Experiences with ANSYS in Ultimate-Load Analyses of Aircraft Fuselage Panels

Wilhelm Rust
Fachhochschule Hannover – University of Applied Sciences
Martin Kracht, Josef Overberg
CAD-FEM GmbH, Hannover Office

Abstract

Wider parts of the fuselage of a commercial aircraft are subject to shear and compression loads in such a way that this is relevant for dimensioning. First elastic, later plastic buckling can appear. The ultimate load is reached when a combined buckling of skin and longitudinal reinforcements (stringers) leads to failure of a panel.

This behavior is tested physically, but over years CAD-FEM and Airbus Deutschland worked together to simulate the test and analyze further configurations under aircraft-like conditions with ANSYS.

The paper describes which difficulties had to be overcome to achieve convergence and obtain reliable ultimate loads. The advantages and disadvantages of different load and solution control features such as force or displacement control, arc-length method, different ways of incrementing are lined out as well as different kinds of imperfections. Furthermore, the use of linear buckling and eigenvalue extraction at some stages of non-linear analysis is considered.

It is shown that ANSYS is an appropriate tool for such kind of analysis, but the importance of some missing features becomes evident.

Introduction

Fuselage shells of commercial aircrafts are dimensioned for a number criteria to be fulfilled. One is the load at failure provoked by buckling in combination with material yielding. At Airbus, Hamburg, besides theoretical predictions an advanced test-rig (Figure 1) is used to determine the maximum load capacity, keeping the influence of the boundary at a minimum.

![Figure 1. Fuselage panel and its position in aircraft (left), shear-compression test rig at Airbus, Hamburg (right)](image-url)

Since such tests are expensive and time-consuming a project was started with the goal to reduce the number of physical tests or allow for a more detailed inspection of the parameter space by numerical simulation.
Furthermore, experienced engineers, still responsible for checking and reviewing the results, should be supported in modeling. Therefore, a special GUI supporting parametric model creation was required, too. After a selection process ANSYS was chosen because it is a complete system of integrated preprocessor, solver, postprocessor, because of the APDL capabilities and its TCL/TK integration for the GUI.

**Modeling Aspects**

The shear-compression panels (Figure 2) mainly consist of the skin, longitudinal stiffeners (stringers), circumferential stiffeners (frames) and connecting components (clips). These parts are connected by riveting, welding and bonding.

![Figure 2. Geometry model of the test-rig panel (left), of cutout for aircraft-like boundary conditions (right)](image1)

For weight saving parts of the skin are made thinner (pocketing). For testing single- and double-sided doublers are added to the skin in the boundary regions (Figure 3). In some panels intentionally introduced delaminations require double meshing and further reference-plane offsets.

Materials are aluminium sheet metal, layered metal, fibre-metal laminate (FML) or carbon-fiber reinforced plastics. For the prepreg used in FML an elasto-plastic material law with different curves per direction is...
desired. As a workaround the behavior of the laminate could be approximated by using some non-physical values in Hill’s yield criterion (for details see Ref. 2).

Panels are considered like in the test rig and - smaller cutouts - with so-called aircraft-like boundary conditions (Figure 2).

The user is supported by a special GUI coded in TCL/TK and interacting with classic ANSYS. Some standard GUI features such as for material and section definition are integrated. After interactive definition macros are created from the GUI input. These macros and other parametric ones, e.g. for profiles (stringers, frames) or components, referring to local coordinates, are used beside parameters for the most possible flexibility.

An APDL coding fully automatically creates a geometry model and have it meshed in /PREP7. Furthermore, a LS-DYNA input can automatically be created. APDL-macros had to be added to ANSYS/LS-DYNA features, such as for transferring shell thicknesses and sections, contact definitions etc. For details see below.

Elements used are SHELL181 for all thin-walled parts, rivets by BEAM188. Furthermore, CONTACT170/173 is necessary. Small shell elements represent the welds, serving to connect profiles like in the drawing with the reference plane of the skin and to determine nominal weld stresses. Laminates and doublers make a layered shell section necessary, doublers and pocketing lead to differences between the mid-plane of the section and the reference plane of the shell element (user offset definition).

For more details refer to (1) and (2) or ask the authors, for example about the adventure of defining mesh-independent strain-gauges by means of CBDOF.

**Test-Rig Model and Boundary Conditions**

In the test rig the compression and horizontal shear force is applied by a thick, nearly rigid beam. This is represented by coupling of nodes. The vertical shear force needed to make the shear load moment-free is equally distributed to a number of claws (clamps) by a special mechanism. These claws are considered rigid and have three functions:

1. They transmit the shear force node to the panel via the pilot,
2. they serve as contact surfaces for the skin boundary,
3. they are guided in a likewise rigid edge beam along the panel and thus guide and stabilize the panel edge.

1 and 2 require one rigid target but with different properties (standard and bonded contact, see Figure 4). These properties are set as keyoptions, i.e. different element type definitions are required. Amazingly it is possible to use different type numbers but the same real number to form one rigid target with one pilot node!
Figure 4. Claws in edge-beam for shear loads and boundary stabilization with different contact types

Other ways such as connecting two pilot nodes by a deformable beam caused convergence troubles from the beginning.

Figure 5. Slightly simplified boundary conditions for edge-beam and clamps

For function 3 a rigid-to-rigid contact would be necessary but is not available. Therefore, the skin is extended by shells of large thickness and contact definition to the rigid edge beam (Figure 5).

Aircraft-like Boundary Conditions

Aircraft-like mainly means cyclic boundary conditions. For the load application the nodes at the horizontal edges are coupled to a master, in axial direction (for compression) the nodes of skin and stringers, in circumferential direction (for shear) the nodes of the skin only. Shear forces in axial direction need not to be applied because they are reactions from the pairwise coupling of dofs.

Static Analysis

The main analysis is static with geometric, material and contact non-linearities. Goal is a reliable determination of the ultimate load, the maximum load the panel can carry.
Load Application

Arc-Length Method

Since in the test the ratio of circumferential and axial shear as well as compression force is controlled, nodal forces must be applied to the FE-model.

The most reliable indication for an ultimate load is reaching the postbuckling region. The arc-length method should do, however, what looks like a maximum in the load-deflection curve emerges as a change in the load direction, i.e. unloading, which is not detected by the algorithm, because loading and unloading path differ due to plastic material (Figure 6).

The ANSYS arc-length algorithm must be improved! Different constraints as alternatives to Crisfield’s one with its two solutions should be to the user’s choice. Maybe the incrementation can be improved, too.

A simpler method, a displacement control where forces are applied but their level depends on the displacement of selected nodes could avoid undesired unloading because the direction can be prescribed.

The following steps must be followed:

• define a (force-type) load to be applied as $\lambda f^{ext}$
• prescribe a displacement $u$ for a selected node $i$
• determine the load level $\lambda$ and the displacements $\Delta u$ from $K_i \Delta u = \lambda f^{ext} - f^{int}$:
  - solve $K_i \Delta u^{ext} = f^{ext}$ and $K_i \Delta u^{int} = f^{int}$
  - calculate $\lambda = \frac{u - u_i - \Delta u^{int}}{\Delta u_i^{ext}}$
  - compose the displacement increment $\Delta u = \lambda \Delta u^{ext} + \Delta u^{int}$

This is similar to but simpler than the arc-length method and can be offered in the context of arc-length, too.

![Figure 6. Load-Displacement-Curve in an Arc-Length Method](image)

Force Control

In case of the test-rig conditions force-control remains the only choice. As it is well known this leads to non-convergence at or near the ultimate load. Furthermore, non-convergence can occur earlier, mainly because of local instabilities, sudden changes in the buckling pattern. That’s why non-convergence is no
reliable criterion for the ultimate load. Thus, these two reasons must be distinguished by engineering
ejudgement, e.g. by comparing with the results of different shear-compression combinations. What can be
used as indicator is described in section “Eigenvalue Analysis” and what is of help to overcome
convergence difficulties is described below.

**Displacement Control**

Standard displacement control where prescribed displacements form the only loads cannot replace
distributed forces as in the case of the test rig.

However, the load of the panel under aircraft-like conditions is reduced to two forces at the same (master)
node. Given a certain ratio between them the direction of their resultant is known. A rotated node with a
single prescribed displacement will get a reaction force in the same direction. All nodes being coupled to
the master must be rotated, too. In this special case this conflicts with the above mentioned fact that the
stringer nodes are coupled in the axial direction only. The solution is a constraint equation which prescribes
the projection of the master in axial direction as the axial displacement of the stringer.

![Figure 7. Displacement control for two force components](image)

An alternative to nodal rotation is a constraint equation for the two related dofs of the master node where
the projection of its displacement components onto the desired load direction is specified. The load then
can be defined by the constant of the equation which can be modified by CECMOD during solution.
However, although it always has the character of a load it is not affected by the time-stepping. Should be
improved!

A workaround is prescribing one displacement of an external node and introducing it into the CE.

This displacement control stabilizes the analyses effectively such that the majority of them reached the
post-buckling region, making sure that an ultimate load has been reached, i.e. improving the reliability.
Overcoming Convergence Difficulties

Especially in the case of force-control a number of analyses ended before approaching the maximum load. Sometimes a different load stepping was helpful, but this requires some trials and is not methodic. The use of imperfections for convergence improvement is described below.

Implicit Dynamic Analysis with ANSYS

If density has been defined in pre-processing it is very easy to change from static to dynamic analysis only by switching time-integration on. A static analysis should be run before as long as the convergence is good. Then the initial velocities are reasonable for a quasi-static solution. When approaching an instability point, a local or a global one, inertias stabilize the system. Thus, much higher load levels could be reached (Figure 9). Sometimes no global oscillations could be seen, in other analyses significant oscillations appeared but the maximum load was reasonable. The latter reduces the acceptance of the result. The post-buckling region could not be reached.

A problem arises from the automatic time-stepping. ANSYS estimates frequencies of the system and sets the time step to resolve the related oscillations. However, implicit time integration only needs to account for the interesting frequencies which in this case are low. On the other hand, small time steps must be allowed to achieve convergence. Beta-damping was of help after some trials with the value of $\beta$, but the dynamic analysis still needed much more time-steps in the stable region than the static one.

The user should be enabled to influence time-stepping in the dynamic analysis resp. switch frequency estimation off.
Alpha-damping was not applied yet, but actually some numerical experiments are done scaling density (inertia) down and choosing a high value of $\alpha$. Furthermore, performing a transient analysis without mass (possible in ANSYS!) but with beta-damping. The latter means, however, that the damping matrix is proportional to the stiffness matrix with its tendency to rank deficiency in when approaching the critical point.

A feature a damping-like stabilization of a static analysis is desired.

**Figure 9. displacement-load curve of the static and implicit dynamic run**

**Explicit Dynamic Analysis with ANSYS/LS-DYNA**

Because of explicit time-integration scheme no convergence problem can appear in LS-DYNA: convergence is not obtained. On the other hand very small time steps are necessary to assure numerical stability. Therefore, the simulation time must be very short i.e. high velocities and accelerations appear. However, it is possible to get a quasi-static result in a reasonable computational time. For details see (Ref. 3).

With this method the post-buckling region could be reached and a reasonable critical load was calculated (Figure 10). Resulting forces at different cross-sections (just below load application, at the opposite edge and in the middle) match (Figure 10) showing that a quasi-static solution is obtained. Thus, ANSYS/LS-DYNA was a very useful add-on for this type of problem.
A lot of APDL coding was necessary to get the model run, such as transforming

- rigid targets with pilot node into rigid bodies with extra node
- contact and target elements into slave and master segments
- sections, partially with offsets, into user-defined integration rules
- MISO material into piecewise linear
- loads and related time into loads with load-curves and so on

Unfortunately there is still the inconsistency between ANSYS load curves (by tables) and ANSYS/LS-DYNA load curves (by parameters).

Some of these functionalities should become features of ANSYS/LS-DYNA, but immediately it should be avoided that

- real constants are cleared when issuing ETCHG (in case of shells a shifting by two positions would be the solution)
  Actually they must be saved to an array parameter before the ETCHG and then put back into real constants if the information should not get lost.
- newer ANSYS elements like 188 have no companion element for ETCHG

**Imperfections**

Imperfections initiate branching to a secondary path in a bifurcation problem like buckling. This is especially useful if this path is still stable, i.e. the load can still be increased. Then imperfections are necessary for convergence.

In the case of the test rig enough natural imperfections arising from the shape of the boundary, the load application and some excentricities are present. The panel with aircraft-like boundary conditions is more ideal and imperfections are necessary.
One often recommended way is to take the scaled first mode from a linear buckling analyses as a geometric imperfection which is supported by the UPGEOM command. Why this method fails here is shown below.

**Force-Type Imperfection**

The most primitive way of creating imperfections is that by applying single forces. In the final stage the panels usually show one half-wave between two stringers and several ones between two frames. In order not to overpredict the number of half-waves only two single loads are applied at one end of one skin region in a separate load step. A certain excentricity is chosen to make the loads appropriate to initiate buckling in compression as well as in shear loading.

![Figure 11. Imperfection (radial displacement shown) resulting from two single loads](image)

**Imperfection from Results**

If the ultimate load is not reached in the first attempt, which can be seen in the load-displacement curve when using the displacement control, the forces are removed. The difference between the last converged solution and the second-last one (Figure 12 and Figure 13) obtained from load-case operation shows a quite different deformation than the last solution itself. It shows how the system will deform further. This displacement state is scaled as the geometric imperfection by the UPCOOR command. The analysis is started again. In nearly all cases a limit point was reached at least in this second attempt.

![Figure 12. Solution (radial displacement) of two succeeding substeps at instability](image)

![Figure 13. Difference between two succeeding substeps (left), last solution (right)](image)

**Eigenvalue Analysis**

Ideal buckling is a bifurcation problem. That’s why in general the solution of an eigenvalue problem can produce knowledge when and how buckling will appear. Imperfections might change the problem to a general non-linear or a snap-through problem. In the case of the panels the load can be increased after the
first buckling. The buckling pattern may change at higher load levels until a combination of buckling and material yielding determines the critical load.

**Linear Buckling Analysis**

If the pre-buckling behavior is linear a linear buckling analysis yields the ideal critical load and the related buckling pattern. If - like in this case - the assumption of linearity does not hold it is recommended to use the mode to determine a geometric imperfection. In the typical example given below, however, the final buckling is similar to the seventh or eighth mode (Figure 14). How to know this beforehand? Thus, in the most cases linear buckling analysis is of no help.

![Figure 14. First, 7th and 8th linear buckling mode](image)

**Eigenvalue Analysis and Contact**

Some of the models contain contact elements. The documentation say that the status of the elements from the preceding static analysis is frozen. However, often messages of “abrupt status change” appear during preparation of eigenvalue extraction. It is not known whether this has influence on the results but seems to be a bug!

**Pre-stressed Modal Analysis**

Since eigenvalue buckling after a non-linear static analysis (sense and necessity are described below) is not working in ANSYS a pre-stressed modal analysis can be used as a workaround after the following consideration:

A natural frequency depends on mass and stiffness. The stiffness changes by the non-linear effects. At the critical point the stiffness against the buckling pattern tends to zero resulting in a zero eigenvalue of the tangential stiffness matrix. No stiffness means that the frequency tends to zero. Thus, a zero natural frequency is an indicator for an instability point. The oscillation mode and buckling mode become identical. This does not hold for the other modes.

The eigenvalue problem reads:

\[
\begin{pmatrix}
K_T + \omega^2 M
\end{pmatrix} \mathbf{p} = 0
\]

where \( \mathbf{M} \) denotes the mass matrix and \( K_T \) the tangential stiffness matrix which already accounts for stress stiffening effects or - here - the opposite, stress softening.
One problem is that \( \omega^2 < 0 \) gives no meaningful frequency and is suppressed by the program (Figure 15 and Figure 16) although calculated in the Subspace method and leading to a negative shift in Block Lanczos method. In conjunction with buckling, however, the suppressed mode is the most meaningful one, telling the user how the system wants to buckle. Furthermore, a negative eigenvalue indicates that the non-linear static analysis has missed a bifurcation point and remains on an unstable path. The example of Figure 17 show that such an indicator is necessary to get reliable ultimate loads. Without it the danger is that the determined critical load is not the lowest one, i.e. the analysis is not reliable.

Figure 15. radial displacement (left), first mode (middle), second mode (right) at a certain load level

Figure 16. radial displacement (left), \textit{first(!)} mode (right) at a slightly higher load level

Figure 17. Three force-displacement paths for the same system depending on imperfections
Eigenvalue Buckling after a Non-Linear Static Analysis

After a non-linear analysis the critical load level cannot be determined directly. However, as stated above, the tangential stiffness matrix at the critical point has a zero stiffness against a buckling mode resulting in a zero eigenvalue in the following problem:

\[(K_f + \lambda I)\phi = 0\]

where \(I\) is the unit matrix. Mathematically this is nothing else but a simplified form of the modal problem from above. However, it is not supported by ANSYS although it has been in the past at least for selected elements.

This feature should be implemented immediately. The effort for coding is minimal.

With this feature it is possible

- to detect critical points and unstable paths,
- to determine appropriate geometric imperfections helping to find the critical path and to overcome convergence difficulties.

What would be a major improvement is the following:

- before starting a (maybe selected) substep determine the first buckling mode
- scale it to a user-defined maximum displacement and add it to the starting vector of the Newton-Raphson iteration (usually the last converged solution) \(u_i^0 = u_i^\infty + c\phi\)

Far away from the critical point this will not influence the iteration much and the solution not at all, but close to a bifurcation point this will help to branch to the critical path with the advantages listed above.

The accuracy of the mode need not to be high, the mode extraction scheme can be a simple one such as inverse (von-Mises) iteration. This can use the same triangularized system matrix but with different right hand side.

All this is known for long (cf. Ref. 4), one of the author has coded and tested it in an academic program.

**Necessary Improvements**

1. Adjust behavior of contact elements for eigenvalue analysis
2. Allow mode extraction for negative eigenvalues
3. Allow buckling mode extraction after a non-linear analysis
4. Implement buckling mode extraction parallel to a non-linear analysis
5. Implement perturbation of initial displacement vector by a buckling mode

**Contact**

If contact is present the analyses need much more solution time because of worse convergence. An option to use a differentiable force-penetration relation (Ref. 5) would be of help for Newton’s method. This will hold for the most problem types but in conjunction with physical instabilities the properties of the tangential stiffness matrix become more important.

Because of the cylindrical surface of the skin, element edges not following the main directions show a slight warping. This and different mesh sizes causes some trouble in contact in conjunction with shell thickness effects included. Furthermore, Keyoption 11 does not work correctly for sections with offset. Therefore, the ICONT logic is used.
An optional $C^1$-representation of the contact surface could be of help (Ref. 5).

The elements of the stringer foot and those of the skin show a certain offset causing a moment when shear is present. In case of welding and riveting structural elements account for that fact. In case of gluing the contact elements offer the MPC contact (must be shell-to-solid, although two shells are concerned). However, this cannot be used here because eigenvalue buckling analyses are needed.

Should be improved, additionally under the aspect that friction causes moments, too, and no MPC is available then!

Actually a tangential stress is integrated over the contact element. A tangential stress times an excentricity is a distributed moment and can be integrated by the same scheme.

As a workaround thin SOLID185 resp. SOLSH190 were placed between the shells, connected directly to one side and by bonded contact to the other one. An orthotropic material law makes these additional elements weak in bending and longitudinal loading but strong in shear and out-of-plane loading.

**Comparison with Test Results**

After overcoming the above mentioned difficulties ANSYS delivers simulation results matching those from the test well to excellently. The ANSYS contour colors of Figure 18 are adjusted to those of the test and it can be seen that size, depth and orientation of the buckles, here from a combined shear and compression load, agree very well. The same holds for one of the curve of a relative displacement versus load (Figure 19). The other differs by a scale factor, however, this result is sensitive: already on the neighbor nodes it differs significantly.

Smaller panels with aircraft-like boundary conditions are good approximations of the full test rig.

![Figure 18. Buckling pattern from measurement and simulation](image)
Conclusion

It has been shown that physical ultimate-load tests of aircraft fuselage panels can be simulated by ANSYS with good accuracy. Furthermore, the numerical analysis allowed for a closer insight into what happens in the test rig, e.g. better knowledge of force fluxes.

Physical tests are still needed for validation but in a reduced number.

Smaller panels with aircraft-like boundary conditions allow for a much faster solution and thus for a lot more parameter variations than in full-scale simulation and physical testing. Amongst others this helped to detect effects which are not considered in analytical method which on the other hand were shown to be conservative.

The complex parametric modeling could be coded in APDL and defined in a user-created GUI.

In spite of the success it becomes evident that some features in ANSYS are missing or need improvements.

References


3) Rust, W./ Schweizerhof, K.: Finite element limit load analysis of thin-walled structures by ANSYS (implicit), LS-DYNA (explicit) and in combination, Thin-Walled Structures 41 (2003) 227-244
