

VIRTUAL TESTING OF AIRCRAFT FUSELAGE STIFFENED PANELS

Peter Linde*, Jürgen Pleitner*, Wilhelm Rust**

*Airbus, Hamburg, Germany, **CAD-FEM GmbH, Burgdorf & University of Applied Sciences, Hannover, Germany

Keywords: *stiffened shell structure, FE limit load analysis, quasi-static dynamic solution, virtual testing*

Abstract

The behaviour until failure of stiffened fuselage panels is crucial in the design procedure. Therefore it is given major attention currently including numerous experimental tests under different load combinations as well as elaborate theoretical computational procedures, all costly and time consuming.

The virtual testing described here is based on a development project aiming at reducing the amount of experimental tests, and narrowing the numerical predictions to experimental results. Parametric modelling of stiffened aircraft fuselage panels, including skin, stringer, frames, clips and connections is described.

The numerical model is based on the finite element method. The development is based on the FE program ANSYS[®] and is using the shell elements of this program. Introductions are given to the element formulations used, to the material models including failure criteria, and to modelling of the boundary conditions of the test rig panels. Options for modelling splices in fibre metal laminates are shown

Different solution procedures included in the development are discussed. Results are presented for welded panels and for fibre metal laminate panels. Comparison with experimental data is made. Conclusions and recommendations for future research are provided.

1 Introduction

1.1 General

In aircraft design theoretical and experimental investigations are important. Fuselage shells are investigated with respect to the static behaviour until failure through the buckling and post-buckling regime. Theoretical predictions are validated by experimental tests. These tests are both very time consuming and expensive.

Therefore a reduction of the number of these tests is desirable. In order to reduce, not eliminate, these tests a virtual test rig has been developed. By this several of the tests can be replaced. The tests are not to be omitted fully and should remain for certain critical load combinations

The virtual test rig described here is capable of analyzing typical fuselage shells, i.e. panels consisting of an outer skin, stiffened longitudinally by stringers, and circumferentially by frames, see figure 1. The virtual test rig is based on the finite element method and considers both material and geometric non linearity [1]. The virtual test rig can simulate either test panels such as those tested in a test rig seen in figure 2, including the specific boundary conditions if this rig, or panels in aircraft conditions.

1.2 Scope

The paper is organized according to the following. Upon the introduction a description of the model is given in section 2. There the major development is described, which is based

on ANSYS [2], including element and material model descriptions, as well as an addition based on LS-Dyna. Section three treats implicit static and dynamic solutions as well as explicit dynamic solutions. Section four presents numerical results with comparison to tests. The fifth and final section presents a summary, conclusions and recommendations.

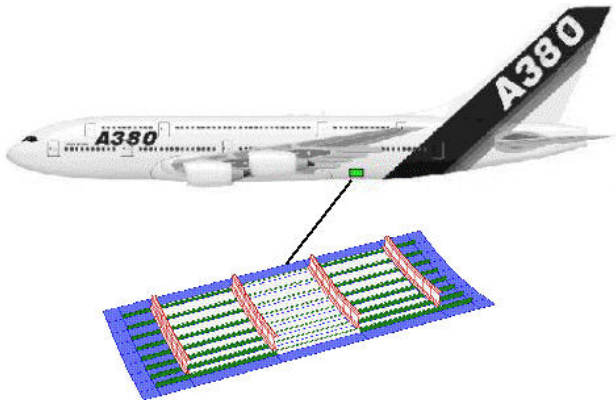


Fig. 1. Stiffened Panel of Aircraft Fuselage

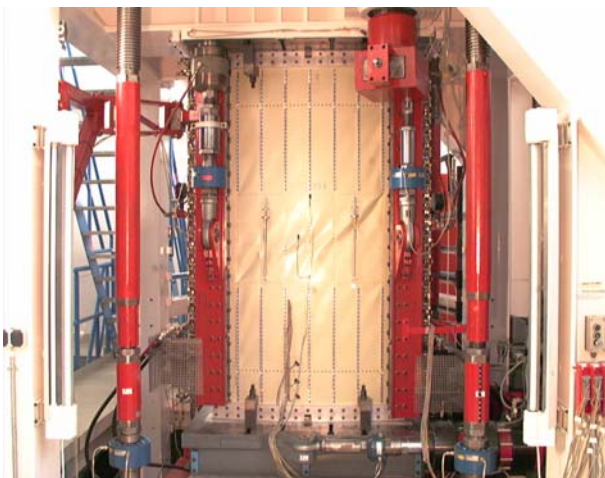


Fig. 2. Test Rig for Experimental Tests of Stiffened Panels

2 Modelling of the Panel

2.1 Geometry Modelling

Before creating a finite-element mesh the geometry of the whole panel is modelled inside the pre-processor of the general purpose FE-tool ANSYS. The modelling is fully parametric and

driven by a graphical user interface designed for this special purpose.

This model is mainly used to create a FE mesh for the implicit ANSYS solver. However, in case of convergence problems an input file for the explicit dynamic FE solver LS-Dyna can also be generated.

2.1 FE-Model for ANSYS

ANSYS is mainly used to perform a static analysis with all kinds of non-linearities. With only very few exceptions the FE model consist of the 4-node Reissner-Mindlin shell element SHELL181. The bending part is formulated according to Bathe/Dvořkin (assumed strain), the membrane part offers the choice between reduced integration (at least used for the skin) and full integration with incompatible modes (mainly used for open shaped profiles like the stringers).

This element allows for a free location of the reference plane (not only mid-plane). This feature is essential because the outer skin is the reference in the real airplane but the thickness can vary for several reasons such as doubling (stiffening of the boundary zones in the test rig) or pocketing (thinner regions between frames and stringers), see figure 3.

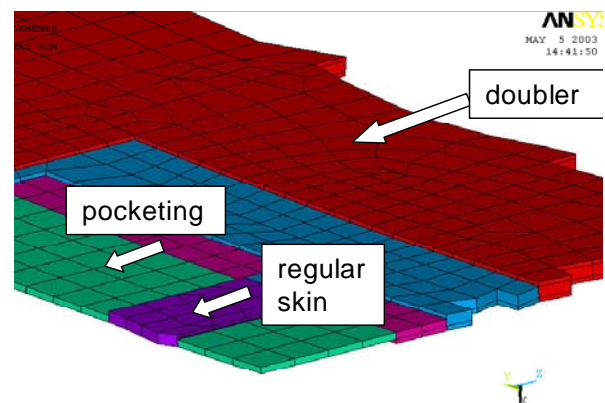


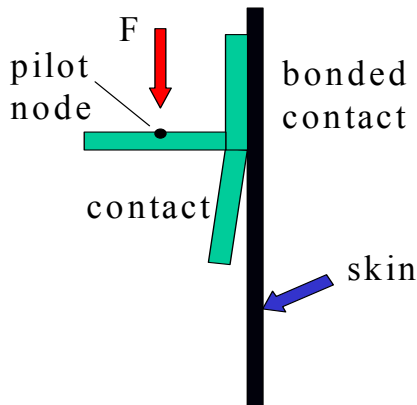
Fig. 3. Varying Thicknesses

Furthermore, the element can account for layered cross-sections. This is necessary in cases where the material is GLARE[®], a fibre metal laminate. The shear correction factors are calculated internally. Each layer can have its own material law.

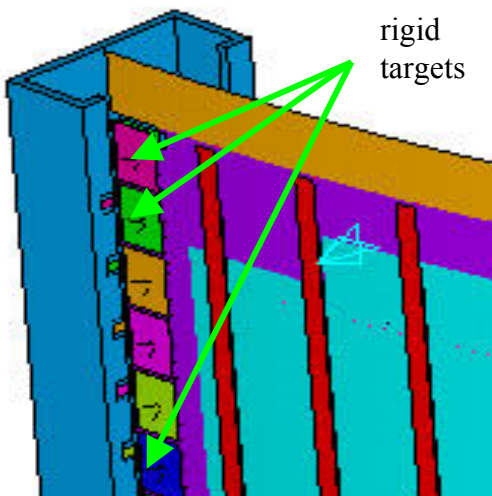
Main connection techniques are welding (modelled via common nodes, but with small elements to get an idea of weld stresses) and riveting (rivets being modelled by Timoshenko beam elements).

Contact is checked wherever necessary. The edge beams are considered rigid, they form rigid targets for contact with shells representing parts of the clamps. The motion of such rigid bodies is constrained at so-called pilot nodes.

Rigid targets are also used for the other parts of the clamps. They are fixed to the edge of the panel by bonded contact, using normal contact for the support of the skin in the adjacent regions, and the pilot nodes are used for the application of shear loads (figure 4a and 4b).



(a). Real Cross Section: Clamps and Loads



(b). Rigid Target Elements (clamps) in Model

Fig. 4. Load Introduction to the Panel by Clamps

2.2 FE-Model for LS-Dyna

ANSYS offers a preprocessing tool for LS-Dyna[®]. However, some features are not supported or different concepts are followed. In these cases some programming in the ANSYS Parametric Design Language (APDL) was performed to complete the transfer of the model. Particular differences are:

- Layered cross sections and reference-plane offsets are modelled by user defined integration rules: the location and the weighting factors of integration points describe the lay-up; the offset from the mid-plane requires a layer of 'nothing', i.e. the sum of the weighting factors is not 1.
- Rigid targets are transferred to rigid bodies, then contact can be defined. The pilot nodes can become a user-defined center of gravity (for constraints) or an extra node placed anywhere (for force application)
- Contact definition is different: In LS-Dyna the standard is automatic overall contact or part related input, but segment based definition like in ANSYS is also possible.

The shell elements used are 4-noded, usually with reduced integration for solution time reason. As in the ANSYS model full integration is chosen for open-shaped profiles.

2.3 Material Modelling

As long as the panel and its components consists of single-layered aluminium sheets, isotropic hardening plasticity was assumed with a piecewise linear stress-strain curve.

The fibre metal laminate consists of alternating layers of aluminium and prepreg. For the prepreg only in-situ properties are of importance, i.e. derived from measurements of a complete lay-up. Nevertheless, the prepreg is extremely anisotropic, especially if the 45°-degree direction is compared to 0° and 90°. It could be shown by experiments that the properties for different number of layers can be

determined from the volume fraction of each material type.

Two levels of detailing have been considered:

- a smeared model considering the material as homogenous over the cross section;
- a layered model.

In the two cases the anisotropy is modelled by using Hill's equivalent stress

$$\sigma_{eqv}^{Hill} = \sigma_{eqv}^{norm} + \sigma_{eqv}^{shear} \quad (1)$$

$$\sigma_{eqv}^{norm} = \boldsymbol{\sigma}^T \begin{bmatrix} \frac{1}{C_1^2} & M_{12} & M_{13} \\ M_{12} & \frac{1}{C_2^2} & M_{23} \\ M_{13} & M_{23} & \frac{1}{C_3^2} \end{bmatrix} \boldsymbol{\sigma} \quad (2)$$

where C_i is a scale factor for the reference stress-strain curve to determine the curve for the specific direction:

$$C_i = \frac{\sigma_{yi}}{\sigma_{y0}} \quad (3)$$

$$\sigma_{eqv}^{shear} = \boldsymbol{\tau}^T \begin{bmatrix} \frac{3}{C_{12}^2} & 0 & 0 \\ 0 & \frac{3}{C_{23}^2} & 0 \\ 0 & 0 & \frac{3}{C_{13}^2} \end{bmatrix} \boldsymbol{\tau} \quad (4)$$

where

$$C_{ij} = \frac{\sqrt{3}\tau_{yi}}{\sigma_{y0}} \quad (5)$$

C_{ij} again is a scale factor for different directions but accounts for the different influence of shear and normal stress on yielding. Thus, the scale factor for normal stress in the 45°-direction can directly be used instead of shear.

Hill's criterion assumes that the stress-strain curves are scalable which is not exactly the case. Therefore, the reference curve has

been chosen after the main loading direction, either shear or axial compression.

In a layered model of the laminate it is possible to get good agreement with measurement in both the axial and the shear direction (see figure 5 and figure 6).

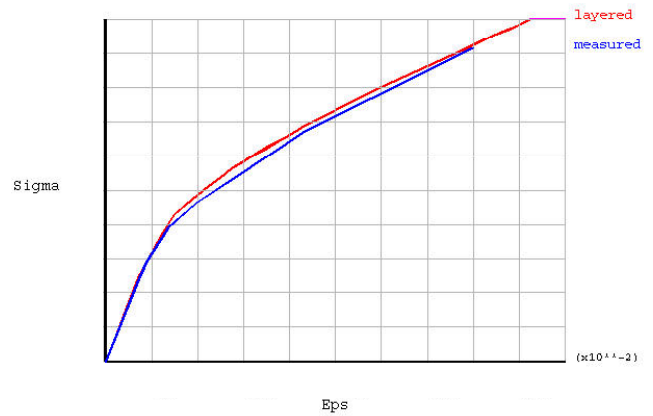


Fig. 5. Stress-Strain Curve for Fibre Metal Laminate, Measured and Calculated in the FE-Model, Axial and Circumferential Direction

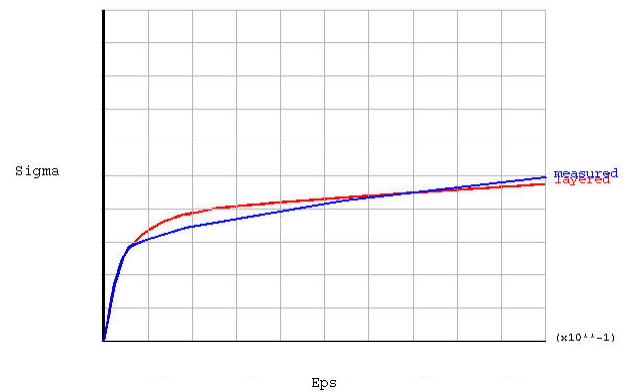


Fig. 6. Stress-Strain Curve for Fibre Metal Laminate, measured and calculated in the FE-model, 45° direction

2.4 Splices

In conjunction with Glare splices are used to join different sections of the fuselage. Within spliced regions two metal layers are glued together whereas one prepreg layer steps to the next aluminium sheet (see figure 7 lower part) and so forth.

Since the skin must be modelled with shells for time and accuracy reasons a layered

shell model is used where the diagonal prepreg is replaced by a horizontal one at an average position (see figure 7 upper part).

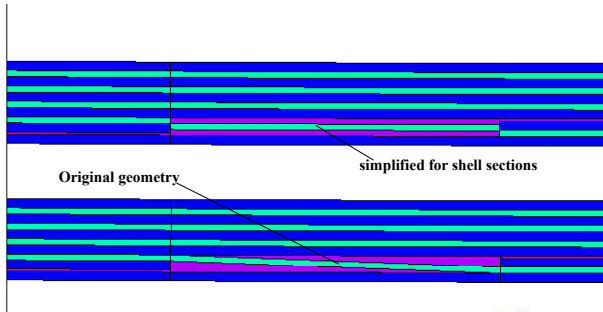


Fig. 7. FE-Shell and Volume Model of a Splice: Metal (dark blue), Prepreg (green), Glue (magenta)

2.5 Boundary Conditions

For simulation of tests carried out in a physical test rig, it is proved to be of particular importance to realistically model the boundary conditions. Already small deviations in this modelling resulted in large deviations in the results. The test shell is fixed to the test rig firmly along the upper edge, as well as to a moving table at the lower edge. Along the sides, shear forces may be introduced into the shell by means of steel clamps, mounted on the sides of the shell's edge, supplied with doubler plates, see detail in figure 8.

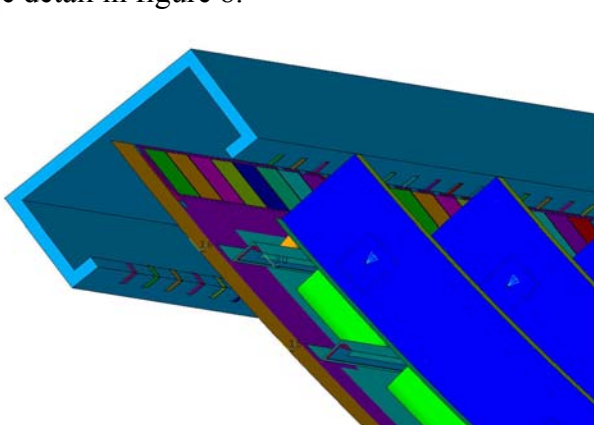


Fig. 8. Detail of Boundary Conditions in Numerical Model, Reproducing a Test Rig

Goals of this concept are moment free shear loads, reducing boundary effects and avoiding to influence compressive behaviour by

shear load attachments. Same shear loads on all application points are technically provided.

The clamps are fixed to a certain skin area (modelled by a rigid target of size and shape of the clamp surfaces with bonded contact), support the edge of the panel in an adjacent region (modelled by standard contact) and get forces at a defined point (modelled as pilot node).

For computational time reasons a slightly simplified model is also used as shown in figure 9 and 10.

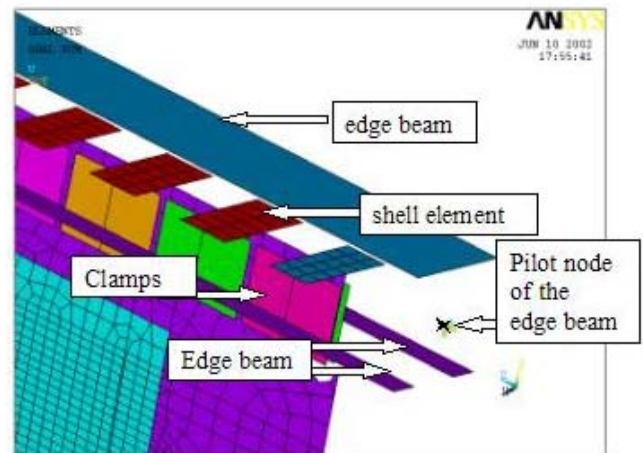


Fig. 9. Slightly Simplified Model of the Boundary Conditions

Since contact between two rigid bodies is not supported the shells perpendicular to the skin are made nearly rigid by a large thickness. This model allows for the same motions of the skin edge as the more detailed one and is proved to be sufficient for good results.

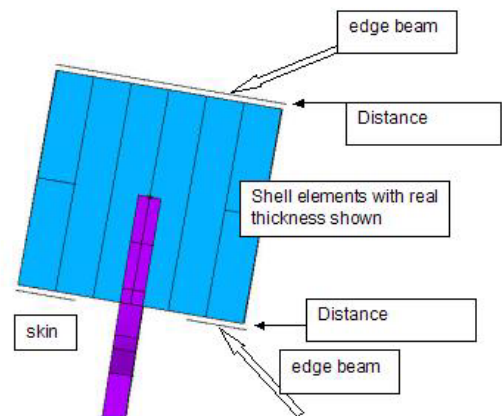


Fig. 10. Detail of Simplified Boundary Conditions

For aircraft-like conditions, the behaviour of the neighbouring panels is to be estimated and here symmetric, anti-symmetric or cyclic conditions may be used.

3 Computation

3.1 General

Non-linear material and geometric behaviour is considered at all times during the computation. Arbitrary shear-compression ratios can be chosen. Both static and quasi static dynamic solutions are feasible, depending on numerical demands of the computed example. In the latter case the choice between implicit dynamic with ANSYS and explicit with LS-Dyna is offered.

3.2 Static Solution

As default the implicit solver of ANSYS is used for a static analysis. It uses a static load step with the default incremental-iterative solution. Full Newton-Raphson iteration is used. Residual tolerances are kept at the default settings of ANSYS.

The load is applied force-controlled. Displacement control is not possible because the ratio of shear and compression forces is prescribed according to the physical testing.

In most cases the failure load is approached to a sufficient level with these settings. Sometimes, however, non-convergence appears before signs of global failure are to be seen. Main reasons are sudden changes in the buckling patterns which lead to local instabilities.

Tests with arc-length method were not satisfying because of the following reason: ANSYS uses Crisfield's method which always determines two solutions. Because of the plastic material behavior loading and unloading paths are not the same. Thus the algorithm did not detect that a decrease in applied load was a simple change in load direction rather than post-failure (figure 11).

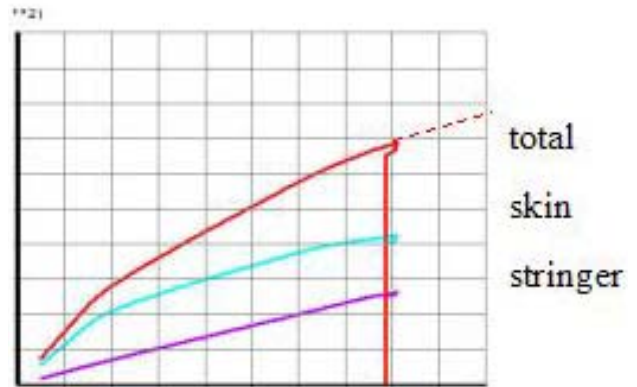


Fig. 11. Load-Displacement-Curve in an Arc-Length Method

3.3 Implicit Dynamic Solution

If there are doubts that the load-level at non-convergence is the failure load, the analysis is restarted at a certainly lower level and continued by a transient dynamic solution using the implicit Newmark scheme, see figure 12. Care must be taken that inertia effects remain negligible. Some beta-damping is used to increase the time-step size in the auto-time-stepping procedure. Small steps must be allowed close to the limit point. In a number of cases this method helped to increase the point of non-convergence until results indicate global failure.

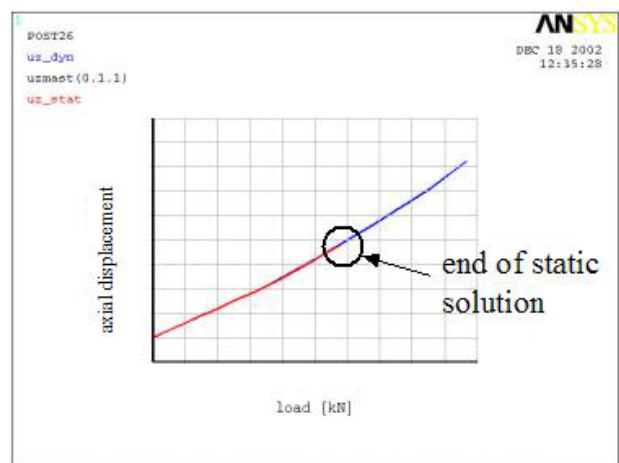


Fig. 12. Continuation of Non-Convergent Static Solution by Transient Dynamics

3.3 Explicit Dynamic Solution

For implicit solutions encountering divergence prior to physical test failure because of numerical convergence problems, a fast transfer to the LS-Dyna software is possible in the virtual test rig. Because of the explicit time integration similar to the central difference scheme no convergence check is necessary, i.e. no convergence problem can appear.

Another purpose could be to analyse the post-buckling behaviour of the panel (figure 13). Firstly this definitely shows that the limit-point is exceeded, secondly one can estimate how sensitively the system will react on imperfections. The latter should influence safety consideration.

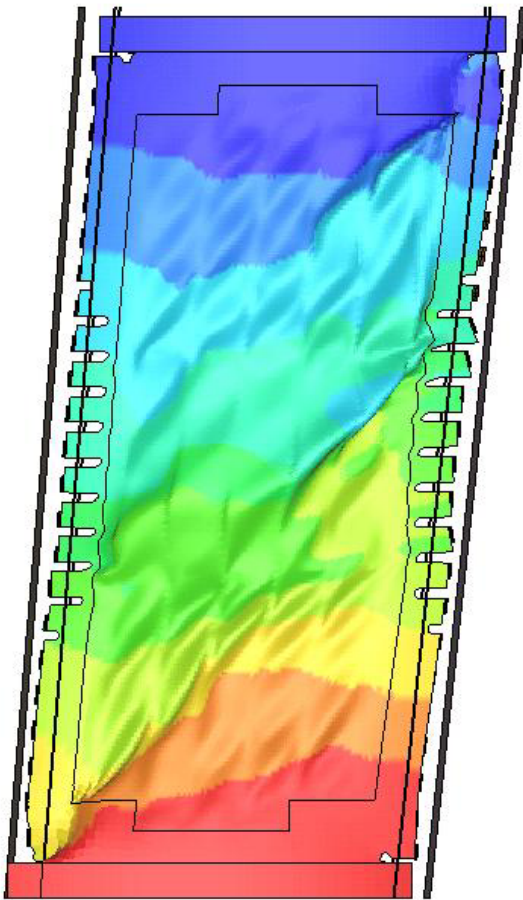


Fig. 13. Post-Buckling Pattern Obtained by LS-Dyna

The computation of the ultimate load is a quasi-static problem, whereas LS-Dyna is a

transient dynamic solver. That is why it needs to be carefully proved in which time the load is introduced. Experiences with this method are good, the static equilibrium (same force at the same time at different sections through the panel) is fulfilled until reaching the limit load (see figure 14). Further experiences and hints are given in [3].

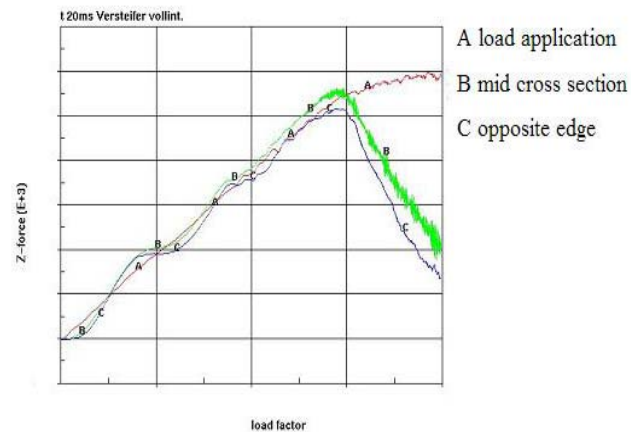


Fig. 14. Load-Displacement-Curve in an Explicit Dynamic Analysis

4 Results

4.1 General

Following computation results may be viewed in the dedicated post processor of the virtual test rig, which constitutes one of the main items in the selection tree. Particularly it addresses aircraft-specific results. Several of these may be displayed in conjunction with test data, which may be read from file. Results can be divided into global and local results described in the following.

4.1 Global Results

Figure 15, 16 and 17 show the skin deformation (buckling field) of the FE-analysis in comparison to the test results. The scaling of both contour plots is equal; it shows that both the amplitudes and the principal direction of the buckles agree very well.

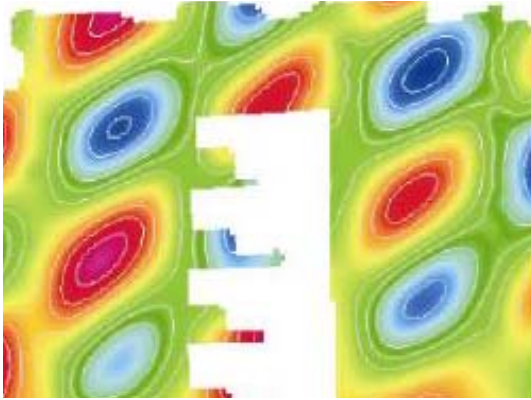


Fig. 15. Measured Buckling Pattern in the Physical Test

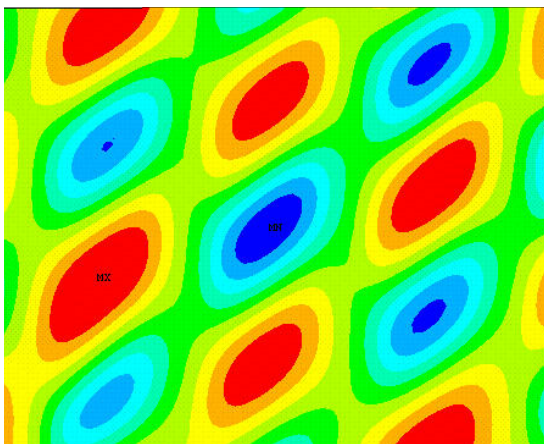


Fig. 16. Buckling Pattern in FE-Analysis



Fig. 17. Buckling Pattern in the Physical Test (Photo)

4.1 Local Results

The load-deflection-curves in figure 18 show a comparison between local deformations of specified nodes in the test result and the numerical solution. The green curve, meaning the test results, and the blue and the purple one,

representing the analysis results, agree very well. However, if one moves the specified points in the analysis by one element length, the result changes significantly, indicating an extreme dependency on the location of the measurement points.

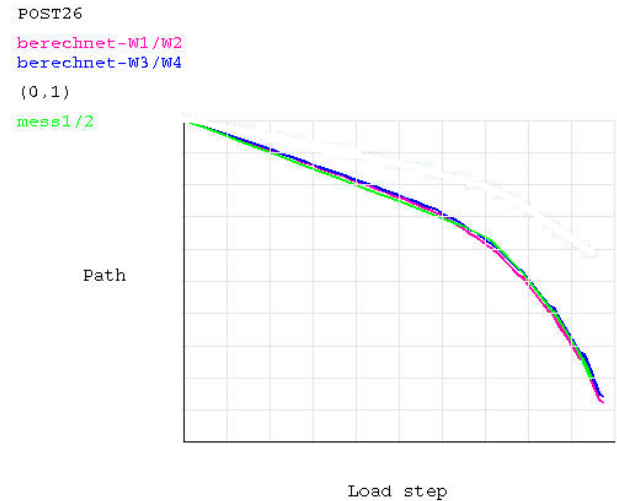


Fig. 18. Local Load-Deflection Diagram, Comparison of Test- and FE- Results

5 Summary and Conclusions

5.1 Summary

It was shown how a virtual test rig for simulation of stiffened aircraft fuselage panels was developed. The motivation for the development: expensive and time consuming physical tests should be partially replaced. The finite element based numerical models of the panels were presented, including elements, material models, as well as boundary conditions for test. The implicit (ANSYS) and explicit (LS-Dyna) solvers were discussed. Aircraft specific results obtained in the virtual test rig were presented divided into global and local results, of which several may be presented jointly with test data.

5.2 Conclusions and Recommendations

The studies presented in this paper allow for the following conclusions:

It has been shown that FE results of ANSYS agree very well with the test results.

However, during solutions it was noticed that the structure reacts very sensitively to the modelling of the boundary conditions.

The practical usefulness of the development was tested positively in the meantime. It is recommended that these studies be complemented with further development. Such development could comprise extension to further test rigs, and to other loading such as frame bending.

Acknowledgement

Dr. Clarice Carmone of Bishop Aeronautical Engineers, Hamburg, is acknowledged for finalizing the manuscript, and for careful proof reading.

References

- [1] Linde, P. Feasibility study: virtual structural test analysis system, Internal report, Airbus Deutschland, GmbH, 2002
- [2] ANSYS, Version 7.0, Users' Manual, Theory Manual, ANSYS Analysis Systems, Canonsburg, PA, USA, 2003
- [3] Rust, W., Schweizerhof, K.. *Finite element limit-load analysis of thin-walled structures by ANSYS (implicit) and LS-Dyna (explicit) and in combination.* Thin-Walled Structures 41, pp. 227-244, 2003.