A comparison of finite element buckling analysis predictions and experimental results for riveted aircraft fuselage panels

K. Koffi, A. Gibson, M. Price
School of Aeronautical Engineering, Queen's University Belfast, Northern Ireland, UK.

Abstract

The prediction of the behaviour and the ultimate load for compressively stiffened aircraft shell structures is a very important non-linear analysis problem for engineers. Herein, finite element analysis and experimental tests are carried out for the buckling of aircraft stiffened fuselage panels. The calculated finite element results for both flat and curved bulbed-T panels correlated well with the experimental results. A change of mode shape is observed for the curve panel, which is very imperfection sensitive. This study reveals that the finite element code used, 'ABAQUS' is capable of modelling the behaviour of the buckling phenomenon for small aircraft fuselage panels.

1 Introduction

Shell buckling problems have been of great interest for aeronautical engineers for many years. The correct design of shell structures for aircraft must consider shell collapse as an important failure mechanism.

The collapse of shell structures is a very complex non-linear analysis problem. Since the early stages of aeronautical development, a substantial volume of research [1] has been conducted to understand this complex phenomenon. An abstract of these first works was presented by Hoffin [2] in 1966. The traditional analytical methods [3] used to predict the strength and behaviour of riveted aircraft fuselage panels are semi-empirical [4-5] and are inherently conservative. This conservative analysis leads to excess weight.
With the development of special or general computer codes [6-7], designers now have the necessary tools for further insight into the complex behaviour of aeronautical shell structures and therefore, a more accurate analysis and a better load prediction.

This paper deals with the problem of finite element non-linear buckling analysis prediction of aircraft riveted fuselage panels and the comparison with experimental tests. This work is an extension of the assessment reported by Lynch [8], which centred on the compression buckling analysis of flat aluminium alloy sub-panels comprising one stringer riveted to a given width of skin. In that work, the finite element prediction of specimens stiffness, buckling modes and strength correlated well with experimental results. The current paper considers compression panels comprising 3 stringers with both flat and curved skin. Two types of stringer commonly employed in the aircraft industry, bulbed-T and Z-section, have been investigated, however in this paper only the bulbed-T stringer panels (flat and curved) will be presented. The numerical computation is performed using commercial structural analysis software ABAQUS [9].

The ultimate aim of this work is to provide accurate analysis of complete sections of fuselage panels, which may be subject to various loading conditions. Experimental tests are being conducted in conjunction with the finite element analysis development in order to provide a benchmark against which the finite element analysis can be assessed.

2 Finite elements analysis

2.1 Type of elements.

There are several different elements in ABAQUS for the modelling of shell structures. Previous work [10] has shown that the S8R5 element is the most suitable for accurate buckling analysis of panel shell structures. This element is a curved quadrilateral second order, 8-noded small strain thin shell with reduced integration. This element uses five degrees of freedom per node. The skin, stringer flange and web are modelled by S8R5 elements (see figure 1). For the flat and bulbed-T stringer, a circular beam, connected to the stringer web by a rigid beam represents the bulb (figure 2).
2.2 Stringer/skin connection modelling

The fastener is modelled by a rigid beam (Multi-Point Constraint) linking a node of the stringer and the corresponding node of the skin (see figure 2 below). Two models have been investigated. In the first model, (a), the contact between the stringer and the skin is not modelled. In the second model, (b), the contact between the stringer and the skin is represented by gap elements preventing interpenetration of the two surfaces. This second model is used in this paper.

![Stringer/skin interface modelling](image)

**Figure 1:** Example of meshes used for panel modelling.

**Figure 2:** Stringer/skin interface modelling
2.3 Loading, boundary conditions and analysis procedure

The loading and the boundary conditions are chosen in such a way to be as close as possible to the experimental conditions. A uniform axial displacement of -0.08" is applied to the nodes at the top of the panel. The bottom of the panel is fixed in the Y direction. The uppermost and lowermost horizontal rows of nodes situated in a distance range up to 1" from the panel's edges (respectively the top and bottom edge) are restrained in the X and Z directions to represent the parts of the panel cast within the tooling resin. Figure 4 shows the experimental boundary conditions used. The numerical analysis is conducted in a two-stage process. First, an eigenvalue buckling analysis is performed to determine the buckling loads and the corresponding mode shapes for each panel. Second, the perfect initial geometry is seeded with the shape of the first buckling mode previously determined. The post-buckling analysis is then performed on the imperfect model by a static non-linear (Newton-Raphson method) analysis or by Riks method [11]. The computation is stopped when the displacement at the top of the panel exceeds 0.08", since at this value the panel has effectively failed.

3 Experimental tests

3.1 Test specimens

The detail of the stringer's geometry is shown on figure 3. Two types of panel are investigated: flat and curved bulbed-T as shown on figure 4(a) and figure 4(b). All the dimensions are in inches. The panel consists of a skin 18" x 13.5" and 3 stringers. For the curved panel, the radius of curvature is 53", which is approximately the radius of the fuselage of Global Express business jet aircraft of Bombardier Aerospace. The stringer is fastened to the skin by two rows of 5/32" diameter MS20426 countersunk rivets. The ends of the panel are embedded in a tooling resin, which is a mixture of Araldite CY 219, Hardener HY 219 and Accelerator DY 219. The panel shell is therefore supposed to have fully clamped conditions at its ends.

![Figure 3: Bulbed-T stringer detail geometry](image)

Figure 3: Bulbed-T stringer detail geometry
Figure 4(a): Geometry of flat bulbed-T stringer panel

Figure 4(b): Geometry of curved bulbed-T stringer panel
The skin material is 2024-T3 clad aluminium alloy and the stringer material is 2024-T8511 aluminium alloy extrusion. The properties of the materials used can be seen in Table 1. The stiffness values in this table were determined by compression tests on a hydraulic testing machine. The grains of the skin and stringer material line of the longitudinal direction (Y-axis).

Table 1: Material properties

<table>
<thead>
<tr>
<th>Material</th>
<th>E (psi)</th>
<th>m</th>
<th>f\text{\textsubscript{n}}</th>
<th>\nu</th>
</tr>
</thead>
<tbody>
<tr>
<td>2024-T3</td>
<td>10.857x10\text{\textsuperscript{6}}</td>
<td>11.274</td>
<td>47329</td>
<td>0.33</td>
</tr>
<tr>
<td>2024-T8511</td>
<td>10.64x10\text{\textsuperscript{6}}</td>
<td>29.7</td>
<td>51927</td>
<td>0.33</td>
</tr>
</tbody>
</table>

### 3.2 Testing machine and test procedure

The tests were performed on a 25-ton hydraulic compression-testing machine. The end shortening displacement is measured by two linear voltage displacement transducers (LVDT) located near the edge of the test specimen. The test data (LVDT, load) is recorded by a data acquisition system, Spectra-DAS that scans every two seconds. Before applying the proper load, a small load of about 0.3kN is applied to the panel and adjustments are made to verify if the panel is perpendicular to the machine platform and also to ensure that the load is applied parallel to the neutral axis of the panel. A initial load of \(1/\text{3}\) of the ultimate load is applied to verify the elastic stiffness of the specimen. If the correct value is obtained, the load is decreased to a value of about 5% of the ultimate load. The final load is then applied at a very small incremental rate of about 0.1kN/s until failure occurred.

### 4 Results and discussion

#### 4.1 Mode shapes

The numerical analysis showed closely spaced buckling eigenvalues, which is the manifestation of an imperfection sensitive structure, even though the mode shapes are radically different. The mode shapes observed for the flat panel are different from those of the curved panel. The computed first 4 buckling eigenvalue of the flat panel are respectively 0.10435, 0.10605, 0.11917 and 0.12434. For the curved panel the values are respectively 0.17441, 0.17462, 0.18407 and 0.18453. The finite element computed skin buckling load for the flat panel is 21.65kN and that of the curved panel is 36.6kN. The skin buckling load for the curved panel is 69% higher than the flat one. The curved panel is more resistant as expected. This result is confirmed by the experimental test. Figures 5a and Figure 5b show the first mode shape for the curved and flat bulbed-T panel respectively.
Figure 5a: Curved bulbed-T stringer panel, mode shape 1, skin buckling: 36.6kN

Figure 5b: Flat bulbed-T stringer panel, mode shape 1, skin buckling: 21.65kN
4.2 Results and comparison

4.2.1 Flat panel
The numerical and experimental results for the flat bulbed-T stringer are presented in Figure 6. The finite element analysis presents a good correlation with the experimental result in the elastic and postbuckling zone. The two curves correlated well with the experimental elastic stiffness curve until the local skin buckling occurred. There is however a discrepancy between the ultimate loads. The finite element analysis under-predicted the failure load, the numerical result is 9.77% lower than the experimental one.

Figure 6: Endshortening versus applied load for flat bulbed-T stringer panel.
4.2.2 Curve bulbed-T stringer panel
The curved panel is more complex due to the fact that it is imperfection sensitive. The imperfection value used here is 10% of the structure skin thickness. The experimental result (see Figure 7) showed a sudden change from deformation mode 1 to deformation mode 2 in the panel buckling behaviour. This behaviour is characteristic of curved or cylindrical panels and may indicate the presence of a bifurcation point. The presence of a bifurcation point in a buckling analysis is the manifestation of multiple solutions and implies loss of uniqueness of solution. Due to presence of geometric imperfections, there is apparently a deformation mode preferred by the structure. This sudden change is also observed in the numerical result, giving confidence in the accuracy of the numerical analysis. Figure 7 also shows a good correlation between the numerical and the experimental result. As in the case of the flat panel, the failure load predicted by the finite element analysis is lower than the test value, in this case by 11%. Both the numerical and the experimental result indicate that the two panels have approximately the same failure load.

![Figure 7: Endshortening versus applied load for curve bulbed-T panel.](image)

5 Conclusion
This paper presents a comparison of results between finite element analysis and experimental test. Finite element analyses have been performed to determine the load carrying capacity of stiffened bulbed-T stringer fuselage panel.
The finite element results suggest that it is possible to simulate the non-linear behaviour of small fuselage panels. The good correlation with the experimental test demonstrates that finite element analysis can predict accurately the buckling behaviour and the design loads for aircraft fuselage structures. The discrepancies between the numerical analyses and test results are respectively 9.7% and 11% for flat and curved panels. This paper also indicates that the curved panel is imperfection sensitive. A sudden change of deformation mode shape is observed for this panel. A high value of initial imperfection can have a great effect on the behaviour of curve panel. The value of imperfection used in this paper (0.1 times the panel skin thickness) is commonly encountered in aerospace industry, therefore the results obtained can directly be used for the failure load prediction of panel shell structures. Some meshing and boundary conditions futures are on investigation to improve the numerical solution. This work was carried out under the Industrial Research and Technology Unit (IRTU) start programme ST198. Their support and the support of our industrial sponsor Bombardier Aerospace Shorts is greatly appreciated.

References

[4] Bruhn, E.F, Analysis and design of flight vehicle structures, Tri-state offset company, USA.