BUCKLING AND POSTBUCKLING ANALYSIS OF A CFRP STIFFENED PANEL FOR A BETTER MATERIAL EXPLOITATION

Richard DEGENHARDT [1]
Jean-Pierre DELSEMME [2]
[1] DLR, Institute of Structural Mechanics, Braunschweig, Germany
[2] SAMTECH, s.a. Liège Belgium

Abstract: *Tthis paper will presents main results of post-buckling analyses led during the activity of the GARTEUR action group. A CFRP curved stringer stiffened panel under axial compression has been studied. The panel, tested by DLR, has been impacted inducing skin-stringer separation. Partners have carried out several analyses with various in-house and main commercial FE software to identify their abilities and deficiencies. The simulation tools and analysis methods will be compared together with available experimental results.*

Introduction

To build light and optimum airplane structures, stiffened panels are intensively used for fuselage, wings and other fins. Buckling and post-buckling is a major concern for such thin parts. To fully exploit the load carrying capacity, metallic panels have been studied for a long time. Buckling of elastic strait and curved shells (Timoshenko, and al.), effective width theory or diagonal tension for the approximation of postbuckling (Bruhn, Niu and al.), Ramberg-Osgood approximation of the plasticity are well known and used since half a century.
For CFRP panels, none of these simplified theories are available as it is. An alternative is certainly the numerical simulation through finite stripe or finite element theory. The finite stripe based method is well suited for simplified pre-dimensioning phase but the most general is of course the finite element one. The quite old but fast finite stripe method is gaining interest and has been and is used by some partners of the finished POSICOSS or the ongoing COCOMAT European project.
The finite element method is available in several main commercial tools but also in several "in-house" codes for research and development purpose. In 1999 the GARTEUR association (**G**roup for **A**eronautical **R**esearch and **T**echnology in **EUR**ope) initiated a specific action group to identify abilities and deficiencies of several tools by the comparison of simulation results to the experimental measurements. Three case studies have been investigated but only the third one, proposed by DLR, will be discussed here.

The GARTEUR Action Group 25

The action group 25 consisted of 10 partners. They came from large industries: QinetiQ (United Kingdom) SAAB (Sweeden) and AIRBUS (France); universities: Karlsruhe (Germany) and KTH (Sweeden); research establishments NLR (The Netherlands), DLR (Germany) and FOI (Sweeden); software vendors MECALOG and SAMTECH.

The benchmark 1, proposed by AIRBUS, consists of a compressed metallic stiffened panel buckling in the plastic domain. It has been analysed with RADIOS (MECALOG), B2000 (NLR) and SAMCEF (SAMTECH).
The benchmark 2, proposed by SAAB, consists of a machined aluminium beam made of 3 stiffened panels was loaded in shear. The aim was to find an optimum shape nowadays available with the help of fast tooling machines. It has been analysed with B2000 (NLR), SAMCEF (SAMTECH) and STRIPE (FOI).
The benchmark 3, proposed by DLR, consists of a CFRP curved stiffened panel loaded in axial compression. The panel has been impacted to induce a skin stringer separation. It has been analysed with ABAQUS and NASTRAN (DLR), FEAP (UKA), LUSAS (QinetiQ), and SAMCEF (SAMTECH).
The main results of the action group work are available in GARTEUR publications (see ref[1]).

The case study

The structure is a CFRP curved panel stiffened by 6 longitudinal stringers (see below). Practically, a full cylinder was first made and divided into 6 equal panels after curing. The internal stress release produced shape distortion and the real imperfections have been measured in 9 points on the skin. The panel has been impacted in 2 areas and the resulting skin-stringer delamination has been drawn by ultrasonic investigation.

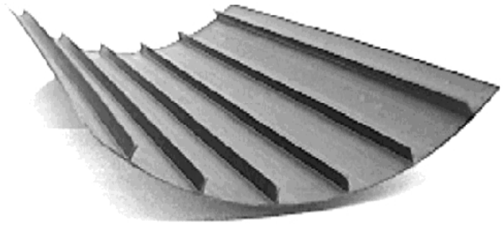


Figure 1 General view

To test the panel, the top and bottom edges have been clamped in boxes filled with gypsum. To prevent lateral buckling of free vertical edges, two sliders have been used. (see below). The circumferential displacement is let free.

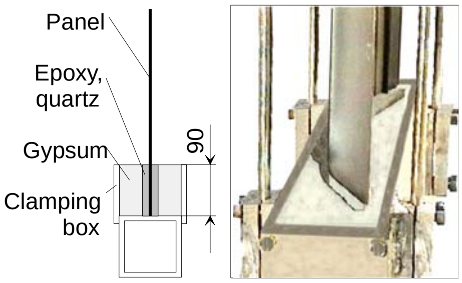


Figure 2 Load introduction device

The compression load was applied 67 times, the first 65 cycles were in the linear domain up to 80% of the first expected buckling load. Test 66 was stopped just after the first visible buckling and finally, the test 67 was led until collapse.

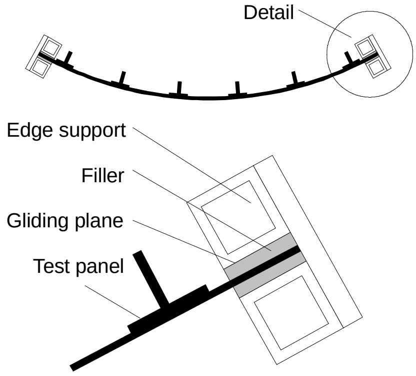


Figure 3 Lateral sliders

Strains were measured by 16 strain gauges bonded with orientations of 0° and 90° only on the skin. For measurement of out of plane deformations, 12 displacement pickups were mounted normal to the panel surface.

The simulation

Several types of simulations have been carried out and different tools have been used.
A linear static analysis has shown a good agreement between ABAQUS, NASTRAN FEAP (2d elements) and SAMCEF in the evaluation of the axial stiffness. Unfortunately, it was around 10 to 15 % stiffer than the experimental results. FEAP with 3d elements lead to exact value while LUSAS gave stiffness 13% less than measured one. To compare with an additional method, a simple computation with Classical Lamination Theory, gave a result not far the first FE simulation. An extensive investigation has been done to analyse the origin of this discrepancy but no clear reason have been found. The good agreement between the first results led to suppose that the problem is in the idealisation of the reality and not in the FE method itself.
Linear stability theory does not allow taking contact into account, that's why buckling analysis have also been carried out with the model without damaged region. To obtain a first idea of the influence of the damage, analyses with skin stringer separation but without contact have also be done. With the intact model, among the partners, the first buckling load was in a range between 93.8 and 166 kN depending on the mesh refinement (660 to 15000 elements) and element type (shell or solid). The corresponding mode was most of the time a local buckling of the skin between stringers (see Figure 4). University of Karlsruhe has compared several models varying number of elements, element type, stringer idealisation, etc. Among the first buckling modes , also global modes have been found.

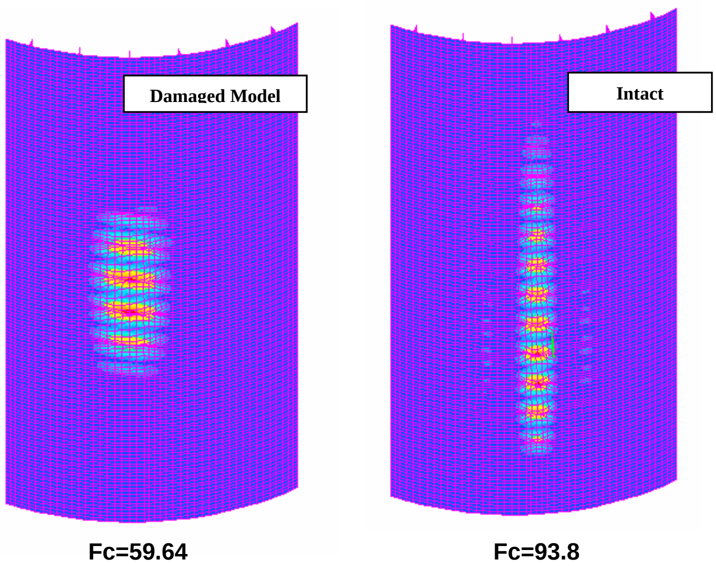


Figure 4 First buckling loads

It must be noticed that the first critical loads in general and in particular corresponding to the first local and first global mode were very close. To give an idea of the eigen-value spectrum, in a range from 0 to 3mm shortening, SAMCEF found 249 buckling factors well distributed over the range with a model made of 12320 shell elements with 50617 degrees of freedom. Among these values, some of them, around 2.2 mm, were related to more global mode.
With skin stringer separation but without contact condition, the first buckling mode was of course located around the damaged area (see Figure 4). Although the corresponding buckling load was underestimated (59.64kN) it was in a good agreement with NASTRAN (66.6kN). The approximation was due to the fact that the penetration between debounded parts was not constrained.

The most important part of the work was the non-linear simulation of the buckling and post-buckling behaviour. Since all the tools were not able to simulate contact conditions, in a first part, the model without degradation has been analysed.

Figure 5 shows the load shortening curves for some simulations in comparison with experimental results. The various analysis characteristics are the following:

1. DLR used ABAQUS for a dynamic analysis with an explicit scheme.
2. QuinetiQ used development version of LUSAS with a non-linear static analysis and an implicit scheme.
3. SAMTECH used SAMCEF/Mecano with a non-linear static and dynamic analysis and an implicit scheme.
4. University of Karlruhe used an extended version of FEAP with solid elements for static analysis with implicit schema.
5. University of Karlruhe also used shell elements for a dynamic analysis with an implicit scheme without damping.

The difference between initial stiffness discussed above appears clearly.

In the pre-buckling region, one can see the good agreement between shell models of analysis 1, 3 and 5. Dynamic analyses with ABAQUS/Explicit of FEAP/implicit allow exploring deep post-buckling region but for such methods the time step must be very small and the computation time increases dramatically. In SAMCEF/Mecano, a damping has been used to allow numerous modes jumping observed between the first buckling and the ultimate load. The damping, proportional to the stiffness, has been introduced through a Kelvin's like material. ABAQUS/Implicit offers a special capability that deals with damping and looks like the method used here in Mecano. DLR has observed with similar tests an important reduction of the computation time if one uses an implicit method with damping instead of an explicit method.

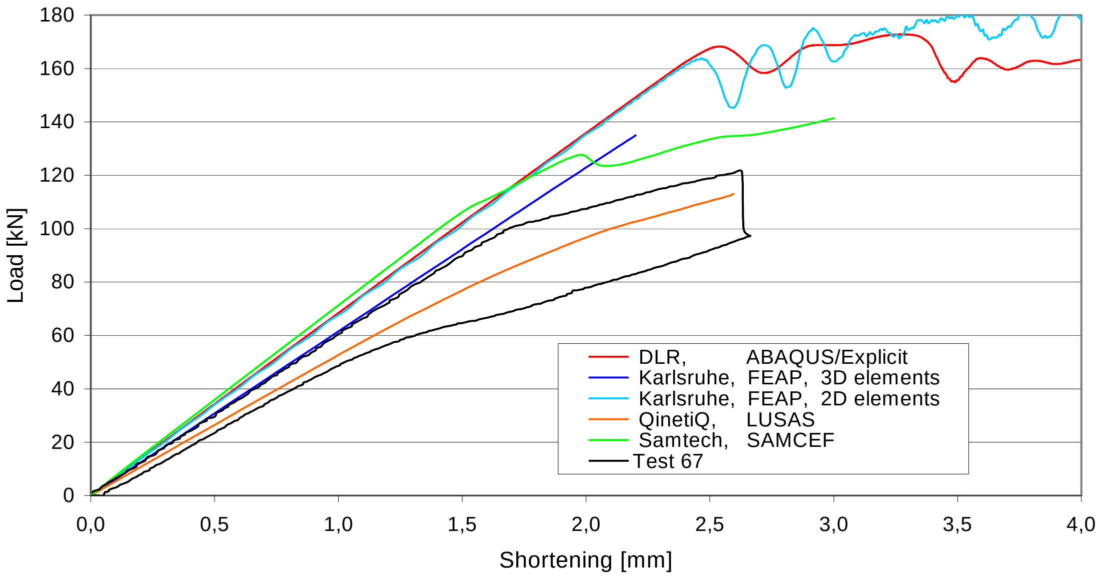


Figure 5 Load shortening curves for undamaged panel

Only SAMCEF/Mecano and ABAQUS/explicit have been used to study the damaged panel. Contact elements were introduced between separated skin and stringers. Figure 6

Gelöscht: <sp>

shows the load shortening curves for both simulations. One can see a very good agreement up to the maximum load. Figure 7 shows transverse displacement contours on the deformed shape for 1, 1.6 and 2.7 mm shortening. The first local buckling is followed by a symmetrical mode and at the end, a non-symmetrical deformation.

Figure 6 Load shortening curves for damaged panel

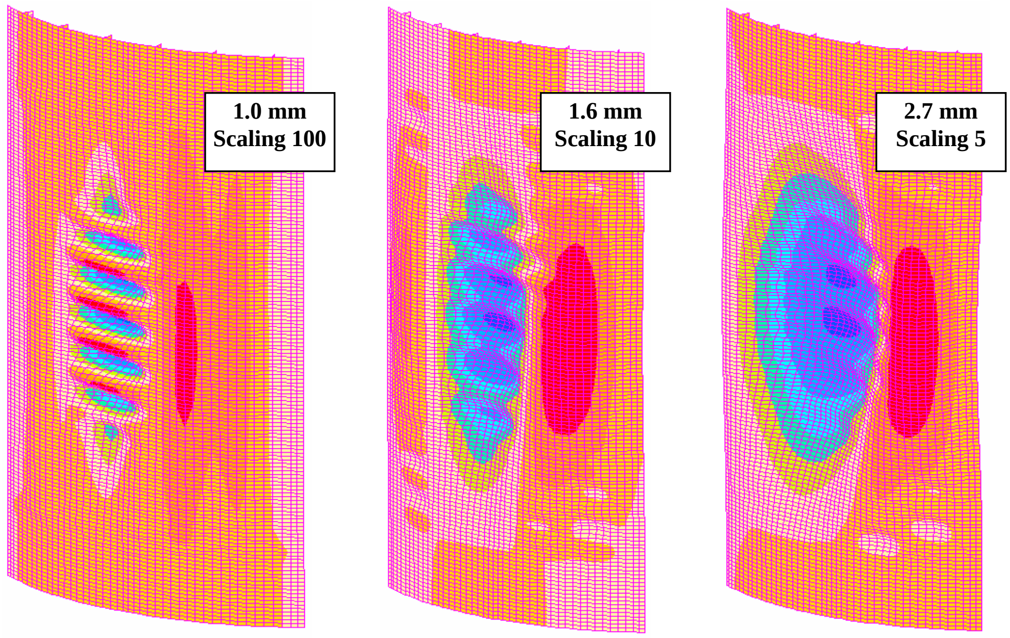
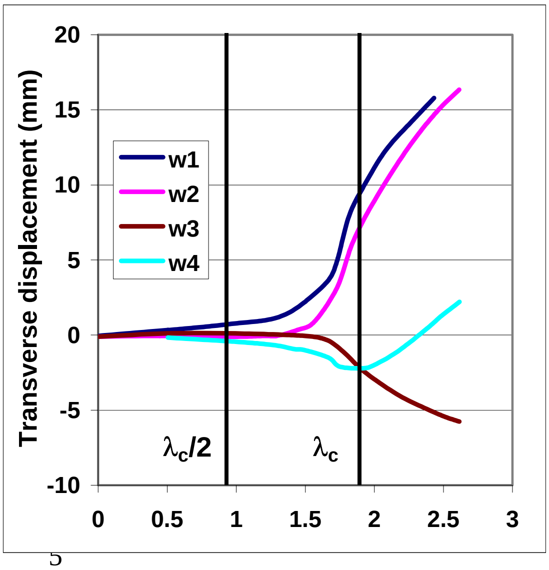


Figure 7 transverse displacement contours on the deformed shape

It is well known that imperfections have an important influence on the buckling behaviour of thin structures like curved panels. Figure 8 shows that measured lateral displacements are important and increasing from the beginning.

To assess the imperfection, SAMTECH has used a special procedure. It was assumed that lateral displacements were parallel to an imperfection mode amplified by a factor $\lambda/(\lambda_c - \lambda)$ if λ is a load factor and λ_c a critical value. One can see on Figure 8 that the behaviour, in a first time, is asymptotic to a vertical line located near 1.8 mm. It is clear that, for $\lambda=\lambda_c/2$, the scaling factor is equal to 1 and the imperfection shape match the displacement given by the curves. Among several solutions (e.g. combination of buckling modes) the deformed shape was computed by a static analysis with enforced displacement on 2 of the 4 points given by the curves for λ = 0.9 mm. This procedure is of course only possible after the test if the



lateral displacements are known; it has the advantage that contact conditions, introduced in the static computation, are satisfied.

Figure 8 Experimental lateral displacements on the equator

A more classical approach has been used by DLR. The real shape of the panel was measured on 9 points uniformly distributed on the outer surface. An analytical shape made of sinus and cosines has been adjusted to match the 9 points and the mesh has been modified accordingly.

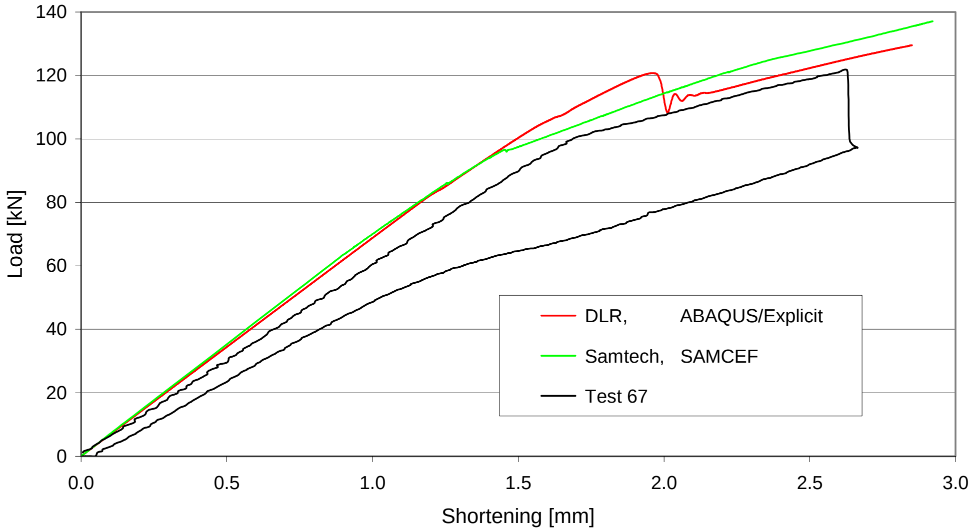


Figure 9 Load-shortening for imperfect structure

Figure 9 shows the load-shortening curves obtained with ABAQUS/Explicit and SAMCEF/Mecano with the mode including imperfection. With ABAQUS, the sudden mode jumping which happens near 2.6 mm without imperfection (see Figure 6) happens now near 1.9 mm. With SAMCEF, the mode jumping is completely avoided and one can observe only some knees in the curves corresponding to smoother path changing.

Conclusions

Several tools and analysis methods have been used to study the behaviour of a compressed CFRP stiffened panel including a damaged region. The simulation results were compared with experimental measurement data.

Most of the tools, using shell elements, have predicted an axial stiffness higher than observed. That doesn't necessarily means that shell elements are too stiff but only that some hypothesis made in the simulation were not coherent with the experiment. Other similar tests made in several other projects have shown a good agreement between simulation and experiment.

Contact conditions, are not compatible with linear stability analysis, that's why the undamaged panel has been analysed. There was a large variation between critical load predictions depending on the mesh size, type of element or stringer idealisation. It seams that to produce local buckling mode with small wavelength a fine mesh is necessary; a too coarse mesh will miss them and produce global buckling modes instead.

The linear stability of the model with skin-stringer separation has also been studied with NASTRAN and SAMCEF. Although that results were not physical they both found similar critical load and a local buckling mode located near the damaged region.
Since some tools were not able to simulate contact conditions, in a first time, non-linear analysis have been carried out with the undamaged panel.
Except the stiffness evaluation problem all tools have studied the pre-buckling phase. In the deep buckling stage, usual static analysis with Newton-Raphson method fails even with prescribe displacements. That's why alternative methods have been used:

- ABAQUS/Explicit has explored up-to 4 mm shortening with success, but the required computation time was very large while the choice of the time step size was very sensible.
- SAMCEF/Mecano has been used for static and dynamic non-linear analysis. To help the computation, an arbitrary structural damping has been introduced through a Kelvin's like material law.
- ABAQUS/Implicit have been used on similar structures for a non-linear static analysis with STABILIZE parameter. That type of analysis is much faster than the Explicit one (ratio of 5) but one have to be careful to choose the parameter value.

Path following methods like Riks or Crisfield have been tried with ABAQUS and SAMCEF. Unfortunately, none of these tools have been able to reach the post buckling stage. Actually, these methods are not recommended to study behaviours made of numerous local buckling.
Finally, of all tools considered, only ABAQUS/Explicit and SAMCEF/Mecano have been used with success to study the damaged panel without and with imperfection.

References

[1] Degenhardt R., Wagner W., Delsemme J.-P. and David A. (2003), GARTEUR Open, SM AG25, "Postbuckling and Collapse Analysis - Benchmark 3 Evaluation Report and Collected Contributions", IB – 131-2003/29, Braunschweig, DLR, 2003
[2] Degenhardt R., Rohwer K., Wagner W., Delsemme J-P. "Postbuckling And Collapse Analysis Of CFRP Stringer Stiffened Panels – A Garteur Activity", Fourth International Conference on Thin-Walled Structures ICTWS 2004, 22-24 June 2004