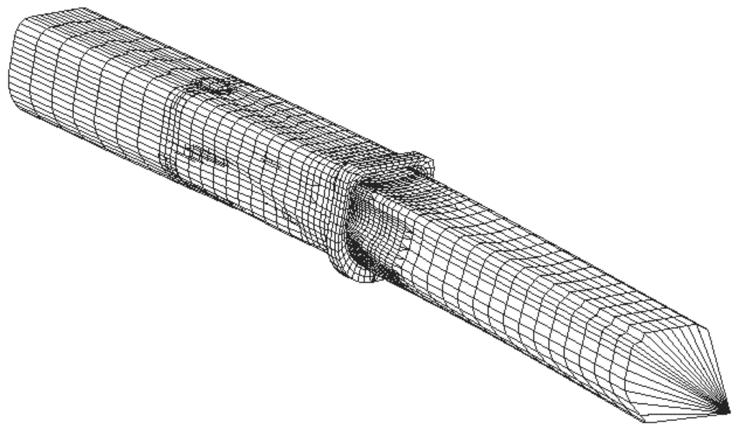
FINITE ELEMENT LIMIT LOAD ANALYSIS OF THIN-WALLED STRUCTURES BY ANSYS (IMPLICIT), LS-DYNA (EXPLICIT) AND IN COMBINATION

Wilhelm Rust

CAD-FEM GmbH, Marktplatz 2, D-85567 Grafing



ABSTRACT

After discussing general properties of implicit Finite Element analysis using ANSYS and explicit analysis using LS-DYNA it is shown when and how quasi-static limit load analyses can be performed by a transient analysis using explicit time integration. Then we focus on the remaining benefits of implicit analysis and how a proper combination of ANSYS and LS-DYNA can be used to prepare the transient analysis by common preprocessing and static analysis steps. Aspects of discretization, solution control, consideration of imperfections and methods of checking the results are outlined.

KEYWORDS

Limit load analysis, stability problems, imperfections, quasi-static dynamic, explicit time integration, plastic material, shell structures, ANSYS, LS-DYNA, ANSYS/LS-DYNA

ANSYS, LS-DYNA, CAD-FEM

ANSYS® is a finite element system developed by ANSYS, Inc., Canonsburgh, PA., USA; the finite element system LS-DYNA™ is developed by Livermore Software Technology Corporation (LSTC), Livermore, CA., USA; the CAD-FEM GmbH is an engineering company with main competence in finite element method and is distributor of both ANSYS as well as LS-DYNA.

STATIC THIN-WALLED SHELL ANALYSIS WITH ANSYS

ANSYS is a general purpose FE-program for static, dynamic as well as multiphysics analysis and includes a number of shell elements with corner nodes only and with corner and mid-side nodes. The implemented bending theory is based on Mindlin kinematics or so-called Discrete Kirchhoff conditions. The behavior of low order Mindlin elements is improved by an „assumed strain“ formulation for the shear strains. Reduced Integration can optionally be chosen for in-plane stiffness. Problems with large rotations as well as with large strains can be handled. Shell elements can be used with a number of material models such as plastic, hyperelastic as well as creep behavior.

Contact analysis is implemented where the user must only specify the two potential contact surfaces. Shell thicknesses (updated due to transverse contraction in large-strain analysis) are included. Penalty and Lagrange methods can be chosen to fulfil the contact conditions. Unprecise discretization and not modeled gaps or interferences can be automatically accounted for by options and parameters. Bonded contact can be used to tie together surfaces with different meshes.

Solutions of nonlinear equations are obtained by a Newton-Raphson method using both direct as well as iterative solvers for the sequence of linear systems of equations.

Last but not least the ANSYS command language (ANSYS Parametric Design Language APDL) includes a lot of elements of a higher programming language. For the application considered in this paper this can be used to translate model and result data into formats to be read by external programs in addition to predefined interfaces.

Buckling and limit load analysis

Limit load investigations of thin-walled structures are usually started with a linear buckling analysis. The results are buckling modes and load factors. Load factors are estimates for an upper limit of the ultimate load; buckling modes show, how the structure will buckle. As it is well known that the structures are most sensitive against imperfections in the shape of the lower buckling modes, they also give an idea of a conservative imperfection and can be applied to the ideal model as geometric (stress-free) modifications which is a simple function in the case of ANSYS.

Buckling and prestressed modal analysis taking into account the current state of loading after a nonlinear static solution is possible to get proper information about the structural behavior. Both can also be used to investigate whether a convergence problem is due to a numerical or due to a physical instability. An example is shown in Fig. 1 (Hanke, Dawani [2]): Unlike in a Zeppelin which have a supporting girder structure the upper hull of the keel airship Cargolifter is a membrane only. The goal of the FE analysis was to determine how far the internal pressure can be decreased due to a leakage before the hull becomes instable. This instability point is indicated by suddenly decreasing natural frequencies (Fig. 2).

Loading control and path-following methods

If imperfections are superimposed on a perfect shell structure, the bifurcation problem usually changes to a non-linear stress problem or a snap-through problem depending on the post-critical behavior. Knowing this behavior is essential for safety considerations. However, in a standard force-controlled analysis approaching the critical load will end up in non-convergence of the solution process; displacement control often is not possible and is only helpful if a characteristic displacement is chosen which is often difficult. Therefore, arc-length methods are preferred as implemented in ANSYS which allow to control the load level together with the length of the displacement increment. This permits to compute the post-critical load-deflection path although force-type loads are applied.

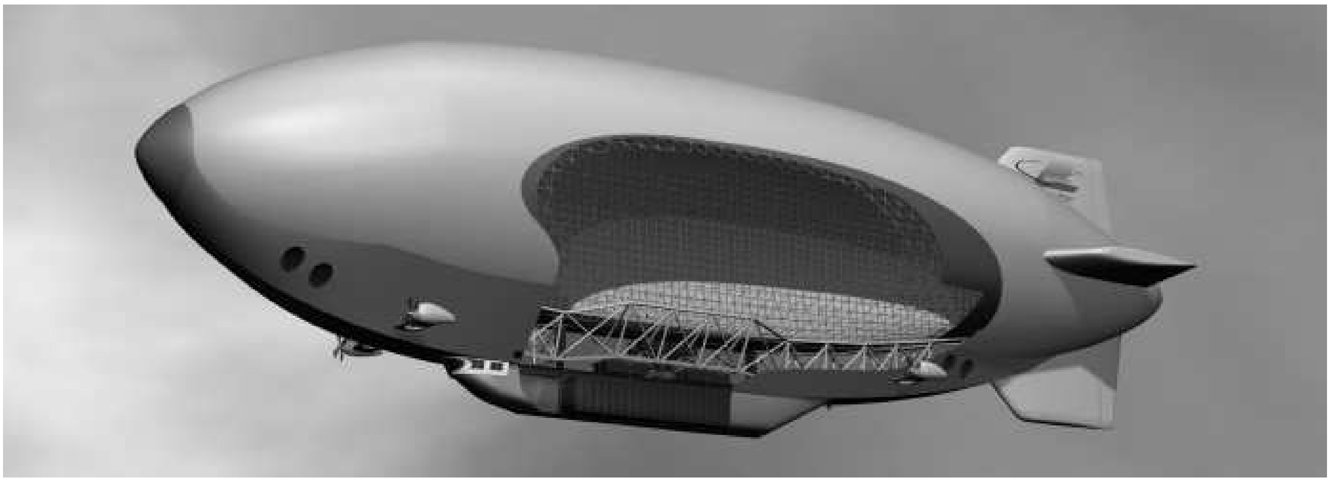


Fig. 1: Construction of the keel airship Cargolifter CL160

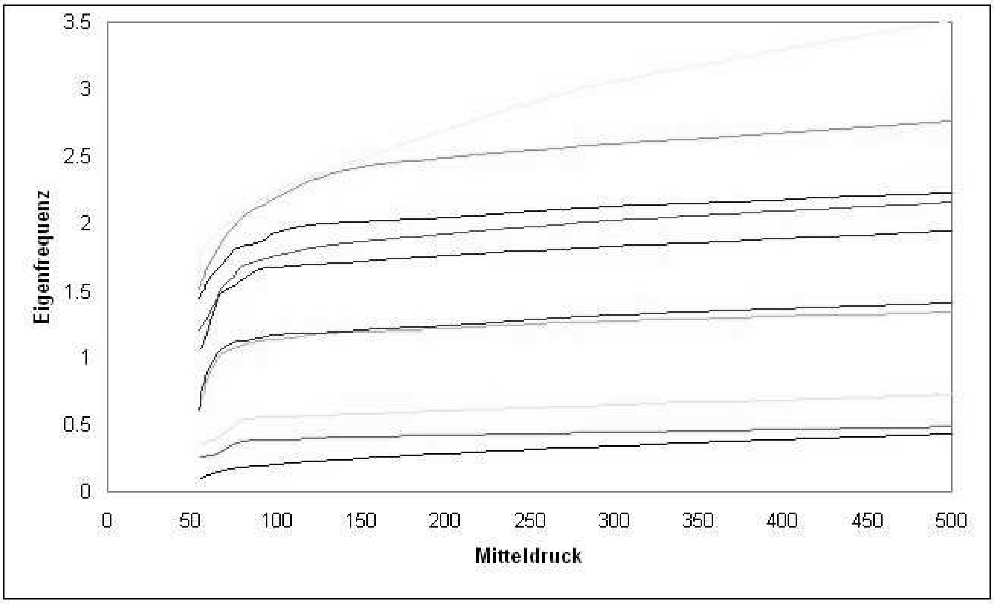


Fig. 2: Dependency of natural frequencies of hull of CL160 from the internal pressure

Possible problems in limit-load analysis

Even when using arc-length methods achieving convergence close to the critical load is often difficult, in particular, if different non-linearities are active. Especially in conjunction with contact difficulties may appear since contact elements change their status (from open to close or vice versa) which is not differentiable for Newton's method. In such cases a lot of effort is needed to achieve proper solution control in order to determine the ultimate load within a sufficient accuracy.

POPERTIES OF LS-DYNA

LS-DYNA in general is designed for transient dynamic analysis of highly nonlinear problems. The essential ingredient determining the solution properties is the use of an explicit time integration scheme which in the case of LS-DYNA is a slight modification of the standard central differential scheme. The equilibrium is fulfilled at the time t_i whereas t_{i+1} is the unknown state. According to Newton's axiom the disequilibrium forces cause accelerations which can be integrated to velocities and displacements. The forces computed with these quantities can be viewed as driving the system towards equilibrium which, however, will never be reached exactly since forces change during the time step. For solution stability and to avoid unpredictable errors a critical time step size based on the so-called Courant-Friedrichs-Lewy criterion must not be exceeded. This time step is determined automatically in a conservative way within the program from the sound speed and the element lengths. These time step sizes usually become rather small for reasonable FE discretizations. However, the solution of the nonlinear systems of equation using this time integration scheme only requires the inversion of the mass matrix within each time step. If a lumped form is assumed and therefore diagonalization each equation is only divided by a scalar. The computational effort is mainly influenced by the formation of the internal forces via the elements and the contact surfaces and can be very efficiently tailored for various computer architectures such as vector and massively parallel computers.

It must be emphasized that considering geometrically nonlinear including large strain problems with contacts with LS-DYNA is not posing any further problem due to nonlinearity. Within the explicit time integration scheme – a simple forward marching scheme - no iteration of nonlinear systems and no convergence control is required. Therefore, no convergence problem can appear.

Besides quasi-static analysis the main applications of LS-DYNA are crash and occupant simulation, metal forming, drop tests and further contact related applications. More recent enhancements are concerned with fluid-structure interaction.

Many LS-DYNA FE discretizations of real industrial problems are dominated by thin shell elements including some beam and solid elements. A large number of element formulations for shells (mainly based on Mindlin theory) is available offering the choice between computational efficiency and improved accuracy. Reduced integration is often preferred in large deformation analyses because of the efficiency concerning the formation of the element vectors and robustness in case of large element distortions. There is a choice of methods and formulations to avoid hourglassing problems.

More than 100 different material models are available to represent many types of highly nonlinear material behavior. The program contains in particular many rate-dependent, viscoelastic, viscoplastic and foam material laws. Many models also offer the capability to consider failure.

Rigid body kinematics is included reducing the description of the body motion – usually any part of the FE model discretized by an FE mesh or other arbitrary geometrical surfaces - to six master degrees of freedom. All other nodes of the considered FE part are coupled to a master via geometric relations accounting for large rotations. The nodes and surface segments of the rigid bodies can be used e.g. to describe contact surfaces.

Advanced robust and computational efficient contact algorithms are the heart of most LS-DYNA applications. Shell thicknesses can be accounted for. Although contact zones can be defined to any level of precision desired, also general contact algorithms are available where the only input is the switch to activate the specified contact algorithm for the whole model or for selected parts only.

Advantages of using LS-DYNA for limit load analysis

The main advantage of LS-DYNA in limit load analysis is the absence of convergence problems inherent to the solution algorithm. Not even arbitrary contact surfaces cause difficulties. Due to status changes in contact the contact forces might oscillate sometimes. This would deteriorate convergence considerably in an implicit analysis but in LS-DYNA analyses this is of minor importance. Due to small time steps amplitudes usually remain within a certain level and the averaged forces remain meaningful.

A further advantage of dynamic analysis is that in the vicinity of a critical point the inertia forces stabilize the system motion even in the post-critical range where the load which the system can carry decreases with increasing displacements. Thus, the character of the post-critical behavior can be studied.

Disadvantages of explicit transient solution in static limit load analysis

The LS-DYNA solution scheme is only applicable to general transient analysis. Thus within the solution always inertia forces, often also damping forces are included. Thus for static resp. quasi-static analyses velocities and accelerations have to be chosen in such a fashion that forces due to inertia and damping remain negligibly small.
In particular, initial conditions must be chosen carefully to avoid oscillations; they should match a static solution very closely and should introduce any motion very smoothly into the system.

The mentioned advantage of not setting up and decomposing a system matrix is a disadvantage within a limit load analysis. Eigenvalue buckling computations or direct detection of stability points cannot be performed.

COMBINING ANSYS AND LS-DYNA

The combination of ANSYS and LS-DYNA is the ANSYS/LS-DYNA suite. It consists of the general ANSYS pre- and postprocessor plus further extensions for specific LS-DYNA features and the LS-DYNA solver. Besides nodes and elements e.g. the LS-DYNA contact definitions, properties for many of the material models, load curve definitions for transient analysis and initial conditions can be prepared within the preprocessor. As in the standard ANSYS FE program the LS-DYNA preprocessing is supported by a graphical user interface. For analysts with some experience with ANSYS there is only a small step towards LS-DYNA.

ANSYS/LS-DYNA in limit load analysis

Since ANSYS and LS-DYNA have elements of comparable theoretical background and thus comparable stiffness it is possible to take advantages of the two programs in a sequential fashion. Once a discretization is modeled for the one type of analysis it is straightforward to switch to the other. A standard application is deep drawing simulation in LS-DYNA and springback or modal analysis with respect to residual stresses in ANSYS, or static pre-stressing of a rotor by ANSYS and subsequent impact simulation by LS-DYNA. For such purposes some ANSYS elements can handle stresses from the LS-DYNA run as initial stresses and LS-DYNA can read predeformations from a file which can be created by ANSYS/LS-DYNA after an ANSYS run.

In case of limit load analysis ANSYS can be used for eigenvalue buckling analysis, for determining and applying imperfections and calculating static initial conditions, whereas LS-DYNA drives the system to the ultimate load and behind. Such a procedure is studied in detail in the following.

REAL-LIFE AND MODEL PROBLEM

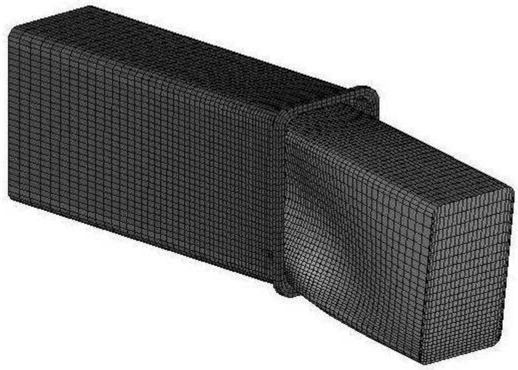


Fig. 3: Simplified model of part of a telescope crane at the onset of buckling

One of the buckling – post-buckling problems solved with LS-DYNA, which was investigated in detail, was the telescope arm of a mobile crane shown on the title page (cf. Kessler/Rust/Franz [3]). At the beginning it was not clear whether the results would be available for publication. Thus, a modified model problem was chosen in addition (Fig. 3). This system was first analyzed using ANSYS only (Bartel [1]).

A look onto the load deflection curve of the industrial system (Fig. 4) shows nearly linear behavior up to the limit load (a). This is typical for optimized designs. A force-controlled analysis will end up in a non-converged solution there, thus only the linear behavior would be visible. The post-critical path (b) gives the most reliable criterion whether a physical or a numerical instability has occurred.

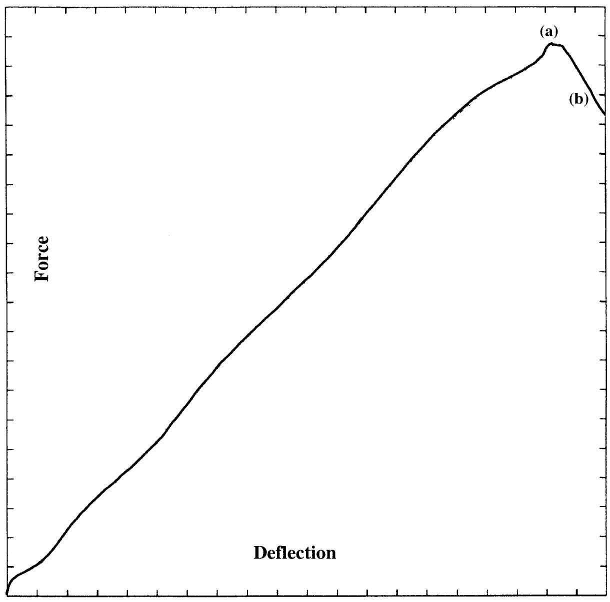


Fig. 4: Load-deflection diagram of the telescope arm of a crane, real system

PREPARING THE SIMULATION WITH ANSYS/LS-DYNA

At first an ANSYS model for implicit analysis is created within the ANSYS preprocessor. It contains shells, some solid elements for the parts between outer and inner tube, some contact areas allowing the tubes to slide and plastic material behavior. At first a static solution at a lower load level was calculated. There an eigenvalue buckling analysis was performed leading to the first buckling mode shown in Fig. 9.

The load factors for the different buckling modes were sufficiently separated so that it appeared reasonable that only the first mode was of interest. The latter was added to the ideal geometry of the model as a geometric imperfection.

In the second stage of the analysis the elements were changed to the corresponding LS-DYNA types, and some specific inputs were created, such as contact and the load-versus-time curves to specify smooth loading conditions as discussed below.

LOADING SPECIFICATION

The loading must be specified in such a way that the computation is as efficient as possible; on the other hand the inertia forces should remain negligible. The first condition requires a high velocity within the process, the second condition requires a small acceleration value. Different ways to overcome this problem are considered.

In general a displacement control is preferable, as then the global motion of the structure can be well controlled. However, the loading consisted of a force F and a moment M at the tip of the part model. Therefore, the LS-DYNA model is extended by a rigid beam (Fig. 9) of the length $e = M/F$ and then the displacement of the end node is controlled.

Constant acceleration

The most direct way of load application is to start with an initial velocity 0 and a constant acceleration, because this leads to nearly constant inertia forces in the model. It is applied to the rigid beam as a linearly increasing velocity of the free node. Whether this acceleration is too high or not can only be seen after the simulation when the ratio of the inertia forces to the total forces has been checked. Therefore, it is recommended to carry out the simulation for a short time only and then consider a modification after checking the static equilibrium.

In the model problem an acceleration of about 28 g led to the result given in Fig. 5. At a first look such a level of acceleration seems to be far away from being static and absolutely too high. However, it must not be compared with the weight of the system but with the limit load and the equilibrium situation there. Fig. 5 also contains a comparison of the results obtained with different element formulations. Since no visible differences can be observed the computationally most efficient element can be chosen.

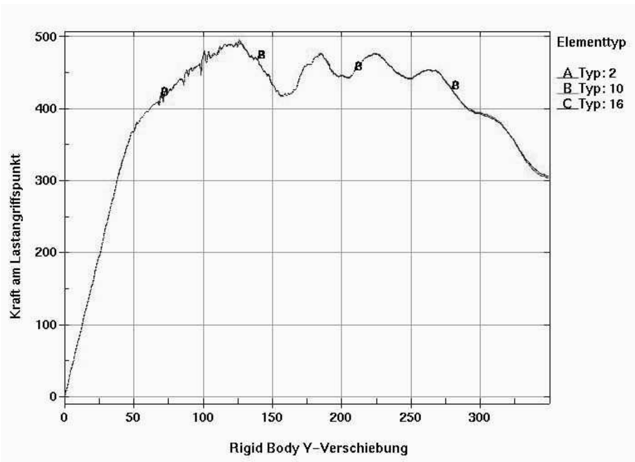


Fig. 5: Load-deflection curve for the telescope crane; model problem; transient analysis

Unlike the real system the curve for the simplified model problem shows two significant points: the limit point and one at 80 % of the ultimate load. It can be noted that buckling and reaching the plastic limit of the shell cross sections is well separated. In this case the plastic limit is well below the elastic limit load. Thus the load-deflection curve of the simplified model problem can be idealized to be a piecewise linear curve; in the real telescope system which is more optimized concerning the plastification the behavior is nearly linear up to the ultimate limit load.

If a displacement control is not possible, e.g. in the case of distributed loads, the acceleration is alternatively determined by the amount of ΔF which is the difference between the static reaction and the applied load. This load increment should be constant. In the case of a nearly linear pre-buckling behavior this can be achieved by a linearly increasing force. Then it is advisable to reduce the loading velocity in the vicinity of the limit load according to the system response. For more complicated pre-critical load-defelection paths the load-vs.-time curve perhaps must be adapted to the system response from a first run, especially when oscillations starts to appear.

An advantage of constant acceleration loading is that it does not need any static precalculations.

Constant velocity

The major disadvantage of the constant acceleration type of loading is that over a larger time range only small displacements are generated resulting in small forces. However, when approaching the limit load, the most interesting point of the analysis, the velocity reaches the maximum and the resolution concerning the states of the results the minimum.

If a constant velocity is applied, the accelerations and the inertia forces result from nonlinear effects only. Thus, high velocity and a linearly increasing displacement can be achieved from the start. This holds under the assumption that the distribution of the initial velocities matches the static deformation as closely as possible. Such a field can easily be computed if a small fraction of the load is applied to the ANSYS model in a static run. If an initial velocity v_{0i} is chosen for the control node i the time increment Δt is known and the vector for the velocity distribution $\underline{v}_0$ can be computed from the displacement field $\underline{u}$:

$$\Delta t = \frac{u_i}{v_{0i}} \quad and \quad \underline{v}_0 = \frac{\underline{u}}{\Delta t}$$

This method is automatically carried out if a static or stationary ANSYS analysis is followed by a transient one. The necessary additional LS-DYNA input is written using an ANSYS macro command sequence.

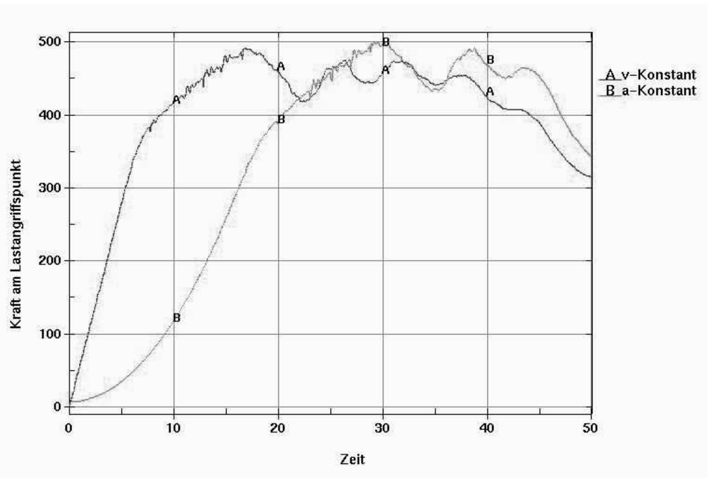


Fig. 6: Load-versus-time curve for A) constant velocity and B) constant acceleration

The procedure described above leads to the better results, the closer the stiffness represented by the ANSYS FE model matches that of the LS-DYNA discretization. The most recently developed ANSYS elements can optionally be used in a formulation being similar to those of LS-DYNA.
The larger is the difference in the formulation, the greater is the danger of getting oscillations. Significant oscillations represent too much deviation from the static solution and can lead to accumulating errors in particular in path dependent problems such as in the case of plastic materials or in the case of friction. Within an initial phase oscillations can be damped out, however, the damping can be (and should be) reduced to zero when approaching the ultimate load.

It is also obvious that the procedure with constant velocity is more sensitive to unprecise contact condition i.e. if contacts being necessary for the equilibrium are not initially closed in the LS-DYNA run. Then the contact closure may be rather sudden and leads to a shock type loading which causes oscillations.

In the model problem a velocity of 7 m/s at the end node of the rigid beam leads to the system response given in Fig. 6. Although the computational cost is reduced by a factor of 1.7 compared with the run with constant acceleration, the result is as good. With the latter procedure the calculated critical load is lower, i.e. probably closer to the static ultimate load. The reason is that the velocity at the time when buckling begins is higher for a = *const*. than for v = *const*. and the motion towards the buckles requires larger changes in the velocities, i.e. accelerations, i.e. inertia forces.

Constant speed with initial displacement

Up to now it was assumed that the LS-DYNA analysis is started with the initial displacement being zero. One additional capability is the „Initialization to a Prescribed Geometry“, where LS-DYNA expects a file containing displacements for all nodes. For preparing this file from the results of an ANSYS run and activating this option an ANSYS/LS-DYNA function is available.

The initial displacement can be taken from a nonlinear static ANSYS analysis at higher load level but in the „well-converging“ range. From this starting point with initial displacements it is easier to achieve the quasi-static limit load. As before an initial velocity distribution is also required for this displacement state which is achieved as described above except that the total displacement $\underline{u}$ is replaced by the displacement increment $\Delta\underline{u}$ from the last two states. This is a secant whereas a tangent is desired. The latter can be best approximated if the last load increment is small, e.g. by applying a small amount in a subsequent load step especially for that purpose.

For this method it is of increasing importance that the response from the ANSYS analysis matches that from LS-DYNA. The danger of obtaining oscillations is slightly higher than in the case of constant acceleration where increasing velocities make constant oscillations negligible after some time.

STATIC CHECK FOR TRANSIENT ANALYSES

Since a quasi-static solution should be obtained by the dynamic analysis executed it must be checked whether inertia and damping forces do not exceed a tolerable level. The comparison of internal and kinetic energy is the easiest way to achieve this, but it can be erroneous, because the latter depends on the velocity which can be high even if no acceleration appears.

If possible the static equilibrium should be checked. The disequilibrium is due to dynamic effects. In the examples shown above the transverse force which can be determined by the program for defined cross sections and the fixed end must be constant, whereas the moments should vary linearly. In a transient analysis time discrepancies in the response curves for loaded and fixed end might appear. In Fig. 7 (left) it is shown that the forces excellently match for the considered sections. In comparison, in Fig. 7 (right) it is shown that significant oscillations in the loading phase may occur due to too fast load application. In Fig. 8 it is demonstrated that for the model problem at *v=const.=7 m/s* tip load and reaction at the fixed end are synchronous whereas at *v=14 m/s* differences near the first significant nonlinearity become clearly visible.

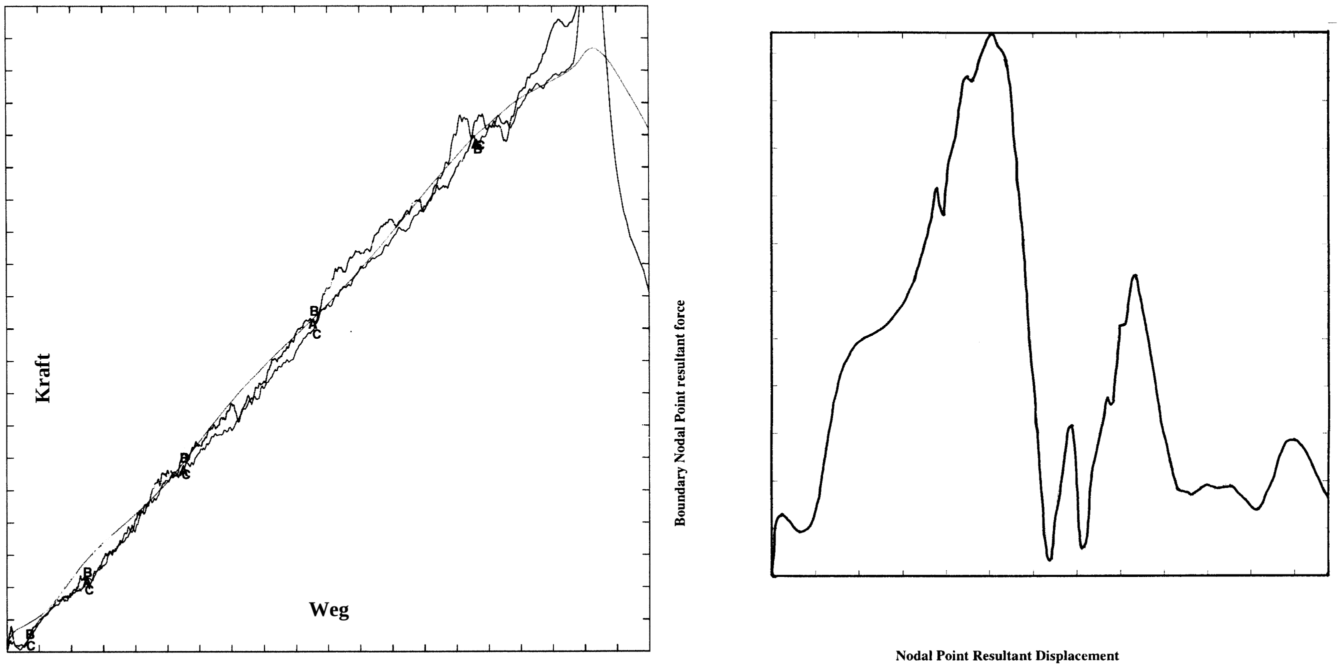


Fig. 7: Telescope arm; real system:
left: Static equilibrium check: force at fixed and free end,
cross section force in the middle
right: Oscillations before buckling due to overly high applied acceleration

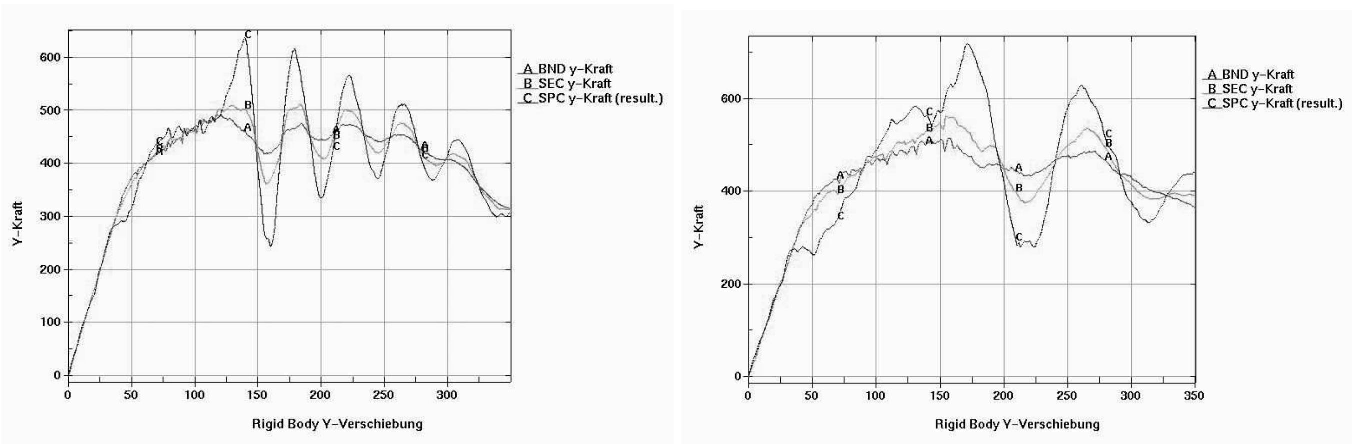


Fig. 8: Force at fixed (B) and free end (A), cross section force in the middle (C)
for *v=const.=7 m/s* (left) and *v=14 m/s* (right)

IMPERFECTIONS

Usually the analysis model of a system is taken with an ideal, perfect geometry. This includes the danger that in a numerical analysis bifurcation points may be missed. In a complex system, however, it is rather unlikely that the characteristic buckling mode never appears but the critical load may be calculated significantly too high and by chance. Furthermore, in reality the limit load and the buckling type often depend on imperfections __(cf. the cylinder buckling example below). If realistic imperfections are known they should be used directly, otherwise a conservative imperfection must be estimated e.g. from an eigenvalue buckling analysis.

Imperfection from eigenvalue buckling

Since the ANSYS analysis of the telescope model led to a buckling mode (Fig. 9) this could be taken as the shape of a geometric imperfection. It must be noted that for eigenvalue buckling the status of the contact elements is frozen. This requires that the contact is well established at the considered load level.
For the scaling of the imperfections the maximum change in a nodal coordinate was chosen to 1/250 of the longer diameter of the main buckle (from inflection point to inflection point). This measure was also taken for the other types of imperfections described below. In the scaled buckling mode there was still space remaining such that no further contact appears between the tubes.

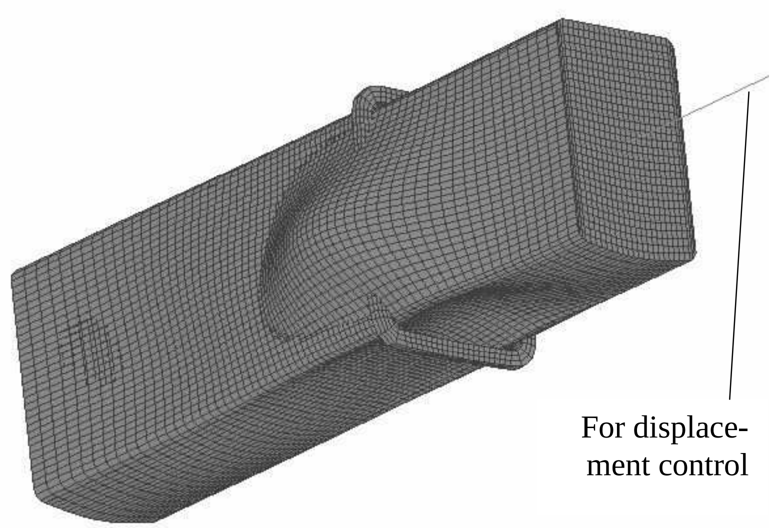


Fig. 9: First buckling mode of telescope arm; simplified model

Arbitrarily distributed imperfection

If no static pre-analysis is desired or it seems too complicated to analyze one, other kinds of imperfections are possible. One type of imperfection is generated by the aid of a random distribution (Fig. 10). It can be expected – as a result from many similar analyses - that the important buckling modes will be initiated by these imperfections and that the mode belonging to the lowest buckling load will govern the load deformation process. For practical purposes the shell normals are averaged at the nodes and the nodes are moved in this direction by the values of an intrinsic random function using an ANSYS/LS-DYNA macro command procedure. In order to avoid excessive warping due to large differences from one node to the other some smoothing may be necessary.

Contact zones should be excluded from adding imperfections to avoid initial penetrations.

Sinusoidal imperfection

Since known analytical solutions often lead to sinusoidal buckling modes, an imperfection of this kind (Fig. 10) can be appropriate provided that the system geometry is regular. The main advantage is that no further smoothing is necessary. The number of half-waves per direction should be set in such a way that the resolution of the sinusoidal shape by the FE mesh is just high enough, i.e. that the discretization of the waves is coarse.

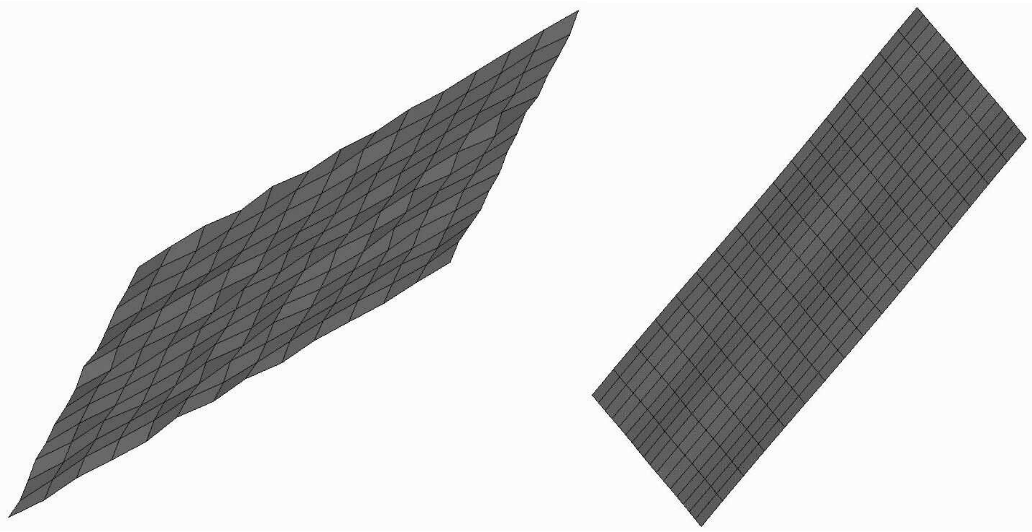


Fig. 10: Random and sinusoidal distribution of imperfections

The results in Fig. 11 indicate that only in the case of the random distribution the limit load is lower than in the other analyses. This seems to be an artificial effect due to the warping introduced by this imperfection.

In total the type of imperfection has little influence on the computed limit load, especially for the model problem. For the real system an increase of the maximum imperfection to twice the value led to a reduction of the ultimate load by 10 %. This appears to be due to the fact that the real system is optimized and therefore more sensitive against imperfections.

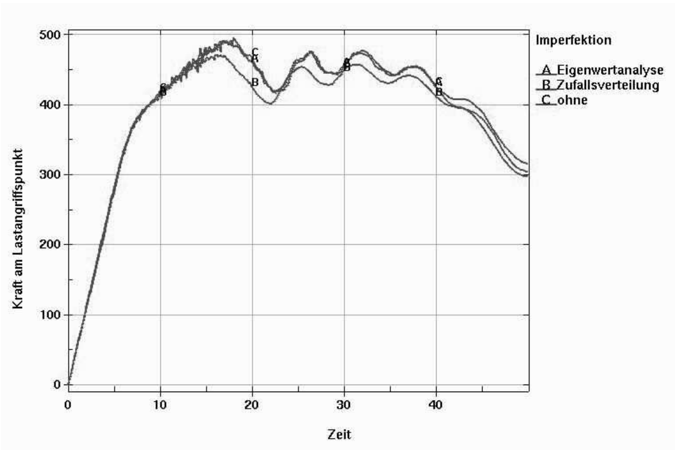


Fig. 11: System response for different types of imperfections:
A) buckling mode, B) random distribution

Dynamic Imperfections

It should be first noted that the transient solution introduces oscillations per se which is also a type of imperfection itself. In Fig. 12 and Fig. 13 two buckling states of the real problem are depicted; the second one is obtained after a slight modification of the wall thicknesses. The same changes in the behavior were observed in experiments, too. Although the imperfection was chosen on the basis of the buckling mode shown in Fig. 12 the buckling mode of Fig. 13 appears, i.e. the „wrong“ imperfection has no fatal influence on the result.

If necessary a further excitation in addition to the static load could be applied. This would be advisable, if the behavior of the structure is not known at all.

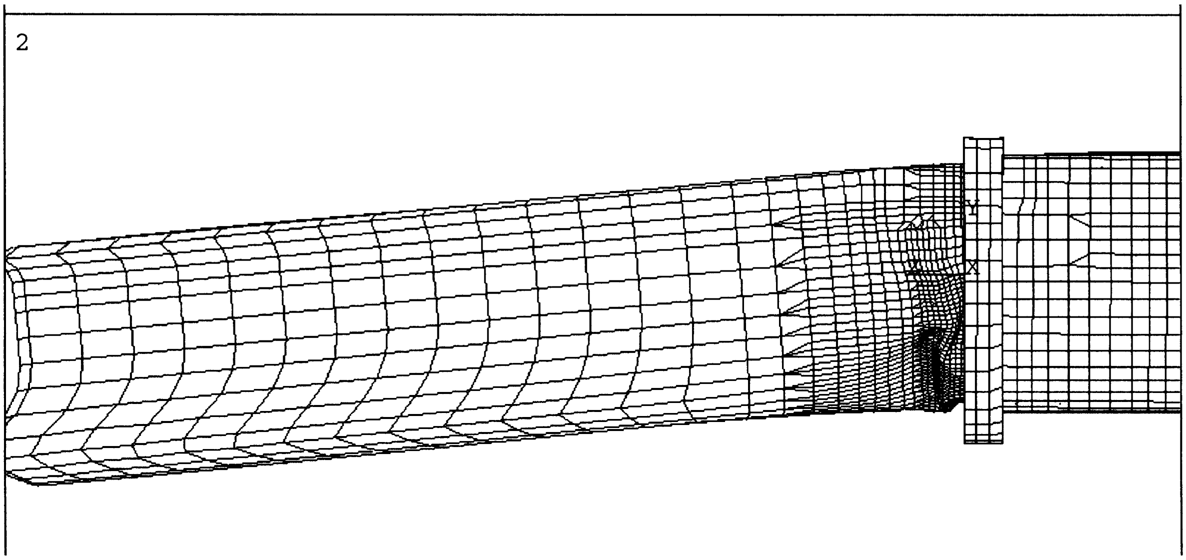


Fig. 12: Buckling state with the original thickness distribution; real structure; transient analysis

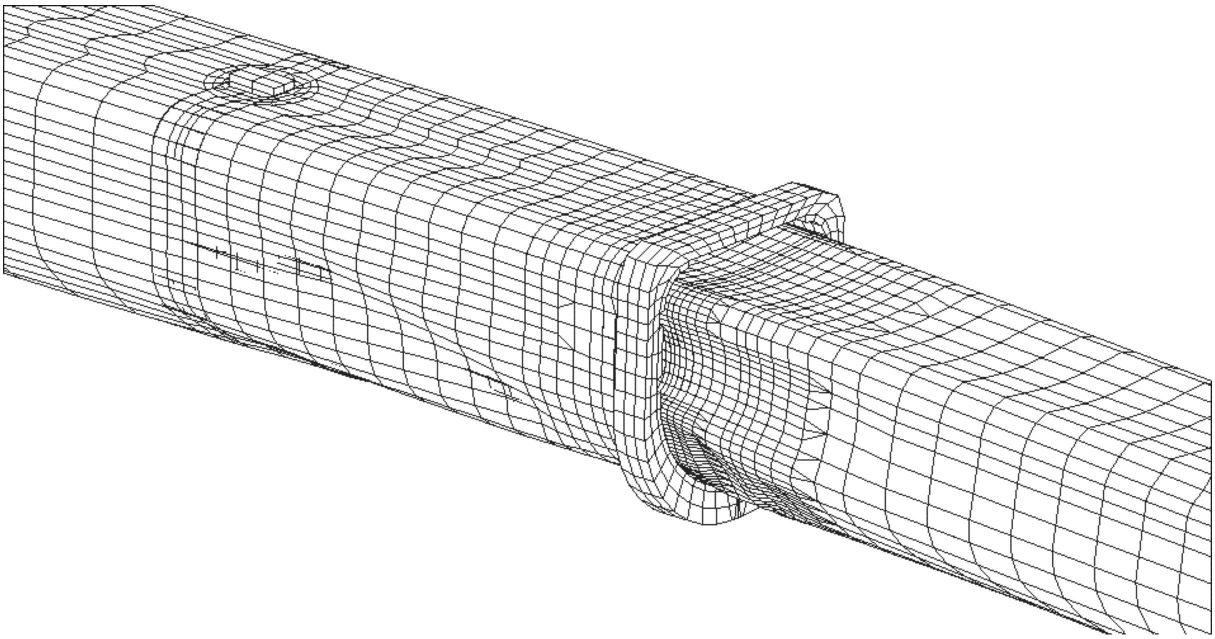


Fig. 13: Buckling state with a slightly modified thickness distribution; real structure; transient analysis

DAMPING

In a quasi-static solution damping should be avoided, as this could lead to overly high estimates for the buckling load. In particular mass proportional damping which decelerates the global motion should not be applied. For monotonous proportional loadings usually no damping is required, maybe in the case of constant velocity some damping for the starting phase may be advantageous.

POST-CRITICAL BEHAVIOR

The post-critical behavior after reaching the limit load is usually highly dynamic (see oscillations in Fig. 8). However, this is closer to reality than any static post-critical equilibrium path because buckling and failure processes usually happen suddenly.

FURTHER EXAMPLE

For details we refer to Schweizerhof/Walz et al. [4] and restrict our considerations to the major effects.

Quasi-static roof crush analysis

A car roof (Fig. 14) is pushed by a more or less rigid contact body with a flat surface with low velocity, i.e. static. It is simulated by LS-DYNA. Since the load is applied by a contact surface with small touching areas at the beginning no initial velocity distribution is required. For reasons of computational costs the simulation speed should be as high as possible. Fig. 16 shows that the solutions for different velocities do not differ until reaching a certain limit. Applying the load faster leads to a qualitative difference in deformation (Fig. 15) due to dynamic wave propagation. Then not only a higher load level is calculated but also higher peaks indicating dynamic effects.

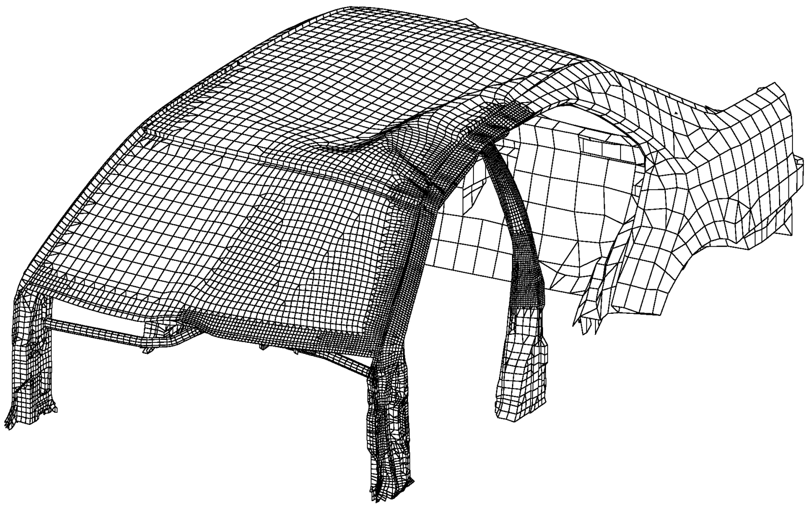


Fig. 14: Quasi static roof crush analysis, loading velocity 2000 mm/s

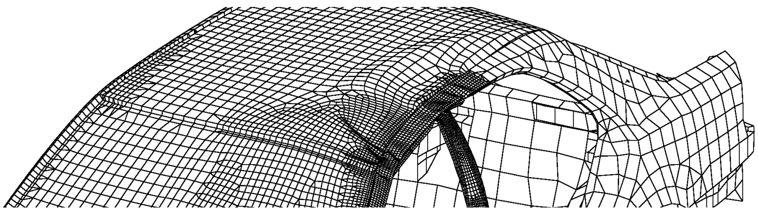


Fig. 15: Quasi static roof crush analysis, loading velocity 10000 mm/s

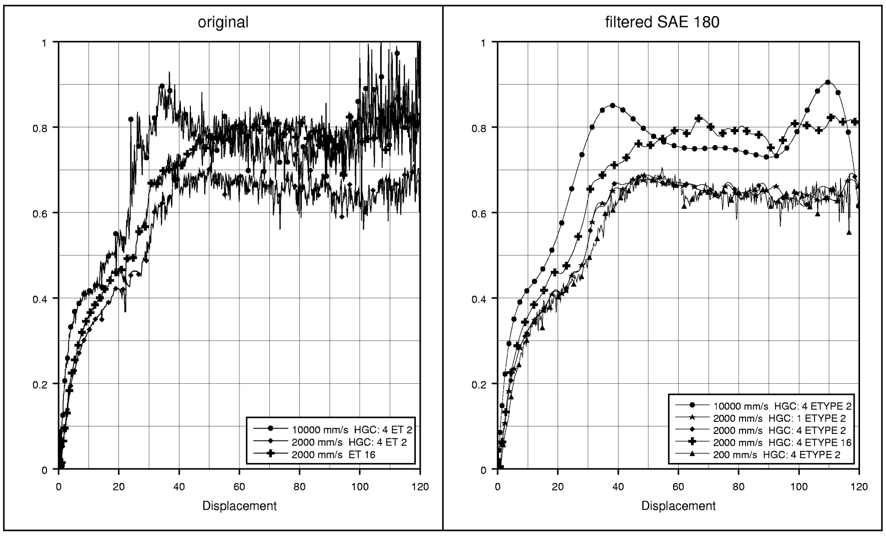


Fig. 16: Contact force (scaled) at loading plate for different velocities

OTHER QUASI-STATIC ANALYSIS

LS-DYNA has been used for quasi-static analyses of different kinds than described here. Of particular interest are processes consisting of a sequence of different loadings resp. motions. Then damping is required to achieve a static solution at the end of one phase before the next load can be applied. These damping periods can increase the computational time significantly. Nevertheless the quasi-static LS-DYNA analysis can be advantageous if highly nonlinear phenomena lead to convergence problems in an implicit analysis which can only be overcome by time-consuming experiments concerning solution control.

In this paper only large deformations and elasticity with some plasticity was considered. The problems of implicit solvers and thus the advantage of LS-DYNA increases significantly if material failure is taken into account because sudden loss of stiffness is crucial for implicit methods but is a „standard“ option for LS-DYNA materials.

CONCLUSIONS

Although standard ANSYS has a lot of advanced nonlinear features, solution methods and convergence tools, a quasi-static LS-DYNA analysis can be an advantageous alternative in the case of systems containing multiple highly nonlinear effects. Limit load analyses are typical examples of this kind but other applications can be solved in this way being at least less work consuming.

ANSYS/LS-DYNA for preprocessing and standard ANSYS for preparing the following transient analyses by static solutions are appropriate tools to gain the maximum advantage of explicit transient analysis. Especially the calculation of initial velocities and initial static displacement distributions can help considerably to reduce computational costs. Eigenvalue buckling analysis is the appropriate tool to determine geometric imperfections. Randomly distributed imperfections should be handled with care.

REFERENCES

[1] Bartel,Th. Stabilitätsuntersuchungen auf der Basis der Finite-Elemente-Methode unter Berücksichtigung geometrischer Imperfektionen und Materialnichtlinearitäten mit dem FE-Programmsystem ANSYS. Diploma Thesis, Fachhochschule Hannover 1998

[2] Hanke,M., Dawani,A. Simulation eines Kielluftschiffes: Berechnung der Hüllenstabilität. *Proceedings of the 17th CAD-FEM Users‘ Meeting,* Sonthofen 1999

[3] Kessler,D., Rust,W., Franz,U. Aspekte von Traglastberechnungen mit (ANSYS-)LS-DYNA. *Proceedings of the 17th CAD-FEM Users‘ Meeting*, Sonthofen 1999

[4] Schweizerhof,K./Walz,M./Rust,W./Franz,U./Kirchner,M. Quasi-static Structural Analysis with LS-DYNA – Merits and Limits, *Proceedings of the Second European LS-DYNA Conference*, Göteborg 1999