in model by applying	Specifies the source and magnitude of the imperfection to apply to the model, to assist the large deflection analysis in converging.

Convergence problems in large deflection analysis

Convergence problems can arise in a large deflection analysis for many reasons.

When this occurs, you will see the following messages in the large deflection analysis job progress table:

* CONVERGENCE FAILURE *

or

* DIVERGENCE OCCURRED *

When you see these warnings, look at the left column of the table, which shows the different strategies (KSTRA) used during the analysis. The strategies are described in the section Solution Methods.

If the last value of KSTRA in the table is in the range 0-4, then the solution will be satisfactory. If it is 5, it means the structural or warpage analysis programs have given up equilibrium iterations, and the results will probably be unreliable.

You can assess the accuracy of the solution by plotting a load-deflection graph (request a load deflection history when reading in results). After a few steps in strategy 5, the graph will usually become erratic, indicating that the solution is not accurate. Strategy 5 is used because, for some non-linear problems, the program may recover after a few steps at strategy 5 and return to lower strategies.

When using the “load control” load incrementation method, the analysis cannot traverse a limit point. As the point is approached, the analysis takes smaller and smaller steps and will attempt higher non-linear strategies. After reaching the limit point, the analysis will probably remain in its highest strategy (KSTRA=5).

Although in many cases you can increase the “factor controlling maximum step size” to speed up the analysis, where there is a true limit point in the load path it is wise to limit the steps to about 5%, otherwise the analysis may overshoot the limit load. It is easy to spot a limit point by tracing the history of a relevant node. The slope of the load-deflection graph will approach zero at a limit point.

If such a situation (prolonged increments in strategy 5) occurs, it doesn't always indicate a limit point. It could indicate a problem with the analysis itself. To find out, follow this procedure:

1. Run a small deflection analysis (if you haven't already). Check that the response is reasonable. A modeling error (or unreasonable shrinkages in the case of a warpage analysis) could cause failure of the non-linear solution.

2. Check your constraints. In a warpage analysis, if you have over-constrained the model so that the shrinkage strains are in conflict with constraints, this will often cause the large deflection analysis solution to fail, even if the linear result is reasonable.

3. Look at the load-deflection graph of some relevant nodes. If there is a true limit point or simply a very non-linear region, then the slope of the graph should gradually decrease until it is nearly horizontal. Alternatively, look at the deflected shapes of several steps just before convergence trouble occurs (this will require you to read several results). A highly non-linear region in the load path (that is buckling of the plastic part) usually shows up as a significant change in shape over a small change in load.

4. Sometimes, if the load steps are too large, the gradual decrease in load-deflection slope is not clear (especially if a limit point was overshot). To look in more detail, re-run the analysis as follows:

1. View the analysis log and examine the Job Progress table. Note the load level (RFAC) at the step before the strategy (KSTRA) becomes 5, call this value RFAC*.
2. Now re-run a structural or warpage analysis, but this time force the analysis to take smaller steps in the region of RFAC*. For example, if trouble occurs at RFAC* = 0.55, then type in a series of steps like the following as load factor increments:

   0.1 , 0.1 , 0.1 , 0.1 , 0.1 .
3. Now set the “Maximum load factor increase per step” to about 0.005. When the analysis takes over the load stepping (after RFAC = 0.5), all steps will be limited to a maximum of 0.005. Then repeat step 3 above, which should show the response in detail near RFAC*.

If this investigation shows that the solution failed because of a highly non-linear region in the load path, then this is strong evidence of a buckling problem with your plastic part. In fact, this is stronger evidence of a problem than any result obtained from the buckling analysis. Usually, however, you would use the buckling analysis for design purposes because it is considerably faster.

Buckling analysis

Whilst the full non-linear incremental/iterative method of following the response of a structure is completely general and relatively precise, it can also involve a great deal of computational effort.

Because of the fundamental importance of buckling, and its design implications, a simplified method that provides an approximation to the critical load level at which buckling can be expected to occur, will clearly be valuable. It turns out that such a method can be devised provided we assume that the prebuckling response is linear and that the effect of prebuckling displacements is negligible. The method, which we call buckling analysis, is also referred to as initial stability or classical bifurcation analysis.

The following sections describe the Generalized buckling analysis method and then the two specific methods used by the analysis.

Generalized Buckling Analysis

In buckling analysis, the aim is to determine the critical buckling load for some known distribution P of applied loading. By definition then, we can write: Pcr=□r□ref where

- Pcr is the critical level of the applied load distribution,
- Pref is an arbitrary level of the same load distribution (reference load), and
- □s a scalar multiplier.

With these definitions, buckling occurs when the load multiplier □reaches a critical value □□ The starting point in buckling analysis is the assumption that each coefficient of the stiffness matrix Kt varies linearly with the applied load. As described above, we can think of the applied load as some parameter (say □multiplied by a constant vector of forces Pref.

Given two known states of the structure, (Kt□, □ and Kt□,□), and our assumption of linearity, the stiffness matrix at any given equilibrium configuration, Kt□is given by:Kt□Kt□+□□□−□□Kt□−Kt□

If we define the fraction term as μ , and the change in stiffness from □ to □ as □Kt , then:Kt□Kt□+μ□Kt At a buckling point, there are two equilibrium configurations, U and U+□U , both at the same load level.

This is illustrated in the Figures below, which shows the equilibrium configurations for the two basic types of buckling (these are discussed in more detail later).

(a) load, **(b)** Deflection

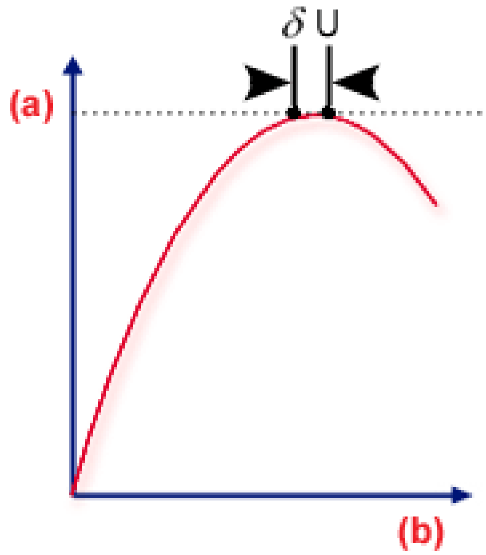


Figure 9: Limit point

(a) load, **(b)** Deflection

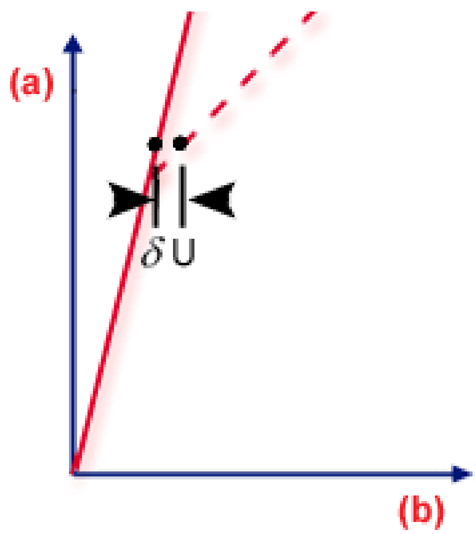


Figure 10: Bifurcation point

Using the basic equation from linear analysis, KU=P, we can therefore write:Kt□U=Pref Kt□U+□U=Pref where U are the displacements corresponding to loads P, and □U denotes the infinitesimal) displacements between the two equilibrium configurations. Subtracting the first equation from the second gives:Kt□U=0 where Kt is a function of □From linear algebra, we know that solving the above equation is equivalent to solving:det□Kt=0 Hence, buckling occurs when the determinant of Kt is zero. The equation to be solved for the generalized buckling problem is therefore:det□Kt□+μ□Kt=0 which is an eigen-problem, in which μ is an unknown to be found. The above equation can be solved by standard methods (for example subspace iterations).

By rearranging the definition of μ, we find:□□+μ□−□ which tells us the load multiplier required to cause buckling. The analysis uses two simplifications of this general method, which we call Linear (Classical) Buckling Analysis and Linearized Buckling Analysis.

Linear (Classical) Buckling Analysis

In this method, we choose □ = 0 and □ = 1, that is take zero and full applied load as the reference states. In this case, μ reduces to □and the equation becomes:det□Kt0+□Kt1−Kt0=0

Updated Lagrangian theory allows us to split Kt into two components: Kt=KL+KG where KL is a first order stiffness matrix and KG is a higher order stiffness matrix (also called stress or geometric matrix). KG is a linear function of material stresses, □

If the part is initially unstressed, then □=0 means that KG0 = 0. Thus:det□KL0+□KL1+KG1−KL0=0

The key assumption made in the Classical method is that the response up to □=1 is purely linear, that is stresses and KG1 will be evaluated using original coordinates. Another assumption made in the classical method is that the first order part of the stiffness does not change with load, that is KL0=KL1. In Total Lagrangian terminology, this is equivalent to neglecting the so-called “displacement-matrix” effect. So the above equation reduces to:det□KL0+□KG1=0

The linear buckling method is used for structural or warpage analyses not based on initial conditions from warpage. Experience has shown that Classical Buckling analysis of warpage problems gives an accurate prediction of the buckling load. The Classical method works well because there is very little change of shape prior to buckling, that is KL0=KL1 is a good approximation.

Linearized Buckling Analysis

In this method, we choose □ = 0 and □ very close to □, that is take zero and a very small fraction of the load as the reference states. Since only a very small step is taken, equilibrium iterations are not required and the analysis performs the step using strategy 5.

We also assume that KL□=KL□. Note that in this method the stresses are evaluated using updated coordinates.

The equation now becomes:det□Kt□+μ□KG□−KG□=0

The simplest and fastest way to analyze pre-stressed components is to take □ = 0 and □ = 0.001. Thus only one step need be taken, and because no equilibrium iterations are done, the cost of the solution is only slightly greater than the cost of the Classical Method.

The linearized buckling method is used exclusively for structural analyses based on initial conditions from a Warp analysis. The classical method cannot be applied to these problems because there is significant residual stress from processing. This violates the assumption that KG0 = 0. Instead, the linearized buckling method must be used. This is indicated by Stress when you select initial conditions buckling analysis.

Buckling analysis

Use this dialog to specify buckling settings for a Warp or Stress analysis.

Buckling Analysis Solver Parameters dialog

This dialog is used to specify the settings for the solver parameters of a buckling Stress/Warp analysis.

To access this dialog, ensure that you have selected an analysis sequence that includes Warp or Stress, click (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Warp Settings** or **Stress Settings** page of the Wizard, set the **Warpage analysis type** or **Stress analysis type** option to **Buckling**, then click **Solver parameters**.

NOTE: All solver parameters have a default value that will be suitable for most analyses.

Stability and buckling

When a structure is loaded up incrementally, it is quite likely that at a certain level of load, the configuration of the structure will start to alter considerably faster than before.

This can easily be recognized on the associated load-deflection path, as a section where there is a definite change of slope and/or curvature. Normally such a change will be of the softening type, that is, the slope decreases and the displacements grow more rapidly than before, accompanied by a noticeable alteration in shape. Such behavior is referred to by the general term buckling. Buckling falls into two main types, namely, the buckling of perfect and imperfect structures respectively.

Buckling of Perfect Structures

As the load increases, a perfect structure initially follows its primary (or fundamental) equilibrium path, AB in *Figure 11: Stability and buckling: straight column* on page 26, *Figure 12: Stability and buckling: flat plate* on page 27 and *Figure 13: Stability and buckling: cylindrical shell* on page 27. During this stage, the stiffness of the structure with respect to its weakest direction (that is, lateral or out-of-plane direction) is being steadily degraded. At point B, called a bifurcation point, the structure buckles by following the (current) line of least resistance. This form of buckling can be viewed as the transition from a stiff membrane (axial) dominated path to a flexible bending dominated path. Because slender beams and thin-walled plates and shells are much stiffer axially than flexurally, it is natural that they are the most prone to buckling.

The paths BC in *Figure 11: Stability and buckling: straight column* on page 26, *Figure 12: Stability and buckling: flat plate* on page 27 and *Figure 13: Stability and buckling: cylindrical shell* on page 27 are called the secondary or postbuckling paths. The stability of the structure following bifurcation is characterized by the slope of the secondary path at B. Thus, the equilibrium states at B are neutral (*Figure 11: Stability and buckling: straight column* on page 26), stable (*Figure 12: Stability and buckling: flat plate* on page 27) and unstable (*Figure 13: Stability and buckling: cylindrical shell* on page 27).

In the unstable case, any arbitrarily small increase in load will cause the structure to jump instantaneously through to an adjacent stable equilibrium configuration (for example, the chain dotted path BD in *Figure 13: Stability and buckling: cylindrical shell* on page 27). This phenomenon is called snap-through buckling. Note that the alternative postbuckling path, depicted by the solid curved path BCD in *Figure 13: Stability and buckling: cylindrical shell* on page 27, traces the theoretical equilibrium states that would occur if snap-through is prevented, and load is progressively removed until a stable path is regained at point C.

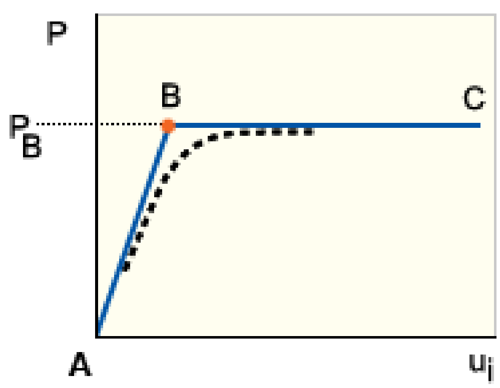


Figure 11: Stability and buckling: straight column

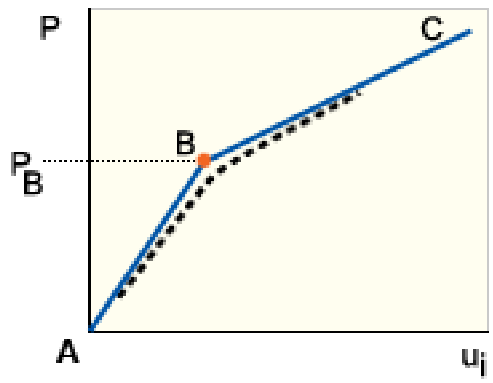


Figure 12: Stability and buckling: flat plate

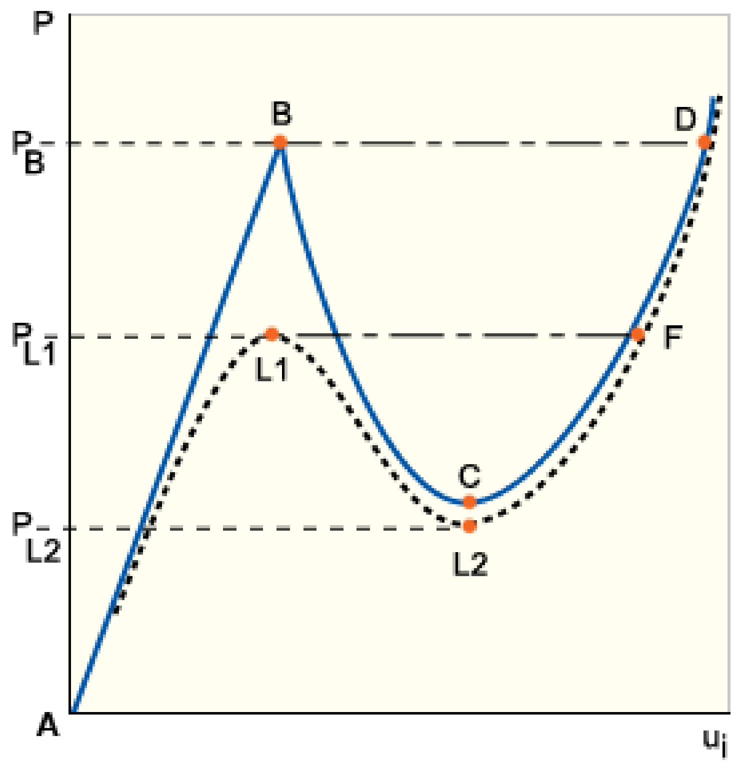


Figure 13: Stability and buckling: cylindrical shell

Obviously, from a designer's point of view, a stable postbuckling path is desirable and, provided displacements remain acceptable, it is not unusual to utilize the increase in postbuckling strength that is available. On the other hand, an unstable equilibrium path will rarely be acceptable, particularly as it will normally be accompanied by dynamic snap-through. Furthermore, structures that exhibit unstable buckling are often imperfection sensitive, that is, the maximum or critical load capacity is sensitive to quite small changes in the magnitude and shape of initial imperfections.

Buckling of Imperfect Structures

A real structure can never be perfect. Even minor disturbances of geometry, loading, boundary conditions or material properties will usually prevent true bifurcations from occurring. Instead of the primary and secondary paths intersecting to form a slope discontinuity, both paths curve locally to form one continuous and unique solution. The resulting postbuckling path is depicted by the dotted lines in *Figure 11: Stability and buckling: straight column* on page 26, *Figure 12: Stability and buckling: flat plate* on page 27 and *Figure 13: Stability and buckling: cylindrical shell* on page 27. As the magnitude of the imperfections decreases, the more localized will the change of curvature become. Where the perfect structure would exhibit an unstable bifurcation point, the imperfect structure exhibits a limit point (point L1 in *Figure 13: Stability and buckling: cylindrical shell* on page 27).

A limit point is defined as a maximum or minimum on the load-deflection curve, but with no discontinuity. At a limit point, the structure stiffness is zero in the sense that

det Kt=0

Further along the postbuckling path we may encounter a second limit point but this time it will be a minimum on the load-deflection curve (point L2 in *Figure 13: Stability and buckling: cylindrical shell* on page 27). Note that, analogous to the perfect structure, the unstable path L1L2 will not actually be followed. Instead, instantaneous snap-through buckling depicted by the chain-dotted path L1 will occur. Although buckling of a perfect structure normally involves bifurcation, this is by no means always the case. Instead, unstable buckling via a limit point can occur whenever the prebuckling mode shape contains symmetries that cannot be broken because of the imposed boundary conditions. For this reason bifurcation buckling is sometimes referred to as symmetry breaking.

Analysis Considerations

When carrying out a non-linear incremental analysis of a perfect structure that contains symmetries, it is necessary to assign a small imperfection to the structure in order to break the symmetry. If this is not done, the solution is likely to continue along the fundamental equilibrium path instead of branching onto the true postbuckling path. Examples where this may occur are a straight column under co-axial load and a flat plate under in-plane load. The latter example can be realized when performing Warp analysis on a flat part with no Cool analysis results. The recommended procedure in such a case is to build in a small geometrical imperfection whose shape corresponds to the expected buckling mode shape. (If the buckling mode shape is not known, then any out-of-plane shape can be used.)

Critical Buckling Load and Buckled Shape

For problems with very few degrees of freedom, the determinant could be evaluated explicitly, giving a polynomial of order *N* in λ (*N* is the number of unconstrained degrees of freedom).

There will be *N* real roots, and λ_{cr} is simply equal to the smallest (in magnitude) of these roots. Back substitution of λ_{cr} into the buckling equations will give the critical buckling displacement mode. This mode is defined as an arbitrarily scaled set of displacements corresponding to the change of configuration that occurs during buckling. Therefore, if we wish to view the postbuckled configuration of the structure, it is necessary to superimpose the critical displacement mode associated with λ_{cr}, on the undeformed configuration of the structure.

For structures having many degrees of freedom, the determinant expansion followed by root extraction method becomes unworkable. To utilize a more general solution algorithm, it is necessary to identify the buckling equations with a generalized eigen-problem. Associated with this eigen-problem there are *N* real eigenpairs λ_i, U_i, where $i = 1,...N$, and λ_i and U_i are, respectively, the ith eigenvalue and eigenvector representing the ith critical load factor and buckling displacement modes. Powerful, general purpose, methods are available to solve generalized eigen-problems. The method used is called the subspace iteration method [1].

Using the subspace iteration method, we can find a series of say *n* eigenpairs arranged in order of increasing magnitude of λ, that is, $\lambda_1 \leq \lambda_2 \leq \lambda_n$ from which it follows that $\lambda_{cr} = \lambda_1$ and $U_{cr} = U_1$.

The number of eigenvalues to find is a parameter which you can specify during a Warp or Stress analysis. Note however that the computational effort involved increases markedly as the number of eigenpair solutions required increases. Moreover, in a buckling problem, one is normally interested only in the "real" critical state rather than in hypothetical higher-order modes. While it is not uncommon, under increasing load, for a structure to change (usually via bifurcation) from the first critical mode into another higher-order mode, it is quite wrong (and dangerous) to assume that the load level and buckled shape corresponding to the latter mode are meaningfully related to the first mode predicted. The reason is that during the transition from the lower to the higher buckling mode, the configuration of the structure is likely to undergo significant change, thus creating an obvious conflict with the basic assumptions of the buckling analysis.

Sign of Predicted Eigenvalues

Note that there is no restriction on the value of λ_{cr}, both positive and negative values are admissible. Thus, for example, if Pref is selected so that it creates a tensile stress field in a flat plate, then λ_{cr} will be negative. On the other hand, reversing the direction of Pref, so the stress state is compressive, will lead to a positive value of λ_{cr}. By definition, however, both λ values will have exactly the same magnitude. If the reference loading

Pref is reversible, then in practice it suffices to find the first eigenpair alone (this follows because we know that when ⊠⊠ is negative, reversing the direction of Pref will simply reverse the sign of ⊠⊠). A difficulty arises, however, when we consider an irreversible load system. In this case buckling will correspond to the lowest positive eigenvalue, and since it is impossible to tell in advance how many negative eigenvalues there are, we must either proceed on a trial-and-error basis, or do a single analysis in which at least two eigenpairs are requested. Experience to date with irreversible load systems (for example, shrinkage strains acting on a plastic component) indicates that the solution of the first two eigenpairs is usually sufficient (that is the ⊠values are either both positive, or one is negative and one is positive).

Sturm Sequence Check

Using the subspace iteration method, it is occasionally possible to skip one or more eigenvalues, as one or more eigenvalues that actually exist may be missing from the values that are predicted. For this reason, a technique called the Sturm sequence check, is used to check for missing eigenvalues. Unfortunately, the check is not valid if any of the *n* eigenvalues are negative.

Validity of Buckling Result

When conducting a buckling analysis, it is important to bear in mind the limiting nature of the initial assumptions. If there is substantial change in shape before buckling, then the results will become unreliable because the method is being applied outside its range of validity. Usually, this means that ⊠⊠ will be over-estimated, but this is not always the case. If there is reason to believe that the prebuckling displacements of the structure are not small, then the buckling analysis should always be followed by a full large deflection analysis.

Types of Buckling

There are two basic types of buckling, both satisfying the equation det⊠t=0:

- Limit point.
- Bifurcation.

(a) load, **(b)** deflection

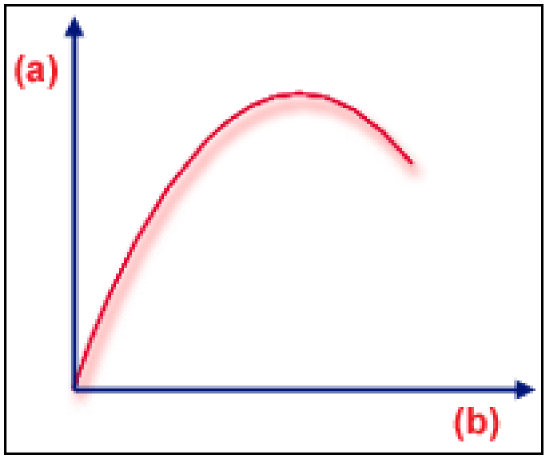


Figure 14: Limit point

(a) load, **(b)** deflection

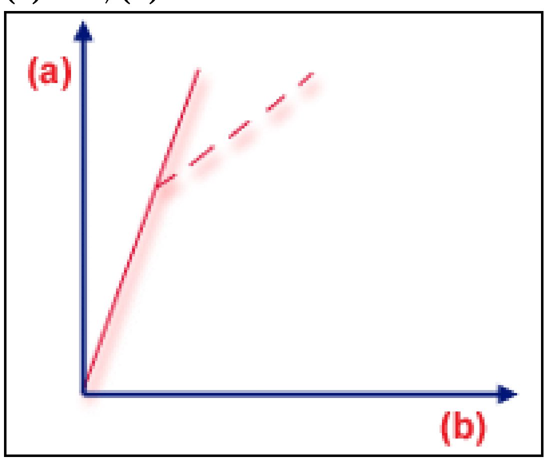


Figure 15: Bifurcation of Perfect Structure

(a) load, **(b)** deflection

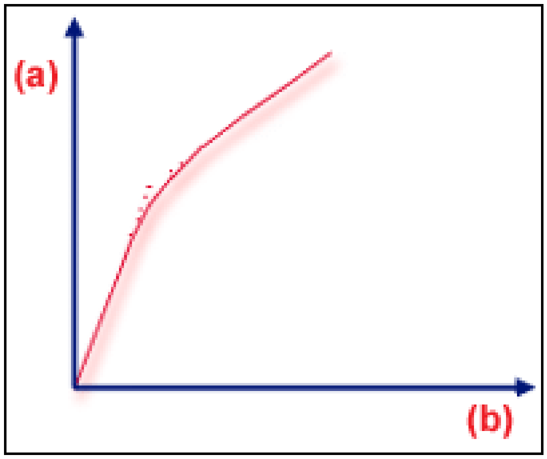


Figure 16: Bifurcation of Real Structure

(a) load, **(b)** deflection

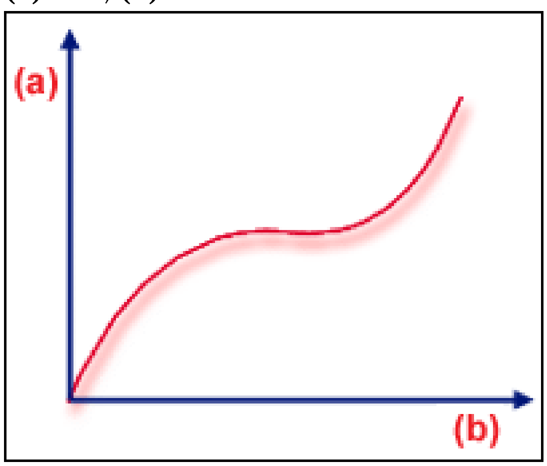


Figure 17: Softening then stiffening

A Limit point, loosely speaking, is a state where the component can accept no more load without a major change of shape (*Figure 14: Limit point* on page 30). This typically occurs in dome-shape components, where a central load may cause the dome to "snap-through". More generally, we expect buckling to be of the limit point type when there is significant geometric non-linearity before the buckling point. In the dome example, the component needs to become fairly flat before the stiffness matrix determinant becomes zero. After the "snap-through" (which may be gentle or severe), the component can accept more load, but generally components must be designed so that expected loads do not reach the limit point.

A bifurcation is a "branching" point where two possible equilibrium states lie close together. If you think of a perfectly symmetrical flat plate, then the response of the plate to an in-plane compressive load is simply linear. However, once the load exceeds a critical value the equilibrium becomes unstable; like a ball at the top of a hill—which way will it roll down? So a real flat plate will buckle at that critical load level. Thus we have two possible equilibrium paths and we call their intersection a "bifurcation point" (*Figure 15: Bifurcation of Perfect Structure* on page 31).

Both these types of buckling can appear in warpage or thermal loading problems, although bifurcation is more common. In the flat plate example, imperfections in the plate cause a blurring of the sharp change of stiffness at the Bifurcation point (*Figure 16: Bifurcation of Real Structure* on page 31). Another common response is "softening followed by stiffening" (*Figure 17: Softening then stiffening* on page 32). In this case, the determinant of Kt falls to some small value, but not zero.

Buckling analysis of cases like *Figure 15: Bifurcation of Perfect Structure* on page 31 and *Figure 16: Bifurcation of Real Structure* on page 31 usually give a good indication of where the bifurcation is. Analysis of cases like *Figure 14: Limit point* on page 30 tend to overpredict the limit point. In problems like *Figure 17: Softening then stiffening* on page 32, there is no clear-cut buckling load, however the buckling analysis will often indicate where the non-linearity starts to become severe.

References

1. Bathe, K.J., Finite Element Procedures in Engineering Analysis, Prentice-Hall, Englewood Cliffs, New Jersey, 1982, pp. 666-696.

Modal frequency analysis

Modal frequency analysis is used to define the natural, undamped frequency response of a structure.

Theoretically it is similar to the buckling analysis, however the physical interpretation of the results is different.

The dynamic response of a structure may be represented by the equation: $M\{\ddot{U}\}+C\{\dot{U}\}+K\{U\}=\{F\}$ where:

- M is the mass matrix,
- C the damping matrix,
- K the stiffness matrix,
- $\{F\}$ the vector of forcing functions which may depend on time and
- $\ddot{U}$, $\dot{U}$ and U are respectively, the vectors of nodal accelerations, velocities and displacement.

If, instead of this general equation, we consider the equation of an undamped and unforced structure we obtain, by setting the damping matrix and vector of forcing functions to zero, the following equation: $M\{\ddot{U}\}+K\{U\}=0$

This equation defines the basic response of the structure and may be used to find the resonant frequencies. To see this, note that a solution to the equation above may be written in the form: $U=\{\phi\}e^{i\omega t}$ where ω is the frequency of vibration.

Substitution of the above equation with the equation before it gives: −[illegible]M[illegible]j[illegible]ei[illegible]j[illegible]+K[illegible]j[illegible]ei[illegible]j[illegible]=0

That is, K[illegible]j[illegible]=[illegible]j[illegible]M[illegible]j[illegible] where [illegible]j[illegible]=[illegible]j[illegible]2

This is a standard eigenvalue problem. It can be solved using the subspace iteration method.

For a structure with *n* degrees of freedom there will be *n* eigenvectors. To each eigenvalue [illegible] there corresponds an eigenvector [illegible] which is often called a mode shape. In practice it is not necessary to determine all eigenvalues. Generally it is sufficient to find the lowest few eigenvalues as these dominate the response of the structure. The Stress analysis program allows you to enter the number of eigenvalues to be found. In addition when a number of eigenvalues are calculated a Sturm sequence check is performed to check that the eigenvalues are consecutive.

Modal frequency analysis

Use this dialog to specify the Modal Frequency settings for a Warp or Stress analysis.

Modal Frequency Analysis Solver Parameters dialog

This dialog is used to specify the settings for the solver parameters of a modal frequency Stress analysis.

To access this dialog, ensure that you have selected an analysis sequence that includes Stress, click (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Stress Settings** page of the Wizard, set the **Stress analysis type** option to **Modal frequency**, then click **Solver parameters**.

NOTE: All solver parameters have a default value that will be suitable for most analyses.

Initial conditions analysis

This Stress analysis is based on the deflected part determined from Warp analysis.

We call this “Initial Conditions Analysis”. The analysis uses both the deflections and the shrinkage strains determined from the Warp analysis, as a starting point for the Stress analysis.

Initial conditions analysis is available for each of the three analysis types described in the preceding sections. In this section, we first describe how the Warp results are passed to the Stress analysis, then present some discussion on how the initial conditions analysis should be used in conjunction with the main analysis types. These include: small deflection, buckling, large deflection and modal frequency analyses.

If we write:

- *G* = model geometry
- *U* = displacements U0 = initial displacements
- *e* = strains e0 = initial strains
- *s* = stress s0 = initial stress

then we can compare the inputs to warpage and initial conditions Stress analysis as follows:

Warp analysis:

- *G = G(mold)*
- U0 = 0
- e0 = *e(shrinkage)*
- s0 = 0

Initial conditions Stress analysis:

- *G = G(mold)*
- U0 = *U(warpage result)*
- e0 = *e(shrinkage)*
- s0 = 0

The key point to note in the above is that initial stresses are not used to carry through the “residual stresses from warpage”. Instead, they are carried through by virtue of U0 and e0 . On the very first step of a Stress analysis, FENAS effectively duplicates the stress calculation that was done at the last step of the Warp analysis. There is no overhead in this because stresses are always calculated at the beginning of a large deflection analysis.

The alternative (simplistic) approach is to pass the warped shape

G = G(mold + U(warpage result))

and initial stresses, s0 = *s(warpage result)* to the initial conditions stress analysis.

The method used, as described above, achieves the same effect but has the following advantages:

- A consistent strain definition (strain=deformation/length) is used in both analyses because the geometry data (*G*) doesn't change. To illustrate this, a warped part was subjected to zero load. The adopted method predicted zero response—there was no movement. This is not the case with the initial stress approach because of the change in strain definition, that is there is an “artificial redistribution” of stresses to cope with the change of definition.
- You can easily scale down the residual stresses from the Warp analysis, by reducing the elastic moduli in the stress analysis. Reduced elastic moduli can be used in the stress analysis to simulate roughly the effect of relaxation of the residual stresses from warpage, and to allow for creep under load (an “effective linear modulus” can be used)).

For example, if for warpage you use E1=1600 and E2=1200 MPa, and for external-loading you use E1=800 and E2=600 MPa, then the stress computed at the start of the first step of the external-loading analysis is half of the residual stress output at the end of Warp analysis. You can therefore pass "reduced residual stresses" through into the external-loading analysis. This would not be as easy to do with the initial stress approach.

- Post-processing is simplified because there is no need to add initial stresses to the stresses from the structural or Warp analysis, to produce the final stress.

Initial Conditions and the Three Analysis Types

This section includes some specific comments on using initial conditions analysis with each of the analysis types.

Buckling Analysis

When a buckling analysis is run, two analyses are performed; a small deflection analysis followed by a buckling analysis.

Generally, the linear (classical) buckling method is used. The initial small deflection analysis is used to determine the part configuration at 100% load. The buckling analysis then uses the known configurations at 0 and 100% load. When post-processing the results, you can examine the deflected shape at 100% load, as well as the mode shape(s) determined from the buckling analysis.

For buckling analysis with initial conditions however, the classical linear buckling theory can not be applied to the problem because the stresses are non-zero before the load is applied. The "Linearized Buckling" Method must therefore be used. In this method, the initial small deflection analysis is used to determine the part configuration at a very small percentage of the load. The linearized buckling analysis then uses the known configurations at 0% and at the end of the small step.

NOTE: This means that when post-processing initial conditions buckling analysis results, the deflected shape at 100% load is not available. You can only display mode shapes.

If you need the linear result, you will need to run a small deflection analysis as well as the buckling analysis.

Since the mode shape determined from the buckling analysis is an incremental mode shape (the change in configurations at the buckling point), it must be superimposed on the warped shape (after suitable scaling).

Large Deflection Analysis

NOTE: The large deflection initial conditions stress analysis should only be performed on large deflection warpage results.

Large deflection Warp analysis results can only be used if the warpage response is known to be highly linear, that is if the eigenvalues from a buckling analysis are very high.

Modal Frequency Analysis

Modal frequency analysis may also use the initial conditions from Warp analysis. Similar comments to those for buckling analysis apply in this case.

Effect of Including Initial Conditions

The stresses from warpage may have a stabilizing or destabilizing effect on the response of the part under load. For example, if the residual stresses cause compression of a slender surface, then this effectively adds to the stress caused by a compressive load. An analysis which does not include initial conditions would over predict the failure load in this case.

Alternatively, if the warpage stresses cause tension, then a compressive load might balance out the residual stress, so that an analysis not including initial conditions would give a too conservative prediction of the failure load.

Prescribed Displacements and Initial Conditions

A useful feature of the software is the ability to prescribe displacements (rather than apply loads) in an initial conditions analysis. This allows you to determine the force(s) required to deform the warped shape into a known configuration. The final result will also show the stresses caused by forcing the part into that configuration.

For each node to which a prescribed displacement boundary condition has been applied, FENAS will output the “reaction force” in the results summary file.

Can Stress Results be used as Initial Conditions?

The following two examples illustrate when it might be useful to prepare an initial conditions analysis based on a Stress analysis:

- A warped part has been forced into shape using a prescribed displacement Stress analysis. You then want to find the response of that part to further external loading.
- External loading has been applied to a part. You then want to find the response of the part to thermal loading.

In this release however, you cannot use structural results as initial conditions for another Stress analysis, for the following reason.

Warp analysis results can be used as initial conditions for the Stress analysis because the constraint reactions from the Warp analysis are close to zero. For an externally loaded Stress analysis, in which the constraint reactions are all non-zero, it is necessary to pass these reactions to the second analysis. If these reactions are not passed, equilibrium requirements may be violated, particularly if the constraints were moved.

In this release, there is no mechanism to pass the constraint reactions from the first analysis to the second. For this reason, the Stress analysis program will not allow you to use a Stress analysis result file as initial conditions.

Local coordinate systems

3

A local coordinate system (LCS) is a set of X, Y and Z axes associated with each node in the model. It is often preferable to use a local coordinate system for assigning constraints and loads to simplify the constraint or load to one direction.

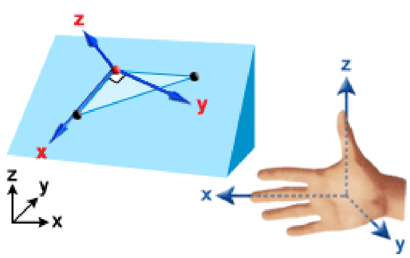
Definition

To define a local coordinate system you need three reference points or nodes in the model. Given three reference locations the X, Y and Z axis is defined as follows:

- Entity 1 defines the origin
- Entity 2 defines the local X direction
- Entity 3 defines the XY plane, with the local Y axis passing closest to entity 3

The local Z direction is defined to be perpendicular to the XY plane, such that the local coordinate system is "right-handed".

A right-handed coordinate system is one where the thumb of your right hand forms the Z direction, your extended fingers form the X direction and the palm of your hand indicates the local Y direction.



Local coordinate systems

A local coordinate system (LCS) is a set of X, Y and Z axes associated with each node in the model.

Defining a local coordinate system

When loads or constraints on the part act in a direction other than the X, Y and Z directions of the global coordinate system, using a Local Coordinate System (LCS) can greatly simplify the settings of such constraints or loads.

1 Select **Geometry tab > Local Coordinate System panel > Create LCS**. The **Create LCS** dialog appears.

2 In the **Filter** drop-down list at the bottom of the dialog, select the type of model entity that you want to snap to when you click in the model pane.

3 For each of the 3 Coordinate boxes in turn, either enter the required coordinates using the keyboard, or click on the required location in the model pane.

4 Click **Apply**.

Activating a local coordinate system

When loads or constraints on the part act in a direction other than the X, Y and Z directions of the global coordinate system, using a local coordinate system can greatly simplify the settings of such constraints or loads.

1 Click (**Select**) and click on the local coordinate system symbol in the model pane.

2 Click **Geometry tab > Local Coordinate System panel > Activate** to activate the selected local coordinate system.

Selecting a local coordinate system

After a Local Coordinate System (LCS) has been created and activated, it must be selected, in preference to the global coordinate system, before its impact can be realized.

1 Click **(Plot Properties > Deflection)** tab and select **LCS_1** from the Reference coordinate system drop down menu. Click **OK**

2 Click **Examine Result** to see the warpage at any given node. Select **local** from the drop down menu at the bottom right corner of the Deflection Query window to switch from the global coordinate system to your LCS.

3 Click **Warpage Visualization Tools** from various aspects. You can select your LCS by clicking **Translation** and then selecting your LCS from the Coordinate system drop down menu. Click **Apply.**

Local coordinate systems

Use this dialog to define a Cartesian local coordinate system.

Create LCS tool

The **Create LCS** tool is used to define a Cartesian local coordinate system by means of global coordinates. Local coordinate systems can be useful when defining constraints and loads that do not act in a global axis direction. To access this tool, click **Geometry tab > Local Coordinate System panel > Create LCS**.

It is not necessary to specify all three coordinates. This function can be used in the following ways:

- Specify only Coordinate 1 to define a local coordinate system with the Coordinate 1 location as the origin and local axes in the same direction as the global axes.
- Specify Coordinate 1 and Coordinate 2 to define a local coordinate system with a defined origin and a local X axis in the direction from Coordinate 1 to Coordinate 2. The Y and Z axis directions will be selected automatically by the program.
- Specify all three Coordinates to define a local coordinate system with a defined origin and defined local axis directions as given by the right hand rule.

NOTE: If you want to define the local coordinate system by selecting nodes in the model, then set the Filter option to Node before clicking in the model display area.

Constraining the model

4

For a Stress analysis, constraints are applied to the model nodes to prevent rigid body motion (global translations and rotations) of the model, in response to the applied loads, whilst not interfering with the shrinkages of the part.

Why constrain the part?

When undertaking the structural analysis, any system of constraints can be used, providing it prevents rigid-body motion. Rigid-body motion is any motion in which the relative positions of all points making up the body remain unchanged.

General rigid-body motion in space involves six components (three orthogonal translations and three orthogonal rotations). This means that the minimum number of constrained degrees of freedom that must be set in the model is also six. In practice, you must decide whether the global coordinate system or a local coordinate system best simulates your perception of the physical situation being modeled.

NOTE: By default, the six degrees of freedom of all nodes in the model are free. To constrain the model, you need to specify which degrees of freedom are to be constrained for one or more nodes.

Model constraint types

The following types of constraints can be set:

Structural constraints	Constrains any required combination of degrees of freedom at node(s). This includes the fixed, pin, and general constraints.
Spring constraints	Applies a spring constraint at node(s).

General constraint

The general constraint is used to constrain any required combination of translational and rotational degrees of freedom at the selected node.

The translational degrees of freedom can be set as fixed, free, or constrained at a certain distance (mm). The rotational degrees of freedom can be set as fixed, free, or constrained at a certain angle (deg).

Fixed constraint

The fixed constraint is used to constrain all degrees of freedom at the selected node.

Fixed constraints are used during structural analyses to prevent rigid body motion of the model in response to applied loads.

Pin constraint

The pin constraint is used to constrain all translational degrees of freedom at the selected node, while allowing rotational movement.

Pin constraints are used during structural analyses to help prevent rigid body motion of the model in response to applied loads.

Spring constraints

In a Stress analysis, springs may be used to simulate discrete elastic supports.

The spring constraint defines the degree of stiffness to translation in the X, Y, and Z plane directions. In order to specify the spring elastic support constraint, you must first have specified a nodal constraint.

The elastic support spring constraint requires that you specify the direction of the spring constraint, the spring constant (N/mm or lbf/in), and the node(s) to apply the spring constraint to. You are required to specify the stiffness of each individual spring. The values and units for stiffness are derived from force/length.

NOTE: By default, the direction specified is a global direction. A spring constraint can be applied in a non-global axis direction by first specifying a local coordinate system at the node(s).

Loading conditions

6

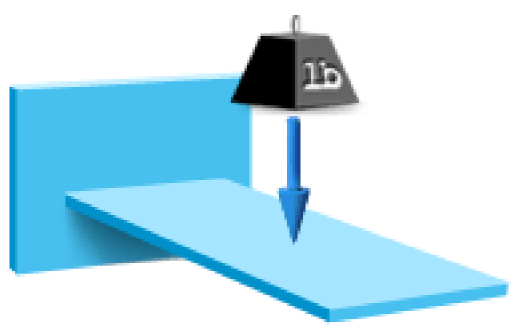
During Stress or Warp analysis, a load, or force, is applied to the part, and the resulting deflections are measured.

There are several different load types that can be applied, depending on the molding situation you want to simulate.

Load Types

The table below lists the types of loads that you can apply to a part. Units for loads must be consistent with the modeling units used.

Type of Load	Metric	Imperial
Force (Point)	N	lbf
Moment	N.mm	lbf.in
Displacement	mm	inch
Pressure	MPa	psi
Surface	MPa	psi
Edge Load	N/mm	lbf/in
Thermal Load	°C	°F
Volume Load	N/cm3	inch



Load incrementation methods

In a large deflection analysis, the total load on the part, whether internal due to the molding process (in the case of a Warp analysis) or externally applied (in the case of a Stress analysis), is applied in a series of steps. This load stepping characteristic of a large deflection analysis allows changes in stiffness of the part as the load increases, that is nonlinear stress-strain behavior, to be predicted and taken into account in determining the deflection of the part under the full load.

There are two ways for this load stepping to occur:

Loading control	The load is applied as load factor increments, that is, specified fractions of the total load.
Displacement control	The load is specified as displacement increments, that is, specified deflections of a specific node on the part up to the final deflection value at 100% applied load.

The following load incrementation method settings are available:

Automatic loading control	Specifies the size of the first load factor increment (the default value is two-tenth of the total load), and then allows the solver to automatically determine the appropriate size for subsequent steps. This is the recommended load incrementation method to use for most cases.
Manual loading control	Specifies the first few or all of the load factor increments to be applied to the part.
Automatic displacement control	Specifies an initial displacement increment to be applied to a control node automatically selected by the solver, and then allows the solver to automatically determine appropriate displacement increments for subsequent steps.
Manual displacement control	Specifies the first few or all of the displacement increments to be applied to the part.
Arc-length loading control	In this method, both load and displacement increments are automatically adjusted based on an arc-length constraint within each increment. This method can best follow a complicated post-buckling path.

NOTE: If you have run a buckling analysis and determined that the part buckles at 6% of load, ensure that the initial load factor increment is less than 6% to ensure the solver doesn't “jump” across the nonlinearity. On the other hand, if a buckling analysis indicates the onset of buckling at 60%, it would be reasonable to take larger steps at the start of the analysis and then let the program choose step sizes automatically at around 50%. It is good practice to do a buckling analysis before any large deflection analysis for these reasons.

NOTE: Regardless of the load incrementation method you have selected, the solver will always increment the load until the **Maximum load factor** value has been reached. If convergence problems arise at a given step, the solver may choose to apply different step sizes or switch to a different load incrementation method.

Load increment methods

Autodesk Warp and Stress analyses provide two load increment methods for large deflection analysis: load control, and displacement control.

Load Control

Under load control, each step corresponds to a particular increment of load. The load increments can be automatically adjusted, or can be specified when preparing the input file for Warp or Stress analysis. During each step, the load is held constant while equilibrium iterations are performed. As the solution approaches a limit point (point of local maximum or minimum load), this algorithm will become unstable and convergence difficulties will occur. Even small unbalanced loads will lead to large changes in the displacements.

In order to successfully cross a limit point and to study the post-buckling behavior, we must abandon the iteration under constant load (load control) method in favor of the iteration under constant displacement (displacement control) method.

Displacement Control

The displacement control method involves the introduction of a displacement based constraint equation, enabling the load parameter ☒to become an additional variable during iteration.

Under displacement control, each step corresponds to a particular increment of displacement for a specific node in the model which we call the control node. As with the load control, the increments (in this case displacement increments) can be automatically adjusted, or can be specified when preparing the input file for Warp or Stress analysis. There are two variations of the displacement control method available: manual or automatic.

With manual displacement control, you select the control node and the displacement direction to be used for the entire analysis. Although this option works well in many situations, it does however lack generality, and breaks down completely in cases where the equilibrium path for your chosen controlled displacement component is vertical or nearly vertical. This is analogous to the break down of the load control method when the equilibrium path is near horizontal.

With automatic displacement control, the analysis applies an initial small increment of load at the start of the first step, to find which node moves the most and in which direction. This node becomes the control node selection and the displacement incrementation method is started. The analysis may change the control node and also the control direction dynamically according to current conditions in the analysis. This has a beneficial effect on both efficiency (convergence rate) and robustness (ability to trace any load path).

Reference 1 contains a detailed description of the automatic displacement control method used in the program. The power of the method is that it is completely general and requires very little prior knowledge of structural response. In principle, by combining the method with the automatic strategy control system described in the section Automatic Control Techniques on Page 26, almost any load-deflection solution path can be traced automatically, including those exhibiting bifurcations, limit points, snap-through and snap-back.

Choice of Control Method

The suitable control method to choose depends on the type of analysis you are running (warpage with prescribed displacements) and the degree of non-linearity of the problem. Both methods have advantages and disadvantages, the most important of which are discussed in this section.

Advantages of Load Control/Disadvantages of Displacement Control

For mildly non-linear problems, load control is the more efficient method because it is less computationally intensive than the displacement control method. A buckling analysis can give an indication of whether a high degree of non-linearity can be expected in the load range that you are analyzing.

The load control method provides better control over the load level at the final step of the analysis. For example, if you set a maximum load factor of 1 in a displacement control analysis, the last displacement step will be the one which just passes the load factor of 1. The final load may be just over 1 (say 1.01) or well over 1 (say 1.2), depending on the displacement step size. If you are interested in the deflected shape at exactly 100% load, this lack of control in a displacement control analysis could be a disadvantage. On the other hand, if you accepted the option to keep temporary files, you could run a restart analysis with the following inputs:

- Specify restart at the final step which exceeded 100% load
- Select the Load Control method
- Specify an initial load increment no greater than that required to reach 100% load
- Specify a maximum load factor of 1

Choosing suitable initial increment and maximum increment values for displacement control analysis is generally far more difficult and less intuitive than for load control. As for load control, increments that are too small can lead to excessive analysis steps, and increments that are too large can lead to inaccurate prediction of non-linear behavior. An exception to this is prescribed displacement Stress analysis using manual displacement control. In this case, displacement increments are more meaningful than load increments and displacement control would be the preferred method.

Advantages of Displacement Control/Disadvantages of Load Control

For highly non-linear problems, displacement control is the more powerful method because it can trace any load-deflection path automatically. In contrast, load control is guaranteed to fail if a limit point exists in the path.

Displacement control is the preferred method for prescribed displacement analysis since displacement increments are more relevant to the problem than load increments.

References

1 Trueb, U., "Stability problems of elasto-plastic plates and shells by finite elements", Ph.D. Thesis, University of London, 1983.

Nodal loads

Loads can be applied to both part nodes and elements. When applying nodal loading conditions to a part, the node(s), direction of the load (force), and the magnitude of the load must be specified.

Nodal loads include

- Point loads, and
- Edge loads.

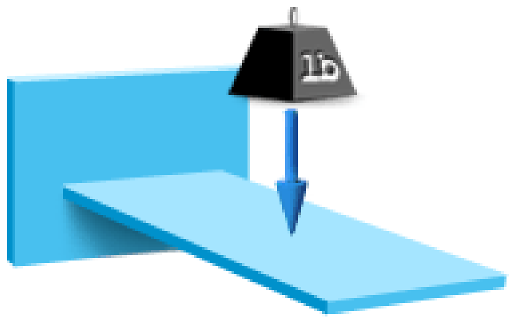
Point loading conditions

To simulate the effect that a load would have on post-molding part stresses, you can set a point load on the model.

Loading conditions are applied in a global coordinate system by default; however, it is often preferable to use a local coordinate system to simplify the load to one direction.

Global Coordinate System

A simple example of a point load acting in a global direction is shown in the following image.



The point is defined by individual nodes. A load of, for example, 0.5 N is applied in the negative Z direction, to the specified node.

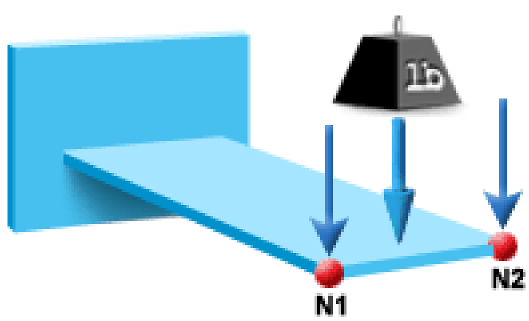
Edge loading conditions

To simulate the effect that a load would have on post-molding part stresses, you can set an edge load on the model.

Loading conditions are applied in a global coordinate system by default; however, it is often preferable to use a local coordinate system to simplify the load to one direction. The following topic describes edge loading using the global coordinate system and a local coordinate system.

Global Coordinate System

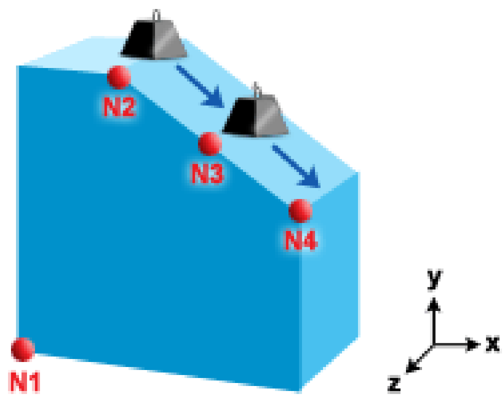
A simple example of a surface edge load acting in a global direction is shown in the following image.



The edge is defined by points N1 and N2. A load of, for example, 0.5 N/mm is applied in the negative Z direction, to the nodes along this edge.

Local Coordinate System

An example of an edge load acting in a local direction (in this case a traction) is shown in the following image.



If the force is not perpendicular to the surface, and does not act in a global direction, you need to define and activate a local coordinate system at a selected node in the model before applying the local edge load.

Elemental loads

Loads can be applied to both part nodes and elements. When applying elemental loads to a part, the element(s), direction of the load (force), and the magnitude of the load must be specified.

Elemental loads include

- Surface loads,
- Pressure loads,
- Thermal loads, and
- Volume loads

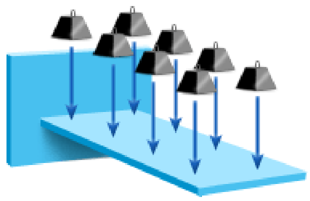
Pressure loading conditions

Pressure loads can be applied to the part model to test whether pressures applied after molding will deform the part.

Pressure loads are applied perpendicular (normal) to elements based on their orientation. A positive pressure value applies the pressure normal to (away from) the top (positive) side of the element. A negative pressure load reverses the direction. For pressure loads in other directions, see surface loads .

Pressure loads

A simple example of a pressure load is shown in the following image. The pressure being applied is a negative value, assuming the side of the cantilevered surface shown is the top (positive) side of the elements.



Surface loading conditions

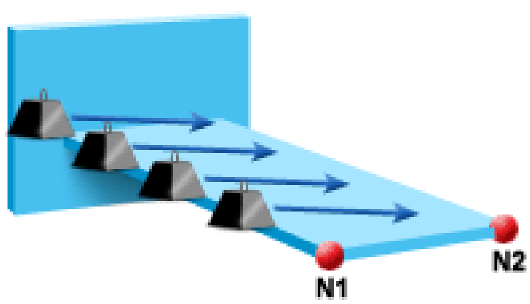
Surface loads can be applied to the model after ejection to predict the part's post-molding stresses or warpage.

Unlike pressure loads, which are always perpendicular to a surface, surface loads can be applied in any direction.

Surface loads

Surface loads are expressed as vectors and are applied in either the global coordinate system, or in a local coordinate system which you need to define and activate at a selected node in the model first.

The following image shows an in-plane traction.



Thermal loading conditions

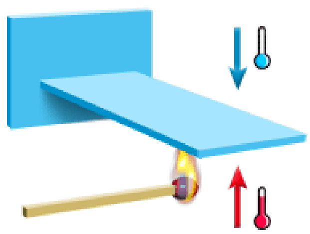
To simulate the effects that an increase in temperature has on part stresses, you can set a thermal load on the model.

Thermal loading

Thermal loads can be applied to selected elements in a model, in either in °C or °F. The thermal load is expressed as the difference in temperatures on the top and bottom sides of the selected elements. The top and bottom sides of mesh elements are defined by the orientation of the mesh. The top side of an element faces in the positive direction of orientation and the bottom side faces away from the orientation direction.

Apply thermal loads to models to simulate a real molding scenario. If you expect certain surface(s) inside the mold to experience increased temperatures, then simulated this by applying a thermal load to the appropriate part model surface(s).

A simple example of a thermal load is shown in the picture below. In this case, a thermal load (temperature increase) of, for example, 30°C is applied to the bottom side of the surface.



Volume loading conditions

In a Stress analysis, a load, or force, can be applied to the part, and the resulting deflections are measured. Volume loading can be used to simulate the effects of gravity in a Stress analysis.

Volume loading

You can apply a uniform load to selected elements in the model, expressed as a volume load (N/cm3 or lbf/in3). You need to specify the magnitude of the volume load and the direction in which it acts (only global directions supported).

The volume load, Pv, is given by the following equation:

Pv=ρg

where

- ρ is the material density.
- g is the gravitational constant.

Density, in units of kg/cm3, multiplied by the gravitational constant in units of N/kg, gives the volume load in units of N/cm3, as is required by this function.

For example: A material with a density of 1.16–3 kg/cm3 would experience a volume load of 1.14E-2 1.14–2 N/cm3 due to the effect of gravity (1.16–3 kg/cm3×9.81 N/kg).