This is the author version of the following:
“Enhanced Buckling Performance of a Stiffened, Variable Angle Tow Thermoplastic Composite Panel”
Which appears in
The final published version is available at
https://doi.org/10.2514/6.2018-0480
Enhanced Buckling Performance of a Stiffened, Variable Angle Tow Thermoplastic Composite Panel

Robert Telford¹, Daniël Peeters², Vincenzo Oliveri³, Giovanni Zucco⁴, David Jones⁵, Ronan O’Higgins⁶, and Paul M. Weaver⁷

School of Engineering and Bernal Institute, University of Limerick

Variable stiffness composites are exciting emerging structures capable of improving structural performance through tailored load redistribution. This technology is particularly relevant to aerospace structures, such as aircraft wings, which rely on stressed skins to resist compressive, buckling loads. Variable Angle Tow (VAT) composite laminates manufactured via tow steering can increase buckling capacity of composite structures, leading to reduced material weight and costs. Numerical models have progressed to the point whereby this technology can be explored for complex aerospace structures. Further progress can be made through incorporating the latest manufacturing methods with simple and representative testing techniques to analyze buckling performance and benchmark numerical models.

This work aims to analyze the buckling performance of a stiffened VAT panel using a novel test method. Laser assisted tape placement is used to manufacture the panel using thermoplastic composite tape, improving manufacturing accuracy and speed. The buckling response of this component is then tested using a newly developed three-point bending test method. The test method was designed using finite element models, experimentally validated, and the results were compared against a numerical model (based on the Ritz approach). It was found that the developed test can produce buckling in the skin, with the buckling mode matching that of the numerical model.

I. Introduction

WEIGHT is a prominent factor in the design of all aircraft structures. Removing weight from an airplane reduces the fuel required to fulfil its missions, which has the knock-on effect of reducing the loading of key structural components such as the wing spar or landing gear. Consequently, these components can be resized to consider the reduced loading conditions, which results in further reductions in weight and thus further reductions in fuel consumption. The benefits from continuing this cycle can be significant, including improved payload-range capabilities, as well as lower operating costs and carbon emissions from reduced fuel burn.

One method of reducing aircraft weight involves tailoring structures to resist only the loads to which they are subjected. This process eliminates excess material, thus reducing the weight of the component. An added advantage from this realization is a reduction in material costs and manufacturing times. In this regard, composite materials provide considerable potential. Carbon Fiber Reinforced Plastic (CFRP) is an exceptionally lightweight yet strong composite material, which combines stiff and strong fibers held together by a polymer resin material. Aligning the fibers along the primary load paths produces a highly optimized structure, whereby the directional properties of the fibers are being used to directly resist the loads applied. Because of this, they are now the material of choice for primary structures on all the latest large passenger transport jet aircraft (including the Bombardier C-Series, Boeing 787, Irkt MC-21 and Airbus A350)¹. In particular, wing and fuselage skins use CFRP panels stiffened by a stringer to resist tensile and compressive loads. These skins typically use straight fibers (i.e. their orientation remains constant within each ply), and the bonding of the stringer to the skin panel requires an additional manufacturing step (e.g.

¹ Postdoctoral Researcher, University of Limerick, robert.telford@ul.ie.
² Postdoctoral Researcher, University of Limerick, daniel.peeters@ul.ie.
³ Postdoctoral Researcher, University of Limerick, vincenzo.oliveri@ul.ie.
⁴ Postdoctoral Researcher, University of Limerick, giovanni.zucco@ul.ie.
⁵ Technical Officer, Irish Composite Centre (ICOMP), University of Limerick, david.jones@ul.ie.
⁶ Lecturer, University of Limerick, ronan.ohiggins@ul.ie.
⁷ Bernal Chair in Composite Materials and Structures, University of Limerick, and AIAA Member, paul.weaver@ul.ie.

American Institute of Aeronautics and Astronautics
adhesive bonding). This manufacturing step results in an increase in production time as well as material and processing costs.

Further optimization of these components is possible by combining the latest design and analysis methods with advanced manufacturing techniques and a novel testing solution. By steering the fibers of a composite within a layer, Automated Tape Placement (ATP) technology has made it possible to locally alter the stiffness of the structure to redistribute the loads and to resist spatial variations in stress. This tailoring approach is particularly suited to panels which are subjected to compression-induced buckling, such as the wing skin upper surface.

Maximizing the potential of VAT panels requires robust design tools, which can be benchmarked against representative mechanical tests. Traditionally, buckling tests use a pure compressive force to induce buckling (see for example, Bisagni et al., Jubiak et al.). Although widely used, this test method has some shortcomings. Firstly, the buckling modes obtained with this test are particularly sensitive to the boundary conditions at either end of the test specimen (where loads are applied). As a result, careful attention is required to ensure that the specimen is properly aligned during this phase. This task can be difficult with thin-skin composite test coupons, which are prone to manufacturing induced warping. This warping can lead to difficulties in correctly aligning the specimen.

This paper aims to enable lightweight composite stiffened panels by presenting a novel, simple test method to analyze the buckling performance of a steered, thermoplastic composite panel. A VAT composite laminate was manufactured using Laser Assisted Automated Tape Placement, (LATP). This manufacturing process facilitates highly optimized parts featuring fiber steering (referred to as Variable Angle Tow – VAT) to improve buckling response. In addition, the use of thermoplastic matrix material eliminates the need for autoclave processing, aligning with current industry trends for faster and cheaper parts manufacture. The manufactured panel is analyzed using an advanced numerical model developed to efficiently predict the buckled and post-buckled response of stiffened thin-walled structures with VAT fibre orientations. Finally, using a new 3-point bending test, the buckling load is characterized. The three-dimensional surface displacements of the wing skin was monitored during this test using Digital Imaging Correlation (DIC), enabling a full-field mode shape comparison to be made against the numerical model.

Objectives and layout

The objectives of this work are:
1. Develop new test method to analyze the buckling performance of stiffened skin panels
2. Apply this test to a stiffened thermoplastic CFRP panel featuring VAT
3. Compare experimental results to predictions from an advanced numerical model

These aspects are covered in the following sections. Firstly, the test method design is described in Section II, with Finite Element (FE) modelling techniques used to design the test set-up and panel geometry so as to achieve buckling. From these data, a VAT stiffened panel was manufactured for testing (Section III). A numerical model, based on the Rayleigh-Ritz approach, is then used to predict the buckling response for the experimental panel with both linear and non-linear regimes (Section IV). The test, performed using DIC to measure surface displacements, is described in Section V, and the results are given in Section VI. These results are discussed in Section VII, along with the developed test method.

II. Test method design

The test method was designed to create a state of compression in the skin of a stiffened panel. This is achieved by placing the stiffened panel in a bending test applying a line load at the middle of the skin surface, while supporting the bottom surface of the stiffeners' ends leads to compressive stresses being induced into the skin (see Fig. 1). With the correct loads, boundary conditions and panel geometry, these stresses result in buckling. This behavior relies on the neutral axis being shifted below the skin, leading to the skin section being subjected to compressive forces, with a slight linear distribution through the thickness.
FE simulations were used to validate the proposed test concept and to determine the boundary conditions and the panel geometry required to initiate compression buckling. A linear buckling analysis was conducted using ABAQUS—a commercially available FE solver. The model used shell elements (type S4R), with simply supported boundary conditions at either end (supporting all horizontal sections of both the skin and stringers). A structured mesh was used, with 103,800 square elements. This mesh density was chosen as it gave a converged solution. The loading was simulated by applying a displacement along a line at the center of the panel, which most closely resembles the actual loading conditions of the test. By altering the geometry of the panel, as well as the distance between loading sections, it was possible to design an experiment that will result in a buckling mode with four half-waves. Finally, a detailed three-dimensional FE model was used to analyze stress peaks which may exceed strength allowables.

3-point versus 4-point

The proposed test could be conducted by either using either a 3-point or a 4-point bending test. A numerical model (described in detail in Section IV) models were used to compare the buckling induced in a stiffened panel by both test set ups. A panel length of 600 mm was used, with a single displacement being used at the center of the panel for the 3-point bending scheme, and two displacements applied for the 4-point bend scheme. Composite omega-shaped stiffeners used were as per those in companion work⁹,¹⁰, while the skin had an 8-layer quasi-isotropic lay-up ([90/0/±45]s).
moment for buckling requires either an excessively long test panel, or excessively high loads (beyond strengths limits of the panel). In addition, simulations showed that the distance between supports in the 4-point bending test had a large influence on the buckling mode, particularly in the section between the loading points (i.e. the area of constant bending moment). This behavior can be seen in Fig. 2, where the position of the supports is moved from the free edge of the panel towards the center, as a percentage of the overall panel length. From an experimental point of view, using a 4-point bending set-up would also block the area of interest from the imaging equipment used for measurement of the panel displacement. Considering these factors, it was decided to keep a 3-point bend set-up.

This test leads to compressive load in the panel. FE models were once again used to verify the strains obtained. The transverse material strain, $\varepsilon_{22}$, of the top and bottom ply of the skin material are shown in Fig. 3. Since these are both 90° plies, the $\varepsilon_{22}$, perpendicular to the fiber direction, is shown. In ply 1, the top ply, the compressive strain is lower than in ply 8, the bottom ply. This result is as expected since the distance from the neutral axis is increasing. In real structures, this type of strain distribution is more common than constant strain through the thickness, which is obtained with a pure compression test.

Panel Geometry
The panel geometry was designed to result in a buckled response in the steered section of the skin between the stringers, and under acceptable loading conditions. The final dimensions of the panel are given in Fig. 4.

This geometry was chosen based on several factors. Firstly, the width of the steered section (i.e. the area between the stringers) was kept at 170 mm, and the lay-up of the skin was [90/0/ ±<35/53>]. These were based on the manufactured wingbox demonstrator, as discussed in companion work. Subsequently, the length of the panel was altered until the desired number of half-waves was predicted. The stiffeners used during this phase of the design were omega shaped, as discussed in accompanying work. When observing the number of half-waves in Fig. 5, it can be seen that, for the smallest length of 400 mm, only two half-waves are clearly present. From 600 mm onwards four half-waves are clearly evident. Increasing the length of the panel did not increase the number of half-waves, nor lead to a significant increase in amplitude. Consequently, a final length of 600 mm was chosen.
Subsequently, the stringer dimensions and materials were chosen. It was initially planned to use carbon fibre reinforced polymer stringers continuously wound using LATP and co-processed to the skin during manufacture (as in Oliveri et al.\textsuperscript{9}). However, due to time constraints, commercially available aluminum square section was used for the stiffeners. The section featured 40 mm side lengths, a 1.5 mm wall thickness, and was made from 6082-T6 aluminum. Compared to the original composite stiffeners, the aluminum stringers were sufficiently large to move the neutral axis below the skin to give a compression state, and were stiff enough so as not to influence the buckling loads or mode shapes (the buckling occurs in between the stiffeners). The stringers were placed at the edge of the steered portion of the skin (i.e. 170 mm apart). The final width of the panel was 320 mm, leaving a 35 mm overhang beyond the stiffeners. This design ensured that buckling occurs between the stringers and not on the sides of the panel.

For the fiber steering case, since the length between the stiffeners is the same as for the wingbox, the fiber angle distribution is chosen to be the same: starting at 35° on one edge, to 52° in the middle, and back to 35° on the opposite edge. Underneath the stiffener and on the sides, the fibers are not steered. Due to the steered layers requirement for a substrate material for steering, the outer layers were not steered. Therefore, these were chosen to be 90° and 0°.

An impression of the tow paths for the steered layers can be seen in Fig. 6. The sides of each tow are shown in alternate green and blue to show the gaps appearing in the steered section. The tow widths are 6.35 mm, as per the material used in the manufactured test article. The vertical lines correspond to the locations of the sides of the stiffener and the sides of the plate.

Strength considerations

The analysis predicted high strains in the skin-stringer interface at the points of load introduction. To reduce these strains to acceptable limits ($-2.5 \leq \mu \varepsilon \leq 2.5$), plates of 2mm aluminum were introduced between the skin of the stiffened panel and the loading frame. These plates featured round edges, to further reduce stress concentrations, and were kept 2 mm apart from the stringers (to avoid sharp contact during the test). In addition, the diameter of the radiused edges used for load introduction (i.e. the simply-supported boundary conditions) was set to 20 mm. The reduction in stress concentrations resulting from these measures was assessed using a 3D FE model. Solid brick elements (C3D8R) were used in combination with a Newton-Raphson algorithm. The panel features two axes of symmetry, which was confirmed by the result from the shell model. This symmetry was used to reduce the number of elements. Both the load introduction and support structures were modelled as cylinders. A front view of the final loading scheme is shown in Fig. 7(a). The analysis assumes a linear elastic behavior, with hard contact between the
panel and cylinders. The mesh was refined near the support and load introduction areas to capture the stresses more accurately in those regions. A close-up of the mesh can be seen in Fig. 7(b). The model was displacement-controlled, as per the actual test described in Section V.

Figure 7. Front view of the test set-up of the finite element model, (b) isogeometric view of the mesh used around the support structure.

The 3D FE model was then used to pinpoint any stress concentrations. For the aluminum stiffener, this was done by analyzing the von Mises stress. The maximum predicted von Mises stress was 146 MPa, well below the yield stress of the 6082-T6 aluminum alloy used in this study (250 MPa).

Finally, the stresses in the composite plate were assessed. It was found that the maximum values of $\varepsilon_{11}$, $\varepsilon_{22}$, and $\varepsilon_{12}$ did not exceed 1600 microstrain, and so no are within allowable limits. The out-of-plane stresses did pose a cause for concern. The maximum strain values predicted were 2500 microstrain ($\varepsilon_{13}$), roughly 2000 microstrain ($\varepsilon_{23}$), and -5700 microstrain ($\varepsilon_{33}$). The two out-of-plane shear strains have the same maximum and minimum value. The peak of $\varepsilon_{33}$ is only in compression; the maximum tensile strain is 1500 microstrain, which was deemed to be acceptable. The compressive strain of 5700 µε is high, and may lead to cracking of the laminate. However, it may also be a numerical issue, and so these results were deemed satisfactory to continue towards testing.

### III. Manufacturing of the Stiffened Panel

The stiffened panel was manufactured, as per the specifications given in Section II. The nominal dimensions of the panel are as shown in Fig. 4. The manufacturing process included two steps. Firstly, the skin section of the panel was manufactured using LATP. Secondly, the aluminum stringers were adhesively bonded to the skin. The material used was a 6 mm Carbon Fiber/Thermoplastic prepreg tape (TOHO IM7/PEEK). The material properties are provided in Table 1. The LATP system used comprised an AFPT thermoplastic tape laying head coupled to a KUKA KR-210 8-axis industrial robot. Thermal treatment of the material is achieved by means of a 3 kW laser controlled on a closed-loop system using a thermal imaging camera.

#### Table 1: Material properties for TOHO IM7/PEEK

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_{11}$ (GPa)</td>
<td>135</td>
</tr>
<tr>
<td>$E_{22}$ (GPa)</td>
<td>7.54</td>
</tr>
<tr>
<td>$G_{12}$ (GPa)</td>
<td>7.54</td>
</tr>
<tr>
<td>$v_{12}$</td>
<td>0.3*</td>
</tr>
<tr>
<td>$v_{21}$</td>
<td>0.021*</td>
</tr>
</tbody>
</table>

The skin section was manufactured directly onto a flat mold surface. As the section had free edges (i.e. it was not continuously wound), laying down some of the layers raised some challenges. Indeed, initial attempts to lay tapes directly onto the heated surface (280°C) were unsuccessful as the first steered layer (layer 3) did not bond well to the material beneath. Consequently, a different strategy was used. This procedure is described as follows:

1. The first two layers ($90^\circ$ and $0^\circ$, no steering) were laid onto a hot tool.
2. The tool was allowed to cool down, with the first two layers taped down to the tool during cooling.
3. The remaining layers were then laid down, with the tool at room temperature.
The first two layers went down on the heated table without a problem. During the subsequent cooldown phase to room temperature, the panel underwent thermal warping due to the unsymmetrical stacking sequence. Weights were placed on the skin at this stage to keep it flat. Subsequently, tape was used at the edges to keep the skin flat and the weights were removed. The subsequent layers went down as expected, but during the fourth consecutively steered layer (i.e., layer six in total), the tows started to wrinkle in the middle. This happened only locally, so is believed to have a small influence on the overall performance, but it should be investigated further.

The shape of the panel once removed from the tool can be seen in Fig. 8(c), with some warping evident. A few days following manufacture, it was noted that some delaminations had appeared near the center of the panel. To remove these delaminations and the warping, the panel was autoclave cured (380° C, 3 hours).

**Stringer bonding**

With the skin manufactured, the stringers could be adhesively bonded. The adhesive used was FM300-2; a thermally activated adhesive manufactured by Cytec. The process involved preparing the adhesive film, assembling the stringer panel with adhesive in place, vacuum bagging the assembly and autoclave curing of the adhesive. The panel was first trimmed to the nominal dimensions. As the alignment of the stringers in relation to the steered section is critically important, attention was paid to maintaining correct alignment during cutting. The surface of the panel where the stringers would be bonded was then lightly abraded and cleaned. This removed any freekote from the autoclave process. Likewise, the surface of the stringers was abraded and washed. The adhesive was then cut into 40 mm wide strips (same width as the stringers), and positioned onto the skin material. Two layers of adhesive film were used per stringer. The stringers were then placed onto the panel. At this stage, the stringer lengths were left oversize.

This process permitted them to be temporarily held in place to the toolplate using tacky tape. As a result, the stringers were restrained from moving during the following vacuum bagging process. Two aluminum plates of 170 mm width were used to keep the stringers the correct distance apart during the cure. A vacuum bag was then sealed to the toolplate and over the top surface of the stringers. This process kept the stringer sections open, exposing them to autoclave pressure internally and preventing them from collapsing.

Finally, the vacuum bagged assembly was placed in the autoclave for cure. The cure cycle followed the manufacturer specified instructions. A 2 hr dwell at 120°C was used, with a ramp rate 3°C per minute. Once cured, the stringers were ready to be cut to their final length. In an attempt to reduce vibrations (which could lead to delamination), the stringers were cut using a bandsaw, 10 mm away from the edge of the plate. Fig. 9 shows the stiffened panel following manufacture.
IV. Numerical Model to Analyze Buckling Performance

It has been shown in sections II and III that it is possible to manufacture VAT stiffened panels, and to qualify their performance by means of a three-point bend test. The final aspect towards the design and manufacture of lightweight stiffened panels is effective and efficient design tools. FE models provide exceptional flexibility in this regard, but at a significant computational cost. When considering VAT (variable stiffness), a large number of elements is needed in FEA to discretize this region. Numerical models can provide significant advantages in this regard as the same levels of accuracy are achievable with a large reduction in the degrees of freedom.

A Ritz-based analysis tool was used for the numerical model. This tool allows for the analysis of panels exhibiting general stacking sequences and subjected to different domain loads, membrane boundary loads, moments, prescribed displacements and thermal loading conditions. Linear buckling analyses have been carried out successfully for different configurations. The presented results follow convergence analyses carried out by varying the polynomial order for the variables approximation, however they are not described here for the sake of conciseness. The analyses have been performed by modelling the stiffened panel with 24 elements and assuming the same order of polynomial approximation, for all variables; the following results refer to the approximation scheme with N=M=8, where M and N are the number of polynomials in the two orthogonal direction respectively, which gives a total of 9216 DOFs. FE analyses were performed with ABAQUS, using S4R shell elements. This model is representative of the final panel.
geometry, and thus is different to that previously used to design the experiment (Section II). To model the fiber angle
distributions, a subroutine was implemented to generate meshes where each element has an independent constant fiber
orientation. Meshes with 14000 square elements and 100 different layups, for a total of 84234 DOFs, were used as
they provided converged results. The results from the linear buckling analyses are shown in Table 1 in terms of the
first buckling factor $\lambda_1$. As regards to accuracy, excellent agreement between the present results and finite element
analysis is observed.

\begin{table}[h]
\centering
\begin{tabular}{|c|c|c|}
\hline
$\lambda_1$ & ABAQUAS & Ritz & $D\%$ \\
\hline
1.7006 & 1.7004 & 0.01 & \\
\hline
\end{tabular}
\caption{Value of the first buckling load factor}
\end{table}

The comparison of the results show that the Ritz method can provide the same accuracy level as FE analysis with
a remarkably reduced number of unknowns. It is worth nothing that the proposed approach also simplifies the data
preparation as the introduced domain decomposition actually relates to geometrical modelling and not to a mesh-
like support for the approximation of variables. The first buckling mode of the VAT stiffened panel is shown in Fig.
10 for both the FE and the Ritz solution.

V. Testing

Apparatus

The 3-point bend test was conducted by installing a bespoke test frame into a Zwick 100 kN universal tester. The
test frame supported the panel during the test, with round edges being used to provide the simply supported boundary
condition described in Section II. The universal tester was then used to apply a displacement across the width of the
panel at its mid-length. A 25 kN load cell was used, mounted above the loading cylinder. The frame mounted in the
universal tester is shown in Fig. 11.

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{figure11.png}
\caption{Test frame and universal tester used}
\end{figure}

Measuring the surface displacements of the panel during loading allows for the buckling modes to be captured.
This task was achieved using a Digital Image Correlation (DIC) system. This system is capable of measuring three-
dimensional displacements, by tracking the movement of a number of dots (referred to as a speckle pattern) on the
surface of the panel during the deformation. The full-field nature of this technique means it is an ideal candidate for
observing the buckling shapes achieved during the experiment. Finally, the load cell channel was connected to the
DIC system for logging during the data acquisition phase. A StrainMaster DIC system supplied by LAVision was
used in this work.
Mechanical Test

Following calibration of the DIC system, the 3-point bend test could proceed. A displacement controlled test was chosen to give the best chance of recording any buckling events. The displacement of the loading head at buckling was expected to be 1.7 mm (from the linear analysis described in Section IV). As such, the experiment was allowed to progress to 2 mm of displacement, with data being collected during the loading phase. The DIC acquired images at a rate of 2 Hz throughout the test. Following an initial test to this displacement, it was confirmed that the loads at load introduction did not cause excessive strains. With a buckling half-wave predicted near the center portion of the panel, the 2mm thick aluminum plate under the loading bar was cut, so as to only cover the area directly above the stringers. This consideration removed a restrictive boundary condition, which may affect the buckling modes achieved. The test was then repeated, this time until a displacement of 3.122 mm (6.7kN).

VI. Results

A contour map showing the out-of-plane displacements recorded by the DIC is shown in Fig. 12. The predicted buckling mode from numerical models is also shown for comparison. This model applied a 3.122 mm displacement, as per the second experiment, and simulated the load introduction through the aluminum plates above the stringer only.

It was noted during loading that the edge of the panel (in the location of the stringers) experienced relatively high out-of-plane displacements. In addition, the displacements about the length of the panel are not precisely symmetrical. Close examination of the panel revealed that the edges of the panel were curved slightly upwards, leading to the asymmetry and the large displacements during the test. It can be seen that the buckling mode predicted by the linear-static numerical analysis was also obtained by the experiment. This mode features a half-wave near the load introduction, which transitions into a half-wave of the opposite sense (in sign). It is worth highlighting at this stage that the numerically predicted buckling mode shows only the buckling response, and not the superimposed displacement from the static analysis. At this stage, displacement data could be compared to further validate the numerical models. However, these results are not presented here for two reasons. Firstly, the buckling displacements are not available from the linear-static analysis. Thus, a non-linear model would be required. Secondly, the initial DIC results from experiments showed some unexpected displacements at the end supports (which should be zero). Whether this is a test frame issue (e.g. unexpected deformation of the test frame or stiffened pane), or a measurement issue (e.g. inadequate speckle pattern) has yet to be determined. As such, the analysis of the displacement results can take place following further investigation. Nonetheless, the experimental result confirms qualitatively that: (a) the test configuration is capable of inducing buckling in the skin between the stiffeners; and (b) the numerical model is correctly able to predict the buckling mode.
Due to the imperfect nature of the structure, it was not possible to observe a clear bifurcation at the point of buckling. Rather, the structure behaved as a classically imperfect structure, whereby there is a smooth transition into the final mode shape. Fig. 13 shows the contour plots of the transverse displacements at four different load values during the formation of the buckle. The out-of-plane displacements along the midline (visible in the contour maps) at each load stage are also given. It can be seen at loading stage A, an initial imperfection (at the top right hand side of the panel) exists, which triggers buckling to occur. The curvature of the panel (shown by the mid-line displacements) shows that the panel is undergoing bending deformation, characterized by the parabolic profile. At stage B, the buckle mode is beginning to form, influenced by this imperfection. This is seen by the increased $z$ displacements transitioning from the right hand side of the panel. In C, the first signs of the final buckling mode are evident, with the shape of the dimple beginning to form. Indeed, a sharp change in gradient can be seen in the mid-line displacement profile. Finally, at stage D, the complete dimple of the buckle is visible, and the buckle can be considered to be settled. From hereon out, increasing the load only amplifies the transverse displacements. It is worth
highlighting that the predicted buckling load from numerical models was 2.7 kN, which is within the range of loads observed during buckling.

Figure 13. Experimentally measured development of the buckling mode. The contour map refer to out-of-plane deformation during four stages of loading (marked A-D). Given below are transverse displacements along a mid-line.

VII. Discussion

It was seen from experiments that the buckling mode closely resembles that predicted by numerical models. This correlation happens despite the presence geometrical and testing errors (for instance, during panel trimming, manufacture, stringer bonding, testing alignment, etc.). It is well known that buckling is sensitive to geometrical, structural and loading imperfections, and so the accurate buckling mode captured by numerical models is a promising result. Further detailed analysis is required to comprehensively compare models and experiments, which can be done through direct comparison of the surface deformations. However, care must be taken when interpreting the results obtained, as the experiment conducted resulted in out-of-plane displacements at areas where there should be none (support region). This response could lead to a potential design oversight in this experiment, in that the stringers (or some other parts) may have deformed elastically during the loading, resulting in a combination of bending and rigid-
body deformations being recorded. In this view, it is important to consider the overall deformation of the structure coupled with the displacement measurement techniques being used.

The test method used was particularly simple to set up, and successfully resulted in a buckled skin. Positioning the panel required only alignment with the test frame to ensure that it was correctly centered. This set-up is in contrast to testing a stiffened panel in pure compression that may require the ends to be cast into a block for alignment and gripping—a time consuming and critical step. In addition, the three-point bend test eliminates the need for grips, which can be a source of slippage and thus distorted load-displacement results. In this test, the accuracy of the experimental set-up largely lies in the test frame, which is easily manufactured to a high degree of accuracy using modern computer controlled milling techniques, resulting in good control over the test parameters. The simplicity of the experiment (e.g. reduced sensitivity to initial alignment) allows for detailed comparison against numerical models. In this regard, a non-linear analysis can be used to compare load-displacement curves as well as full-field displacements.

In terms of tailoring the compressive stress profile taken by the skin, the test is highly adaptable. Stiffened skin panels in aerospace structures are not typically loaded in pure compression. Rather, they experience bending which is resisted by the stiffeners. Depending on the loads, geometry and construction of the structure, this will lead to a certain stress profile being induced into the skin material. By tuning the height of the stiffeners (and thus the distance between the neutral axis and skin) along with the loads used, it is possible to alter the stress profile in the skin to resemble that of an actual structure. This can lead to more realistic assessment of actual buckling modes in aerospace structures, as well as providing a realistic benchmark for validation of numerical models.

With the results showing promise for the 3-point bend being used as a buckling test for stiffened panels, it is now worth exploring improvements that could be incorporated. Firstly, it was noted that the adhesive bond between the stringer and the skin is particularly important in this test, as all the bending loads from the stringers is transferred into the skin material through this joint. Should higher loads be required, ensuring good load transfer through shear could be done by also including mechanical fasteners. Future work aims to use the test method to benchmark the numerical model further. Once displacement data is extracted, it will be possible to compare the full field displacements, as well as the load/displacement required to induce buckling. In addition, the stress transfer between the stiffener and the skin will be improved by incorporating mechanical fasteners. This development will give a more robust experiment, more closely matching the perfect bonding condition between the stiffener and the skin used in modelling.

VIII. Conclusion

The buckling mode of a stiffened panel featuring VAT and an integrated stiffener was assessed using a 3-point bending test fixture to induce compression to the skin. The concept of inducing a state of compression leading to buckling by such a test was validated using numerical and FE modelling techniques, and is particularly suited to analyzing stressed-skin aerospace components. The experimental test performed confirmed that the method can produce a buckled state in a stiffened panel. The test was relatively simple to perform compared to traditional pure compression tests, as no critical and time-consuming alignment steps were required prior to testing. In addition, the bending stresses used to induce buckling resemble more closely those in actual aerospace structures compared to the pure compression induced by traditional tests. These factors provide a simple test configuration to obtain representative results for assessment and model benchmarking purposes. However, careful design of the experiment is required to ensure that no spurious deformations take place, leading to misleading displacement results.

A comparison of the buckling mode obtained experimentally and that predicted from numerical models showed good agreement between the two. Future work aims to further benchmark the numerical model by direct comparison the full-field stresses.

Acknowledgments

The authors would like to thank Science Foundation Ireland (SFI) for funding Spatially and Temporally VARIable COMPosite Structures (VARICOMP) Grant No. (15/RP/2773) under its Research Professor program. The authors would also like to thank ICOMP for its help with the LATP.

References


American Institute of Aeronautics and Astronautics


