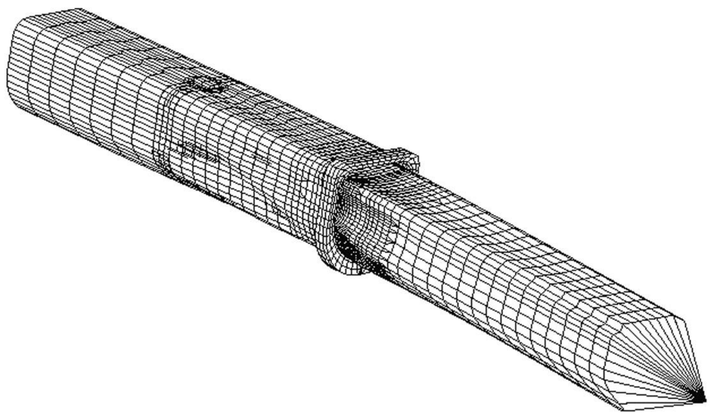
QUASI-STATIC LIMIT LOAD ANALYSIS BY LS-DYNA IN COMBINATION WITH ANSYS

Wilhelm Rust, Ulrich Franz

CAD-FEM GmbH, Schmiedestraße 31, D-31303 Burgdorf



ABSTRACT

It is shown when and how quasi-static limit load analyses can be performed by a tran-sient analysis using LS-DYNA. 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, LS-DYNA, ANSYS/LS-DYNA

ANSYS, LS-DYNA, CAD-FEM

The CAD-FEM GmbH is distributor of both ANSYS as well as LS-DYNA. Therefore, CAD-FEM was interested in an interface between these two programs. For some years ANSYS, Inc., developer of the ANSYS code, is creating the ANSYS/LS-DYNA product offering the classical ANSYS users the benefits of explicit LS-DYNA analyses. Typical applications are drop tests of consumer goods and – as described here – ultimate-load analysis

IMPLICIT STATIC LIMIT-LOAD ANALYSIS

Determining the limit load of a structure primarily is a static problem. It has to take into account at least geometric non-linearities but often non-linear material and contact, too. Since no static equilibrium is possible after the ultimate load some steps are necessary to reach this point in a reliable way and to be sure that it was found.

In conjunction with limit-load analysis often buckling as a bifurcation phenomenon appears. Such a problem can be analyzed by eigenvalue solution, but this requires a system matrix which is not available in an explicit analysis.

If imperfections are superimposed on a perfect 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.

At least in the case of a force-type load path-following methods like arc-length algorithms are necessary to reach the post-critical load-deflection path in an implicit static analysis.

Possible problems in implicit static 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 some difficulties may appear since contact elements change their status (from open to close or vice versa) which is not differen-tiable 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.

PROPERTIES OF LS-DYNA CONCERNING QUASI-STATIC ANALYSIS

Advantages

The main advantage of LS-DYNA for quasi-static problems is that – due to the explicit time integration scheme - no iteration of nonlinear systems and no con-vergence control is required. Therefore, no convergence problem can appear.

A further advantage of transient analysis is that in the vicinity of a critical point the inertia forces stabi-lize the system motion even in the post-critical range where the load which the system can carry de-creases 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 detec-tion of stability points cannot be per-formed.

COMBINING ANSYS AND LS-DYNA

The combination of the general purpose FE-program ANSYS (with implicit solution) and LS-DYNA is the ANSYS/LS-DYNA suite. It at least 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 pre-processor. 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 (including the ANSYS solver) in a sequential fashion. Once a discretiza-tion is modeled for the one type of analysis it is straightforward to switch to the other. A standard appli-cation 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 ANSYS can write the control cards for “Initialization to a prescribed geometry” and create the file (m=...) containing the deformation state from 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

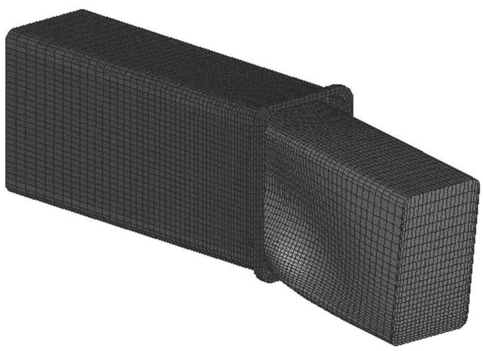


Fig. 1: Simplified model of part of a telescope crane at the onset of buckling

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. 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. 1). This system was first analyzed using ANSYS only [Bartel (1998)].

A look onto the load deflection curve of the industrial system (Fig. 2) 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.

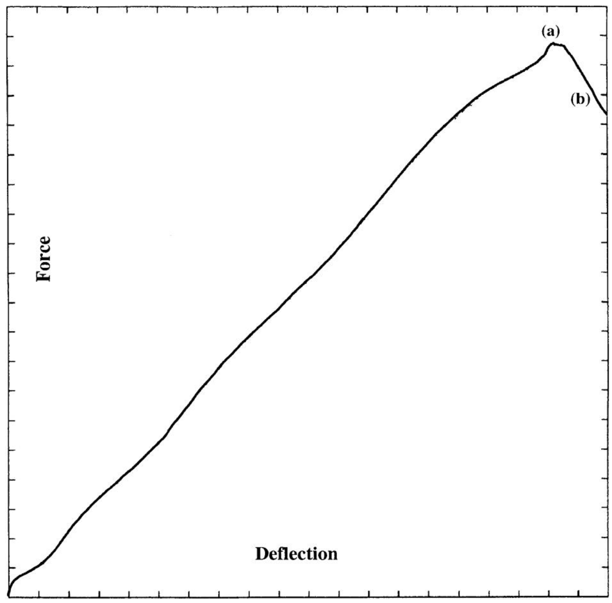


Fig. 2: Load-deflection diagram of the telescope arm of a crane, real system

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 and a subsequent eigenvalue buckling analysis was performed. The modes guided the application of imperfections (see below).

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 prob-lem are considered.

In general a displacement control is preferable, because then the global motion of the structure can be well con-trolled. However, the loading consisted of a force F and a moment M at the tip of the part model. There-fore, the LS-DYNA model is extended by a beam of rigid material (Fig. 7) of the length $2e$, where $e = M/F$, and then the dis-placement of the center of gravity 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 rigid beam as a line-arly increasing velocity (because acceleration boundary conditions are not allowed for rigid bodies, *BOUNDARY_PRESCRIBED_MOTION_RIGID). 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. 3. At a first look such a level of acceleration seems to be far away from being static and absolutely too high. How-ever, it must not be compared with the weight of the system but with the limit load and the equilibrium situation there. Fig. 3 also contains a comparison of the results obtained with different element formula-tions (Type 2, 10 and 16). Since no visible differences can be observed the computationally most efficient element (Belytschko-Tsay) can be chosen.

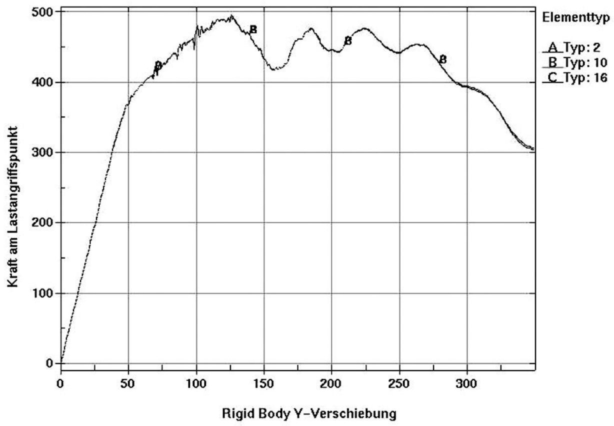


Fig. 3: Load-deflection curve for the telescope crane; model problem; transient analysis

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.

Only if a displacement control is not possible, e.g. in the case of distributed loads, force control is an alterna-tive. Then the acceleration is determined by the amount of DF which is the difference between the static reaction and the applied load. This load increment should be constant. In the case of a linear pre-buckling behavior this can be achieved by a line-arly increasing force. It is advisable to reduce the loading velocity in the vicinity of the limit load according to the system response.

An advantage of constant acceleration loading is that it does not need any static pre-calculation if imperfections are handled otherwise.

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 concern-ing 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 un-der 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 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 tran-sient one. The necessary additional LS-DYNA input (using *INITIAL_VELOCITY_NODE) is written using an ANSYS macro command sequence.

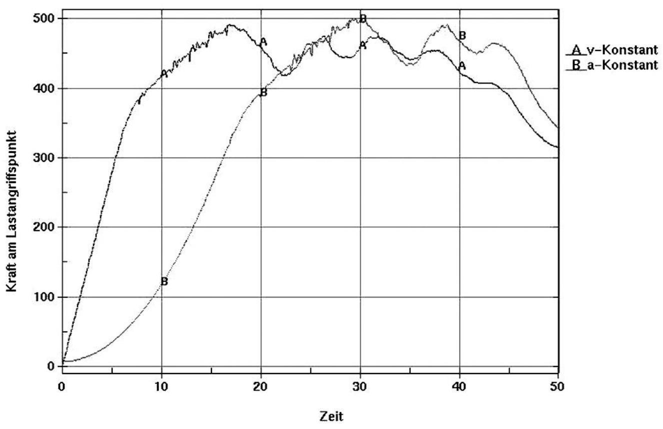


Fig. 4: Load-versus-time curve for A) constant velocity and B) constant acceleration

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 ele-ments 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 condi-tion 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 oscilla-tions.

In the model problem a velocity of 7 m/s of the rigid beam leads to the system response given in Fig. 4. 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“ (*CONTROL_DYNAMIC_RELAXATION, IDRFLG=2), where LS-DYNA expects a file (m=filename in the program call) containing displacements for all nodes. For preparing this file from the results of an ANSYS run (or a previous LS-DYNA run producing results for the ANSYS postprocessor) and acti-vating 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 dis-placement state which is achieved as described above except that the total displacement u is replaced by the displacement increment _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 accel-eration 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 transient analysis executed it must be checked whether inertia and damping forces do not exceed a tolerable level. The comparison of internal and ki-netic 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 transient effects. In the examples shown above the transverse force which can be determined by LS-DYNA for defined cross sections (*DATABASE_CROSS_SECTION, resulting in a SECFORC file) and the fixed end (SPC constraints to get SPCFORCes) must be constant, whereas the moments should vary linearly. In a transient analysis time discrepancies in the response curves for the loaded end (resulting in reaction forces in file BNDOUT) and the fixed end might appear. In Fig. 5 (left) it is shown that the forces excellently match for the considered sections. In comparison, in Fig. 5 (right) it is shown that significant oscillations in the loading phase may occur due to too fast load application. In Fig. 6 it is demonstrated that for the model problem at *v=const.=7 m/s* tip load and reaction at the fixed end are syn-chronous whereas at *v=14 m/s* differences near the first significant nonlinearity become clearly visible.

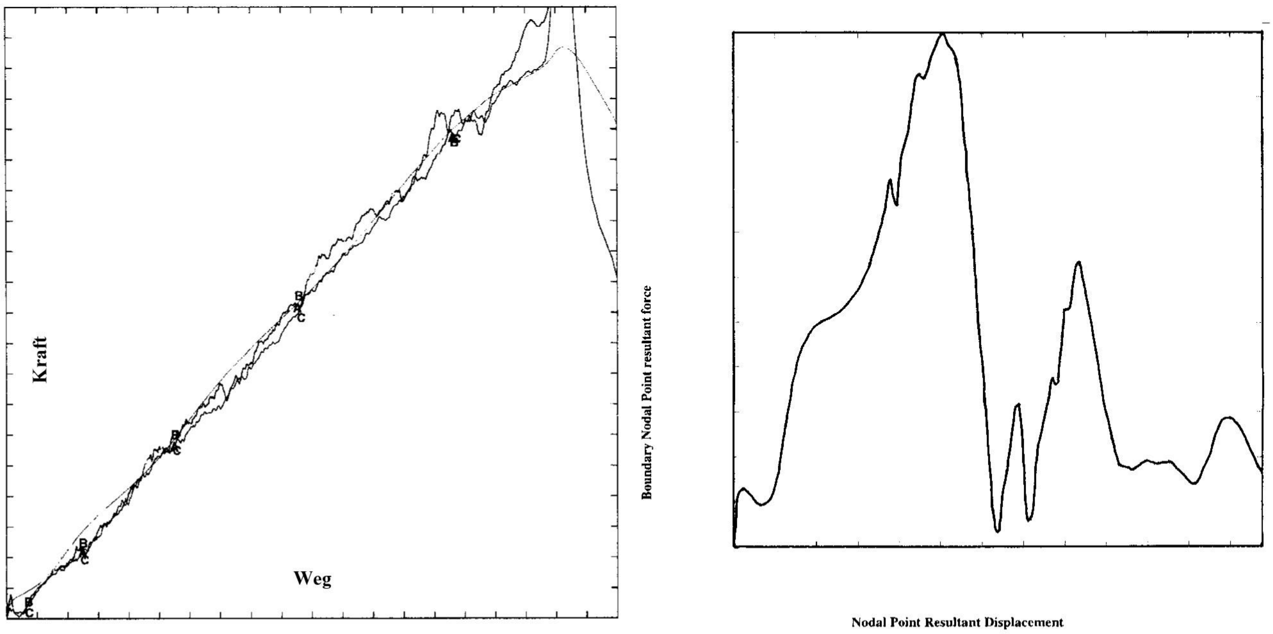


Fig. 5: Telescope arm; real system:
left: Static equilibrium check: force at fixed and free end, cross section force in the middle
right: Oscillations before buckling due to overly high applied acceleration

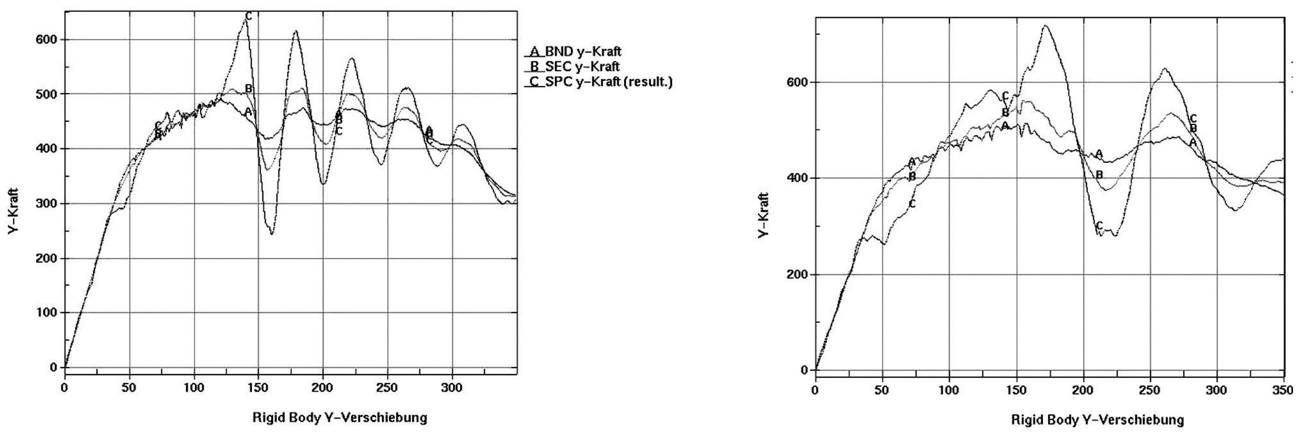


Fig. 6: Force at fixed (B) and free end (A), cross section force in the middle (C)
for *v=const.=7 m/s* (left) and *v=14 m/s* (right)

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 ap-pears but the critical load may be calculated significantly too high and by chance. Furthermore, in reality the limit load and the buckling type often depends on imperfections. If realistic imperfections are known they should be used directly, otherwise a conservative imperfection must be estimated.

Imperfections from eigenvalue buckling

Static limit load investigations, especially of thin-walled or slender structures, are usually started with a linear buckling analysis. The results are buckling modes and load factors. Since one assumption is linear behavior until buckling load factors are only estimates for an upper limit of the ultimate load; and buckling modes show, how the structure will buckle if the system is not significantly deformed beforehand. 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 conser-vative imperfection. That means eigenvalue buckling analysis which needs a system matrix can be essential for a reliable final solution being obtained either explicitly or implicitly.

The buckling mode, usually the first one, can be applied to the ideal model as geometric (stress-free) modifications. In the case of the ANSYS preprocessor this is a simple function.

For more complex pre-buckling behavior others than the first mode can become most important. The modes can remain similar, but the eigenvalues change their order (so-called mode jumping), or the modes can become qualitatively different due to large deformation or contact. In this case it may be necessary to repeat the eigenvalue analysis after applying parts of the load. This also requires a static implicit solution.

The importance of knowing the right imperfection in a transient dynamic analysis is discussed in detail below.

For the considered system an eigenvalue buckling analysis was performed leading to the first buckling mode shown in Fig. 7. The load factors for the different buckling modes were sufficiently separated so that it appeared rea-sonable that only the first mode was of interest. The latter was 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.

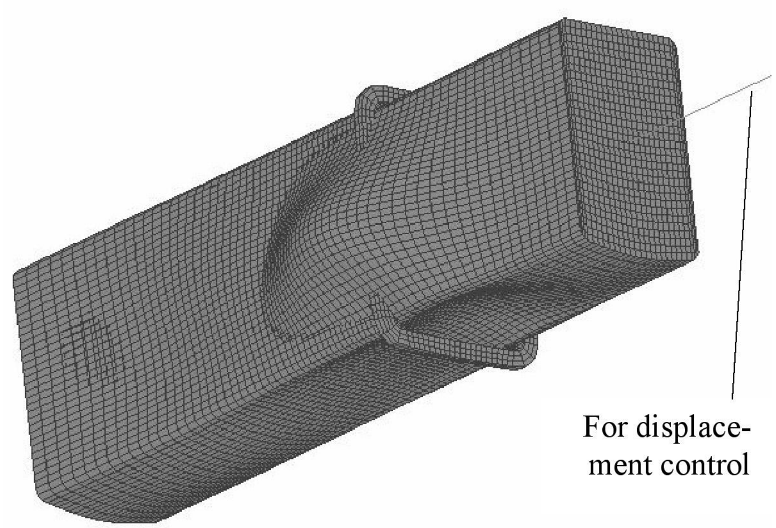


Fig. 7: First buckling mode of telescope arm; simplified model

Arbitrarily distributed imperfection

If no eigenbuckling pre-analysis is desired or it seems too complicated to set up and solve the static problem, other kinds of imperfections are possible. One type of imperfection is generated by the aid of a random distribution (Fig. 8). 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 defor-mation 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. 8) 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.

Although no pre-solving is necessary it should be noted that the preprocessing tool must be able to support such function-controlled modifications to the nodal coordinates including the determination of the shell normals.

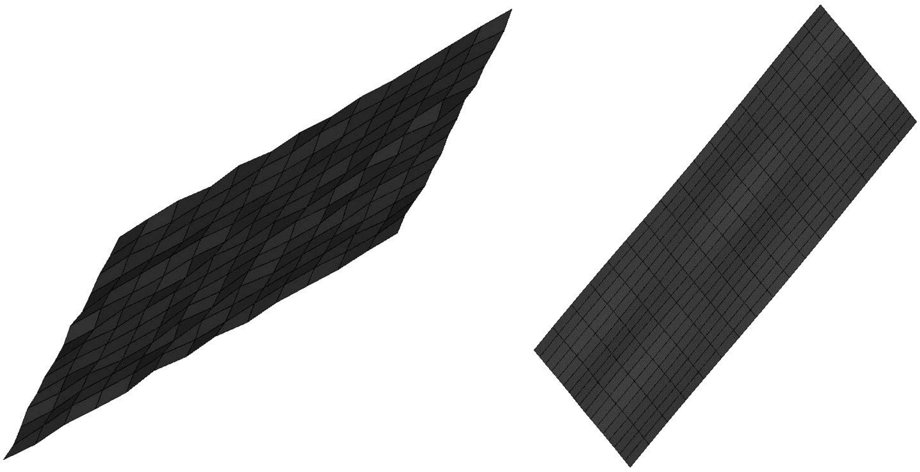


Fig. 8: Random and sinusoidal distribution of imperfections

The results in Fig. 9 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 imperfec-tion.

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 reduc-tion 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.

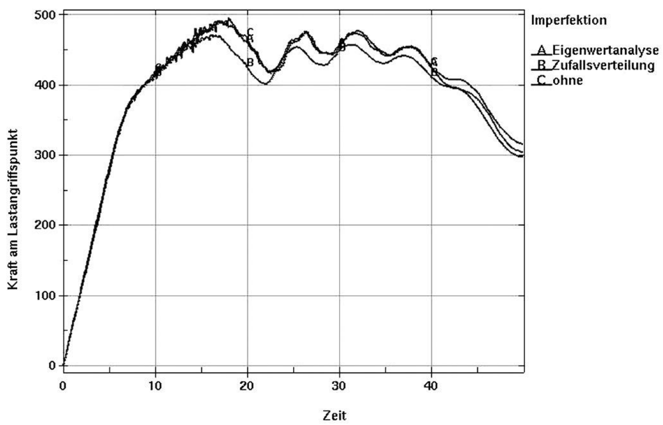


Fig. 9: System response for different types of imperfections:
A) buckling mode, B) random distribution

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. 10 and Fig. 11 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. 10 the buckling mode of Fig. 11 appears, i.e. the „wrong“ imperfection has no fatal influ-ence 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.

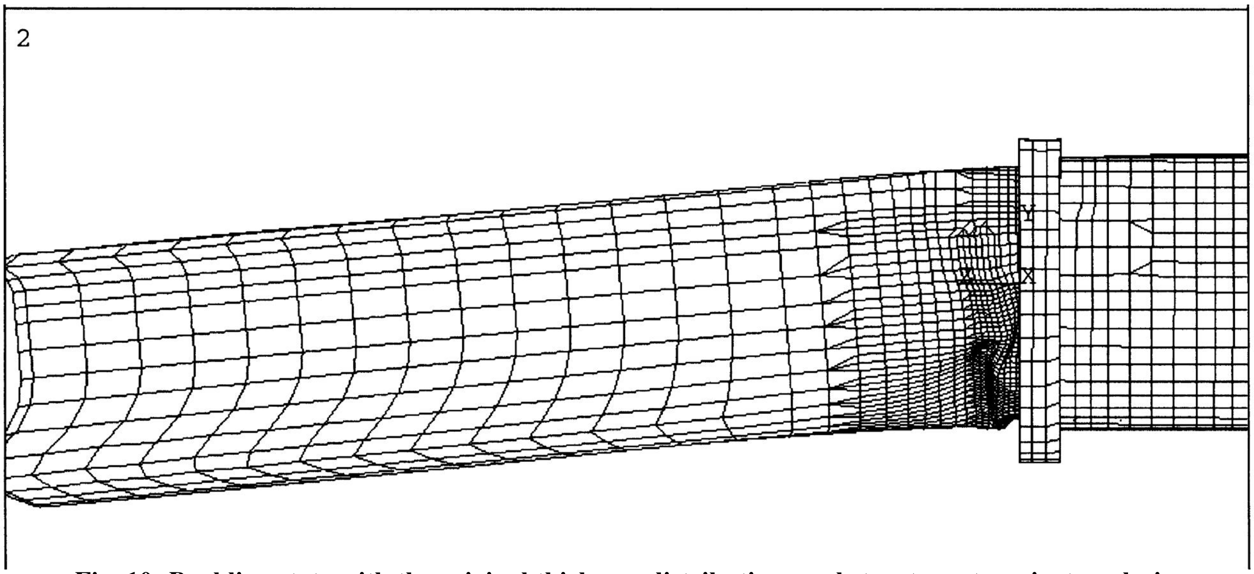


Fig. 10: Buckling state with the original thickness distribution; real structure; transient analysis

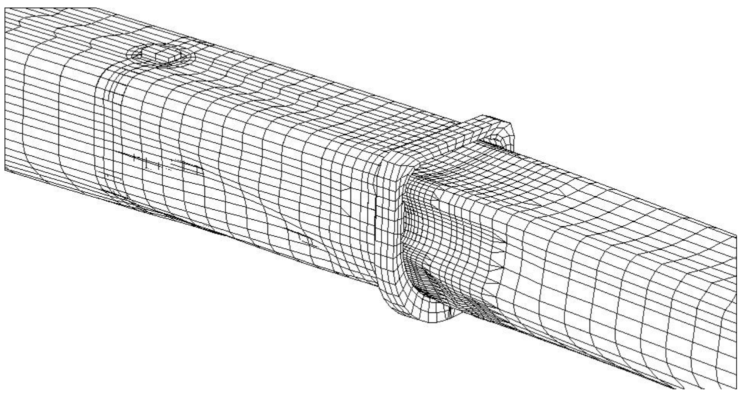


Fig. 11: Buckling state with a slightly modified thickness distribution; real structure; transient analysis

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 con-stant 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. 6). However, this is closer to reality than any static post-critical equilibrium path because buckling and failure processes usually happen suddenly.

FURTHER APPLICATIONS

Limit-load analysis

Further examples of quasi-static ultimate-load analysis with LS-DYNA considered under different points of view are described in [Schweizerhof, Walz et al. (1999)] and [Schweizerhof, Münz et al. (2000)].

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 a 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 were considered. The prob-lems 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“ op-tion for LS-DYNA materials.

REMARKS AND CONCLUSIONS

Although standard ANSYS has a lot of advanced nonlinear features, solution meth-ods 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 consid-erably to reduce computational costs. Eigenvalue buckling analysis is the appropriate tool to determine geometric imperfections. Randomly distributed imperfections should be handled with care.

An implicit solver within LS-DYNA is under development and will be able to partially replace ANSYS pre-calculations. The obvious advantage is the common definition of specific features for implicit and explicit analysis. However, the sporadic user taking LS-DYNA into account for very special problems only will be glad if he can do most of the work in the environment he is used to.

REFERENCES

Bartel,Th. (1998). Stabilitätsuntersuchungen auf der Basis der Finite-Elemente-Methode unter Berücksichtigung geometrischer Imperfektionen und Materialnichtlinearitäten mit dem FE-Pro-grammsystem ANSYS.
Diploma Thesis, Fachhochschule Hannover

Keßler,D./Rust,W./Franz,U. (1999) Aspekte von Traglastberechnungen mit (ANSYS-)LS-DYNA,
Proceedings of the 17th CAD-FEM Users’ Meeting, Sonthofen

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

Schweizerhof,K:/Münz,Th./Graf,O./Walz,M./Tsay,C./Hallquist,J. (2000). On Applications of Adaptive Strategies
for General Shell Structures in Large Deformation Analysis Using LS-DYNA, Presentation
to the International LS-DYNA Conference, Detroit