Aerospace Technology Congress, 8-9 October 2019, Stockholm, Sweden
Swedish Society of Aeronautics and Astronautics (FTF)

Post-Buckling Analysis in Thin-web Laminated Composite Beams

Nolli, B. L., Cimini Jr, C. A., Pena, C. A.*

Federal University of Minas Gerais, Av Presidente Antônio Carlos, 6627, Belo Horizonte, Minas Gerais/Brazil
E-mail: nolli.bruna@gmail.com, carlos.cimini@gmail.com, cristiano.pena@embraer.com.br
*Embraer S.A, Rua Sete, 498, Belo Horizonte, Minas Gerais/Brazil

Abstract

Aeronautic industry is aiming to increase the efficiency and to reduce the costs of their aircrafts, in order to develop airplanes with better performance and lower fuel consumption. Researches have demonstrated that shear panels can carry a significant amount of load after reaching its initial buckling load. Consequently, exploring the post-buckling capacity of composite materials reinforced panels results in lighter and less expensive structures. For the metallic reinforced panels there is a consolidated methodology developed by NASA to calculate the diagonal tension, NACA TN2661. However, a theory for the panel's post-buckling behaviour in composite reinforced panels is yet to be developed. Therefore, the purpose of this study was to contribute to this new theory and focus on the influence of stacking sequence and fibre orientations on the post-buckling behaviour of composite reinforced panels. The main goals were to develop a method to build a FE model to represents the post-buckling behaviour of the composite reinforced panel in order to avoid having to use experimental results in future projects; study the variations in stacking sequence in post-buckling analysis; choose the material to build a reinforced panel that have the best behaviour during the post-buckling analysis: metal or composite. The results have shown that the FEM, considering the load, boundary conditions and materials described in this study, can represent the behaviour of the composite reinforced panel and its post-buckling behaviour. Also, after the comparison between six composite reinforced panels models with different stacking sequence, the layup that presented the lowest values for the failure indices was the one with +45 and -45 at the outside layers. This laminate was chosen to be compared with the metallic reinforced panel model, and this comparison has shown that the composite reinforced panel could withstand higher loads, so it is considered the best for the post-buckling behaviour analysis.

Keywords: Diagonal Tension, Post-Buckling, Finite Element Model, Composite Material

1 Introduction

The challenges of aeronautic industry are to save weight and to reduce the costs of their aircrafts through the combination of different materials and calculation methods. Applications of composite materials have grown, consequently, the challenges to find optimal designs and new developments emerged [1]. The use of composite materials to build panel's web at wings and fuselage where they are allowed to undergo an elastic buckling (diagonal tension state), is one way to achieve this goal [2].

It is possible to design thinner web structures made of composite material using the methodology that evaluates the panel post-buckling behaviour, comparing it to other methods, resulting in a lighter final product. This improvement in weight is feasible because the diagonal tension method allows the reinforced panel to buckle after a pre-defined load, but this is not the failure of the structure. At this moment, diagonal folds appear in the panel's web, and the shear forces that caused the buckling are resisted by tension in the web by the diagonal folds, and by compression in the stiffeners [3]. Therefore, the final structure can be designed to support tension loads, and the tensions allowable are greater than the compressions one, so the complete structure withstand greater loads.

For the metallic reinforced panel there is a consolidated methodology developed by the *National Aeronautics and Space Administration* (NASA) to calculate the panel's diagonal tension, NACA TN2661 [4]. This semi-empiric method was developed based in several tests performed with aluminium panels using different geometries and loads, and is widely used by aircrafts manufactures. However, a theory for the panel's post-buckling behaviour in composite reinforced panels is yet to be developed. There are some researches ([5] and [2]) trying to adapt the NACA TN-2661 [4] method for composite materials, making it account for the anisotropy of the material and corroborate the results with tests. Most of them used *Glass Reinforced Aluminium* (GLARE) to build the panel. Some other studies [6] were

DOI
10.3384/ecp19162019

based in modelling the reinforced panel in finite elements and compared the results with tests data. Yet it was used a one bay panel with unidirectional layup.

To improve the understanding of post-buckling behaviour in composite reinforced panels, the studies should be focused in the variables that affect this behaviour, like stacking sequence, and orientation of the layers. And to overcome the expenses with tests, the reinforced panels modelled in finite element should have more than one bay, vertical and horizontal stiffeners and a complete layup using layer with different orientations. Therefore, the main goals of this study were to develop a method to build a *Finite Element Model* (FEM) to represents the post-buckling behaviour of the composite reinforced panel to avoid having to use experimental results in future projects; study the variations in stacking sequence in post-buckling analysis; and choose the material to build a reinforced panel that have the best behaviour during the post-buckling analysis: metal or composite.

The methodology developed in this study consist in five steps described hereafter. The first step was to build a FEM for an aluminium reinforced panel. One side of the panel is fixed, and a force is applied on the opposite side. A linear static analysis, a linear buckling analysis and a non-linear analysis were performed.

The second step was a comparison between results from the hand calculation for diagonal tension using NACA TN2661 [4] method and FEM analysis, in order to guarantee that the model is correct. Then a comparison between the buckling loads obtained from the linear and non-linear analyses was performed. If the differences are smaller than the error previous defined, it is possible to continue to the next step. Otherwise, the FEM must be modified in order to achieve that error. By the end of this step the model and mesh are considered validated.

Step three is a verification of the use of the validated mesh to build a FEM for the composite reinforced panel. In order to perform the verification, it was built two models: one with the equivalent properties for the layup chosen, and the other one modelling each layer and using the material properties of the tape. The models result from linear buckling analysis, and from the non-linear analysis, were compared. If the differences are smaller than the error previous defined, it is possible to continue to the next step. Otherwise, the FEM must be modified in order to achieve that error.

Step four consists in select the stacking sequence and the angle of fibre orientations for the layups, which have to be symmetric and balanced. For each layup, a FEM is created modelling each layer and using the material properties of the tape. Also, for each layup it will be applied several different loads values at the panel, in order to evaluate the reinforced panel behaviour for different loads magnitude. A non-linear analysis for each model is performed, and the failure index of the step when the diagonal tension is complete, and the panel's web redistribute the compressive load to the stiffeners for all of them is compared. The layup that presents the lowest values for the failure index for all the loads is chosen to be the laminate having the best post-buckling behaviour.

In step five a metallic reinforced panel FEM with the same weight that the composite reinforced panel layup chosen in step four is built. Then compare the buckling load from linear buckling analysis, and the load step when the diagonal tension is complete, and the panel's web redistribute the compressive load to the stiffeners from the non-linear analysis, in order to determine which one has the best behaviour for post-buckling analysis.

2 Results and discussion

2.1 Metallic reinforced panel FEM

The first step of this study was to build a FEM for the metallic reinforced panel with horizontal and vertical stiffeners. The materials and dimensions for the panel and stiffeners, and the load applied in the structure, were taken from an example of diagonal tension calculation presented in [7]. A schematic 3D model of the reinforced panel components and dimensions is presented in fig. 1.

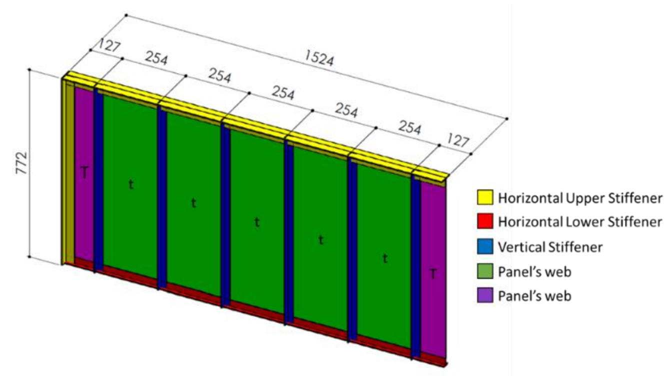

Figure 1: Metallic reinforced panel 3D model – geometry (all dimensions are in mm).

The panel's web was built with Aluminium 2024 T3, and the thickness for the panel shown in fig.1 as green is 0.635mm, and the purple one is 1.27mm. For the horizontal and vertical stiffeners, it was chosen Aluminium 7075-T3 Extruded.

The FEM was built using the software FEMAP 11.2® as pre- and post-processor and NASTRAN® as the solver. The panel's web was modelled using two-dimensions, four-nodes plate elements CQUAD4, which uses plane stress theory, capable of carrying in-plane forces, bending forces, and transverse shear forces [8]. The vertical and horizontal stiffeners were modelled using one-dimension elements that connects two grid points CBAR, which is a straight prismatic element with axial, bending, and torsional stiffness [8]. The CBAR element was chosen because the change in stiffener geometry using CBAR is easier to perform than it would be if it was modelled as plate elements. Consequently, the model is more versatile. Following the boundary conditions applied by [9], on the left-hand side of the panel, fixed boundary conditions were applied in the last column of nodes. Based on an example presented in [7], a force of 60075 N was applied in the –Y direction on the right-hand side of the panel. On the right-hand side of the panel, a condition of displacement only in Y direction was applied in the first column of nodes. Also, the adjacent elements of the CBAR elements, representing the horizontal stiffeners, had their thickness increased. Therefore, these elements represent the web thickness and the stiffener thickness together. The FEM is shown in fig. 2.

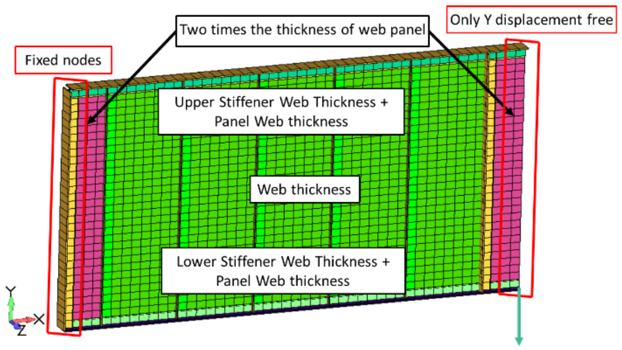

Figure 2: Metallic reinforced panel FEM final model.

It was performed a static analysis (SOL101) to evaluate the stress in the panel's web, a linear buckling analysis (SOL105) to calculate the buckling load, and a non-linear analysis including large displacements and elastic material properties (SOL106) to evaluate the post-buckling behaviour of the structure. The results obtained from the model and the hand calculation for diagonal tension using NACA TN2661 method [4] were compared.

The value of the shear stress from FEM was compared with the stress in the middle of panel (f_{max}) and the shear nominal stress in the web (f_S) in the region where appears the first eigenvalue (from linear buckling analysis SOL105). The comparisons are shown in fig. 3 and tab. 1.

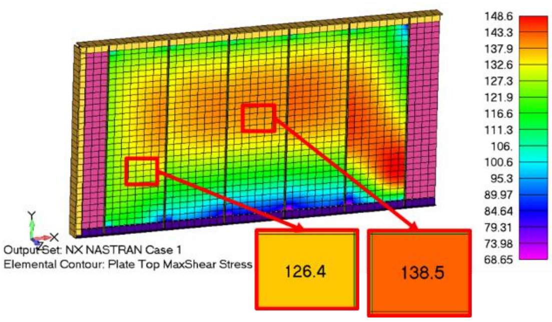

Figure 3: FEM stress (units in MPa).

Table 1: Comparison between shear stress from FEM and NACA TN2661 [4] method

FEM shear stress	NACA TN2661	Difference
138.5 MPa	134.2 MPa	3.2%
126.4 MPa	126.4 MPa	0.0%

Using the results from the linear buckling analysis (SOL105), it is possible to determine the buckling load (P_{cr_i}) for the metallic reinforced panel, shown in eq. (1).

$$P_{cr_i} = P_a\,\lambda_i = 60075N * 0.0227358 = 1365.85N \qquad (1)$$

Where P_a Is the load applied in the panel and λ_i is the eigenvalue calculated by the software FEMAP 11.2® in the linear buckling analysis (SOL105).

For the non-linear analysis (SOL106) the determination of the buckling load was performed plotting the non-linear load versus the total displacement of one node (it was chosen the one with greater displacement in the buckling region from the linear buckling analysis SOL105), as mentioned by [1]. On the plot load *vs.* displacement for the non-linear analysis (SOL106), a line A–A tangent to the first linear ramp was traced. Then a line B–B tangent to the second linear ramp was drawn where there is a significant change in the rigidity (fig. 4). Zooming into the non-linear region of the same plot, the buckling load was obtained by the intersection of these two lines, as showed in fig. 5.

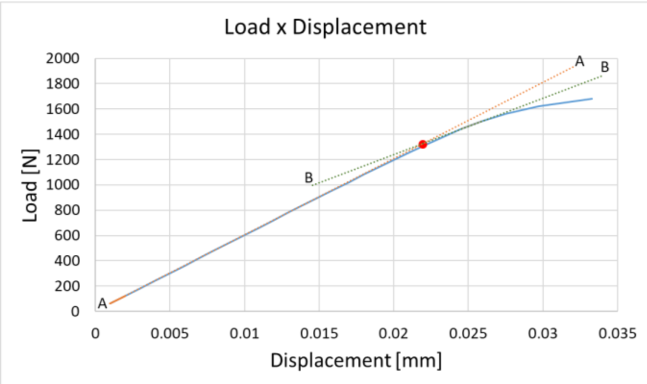

Figure 4: Non-linear analysis load-displacement graph

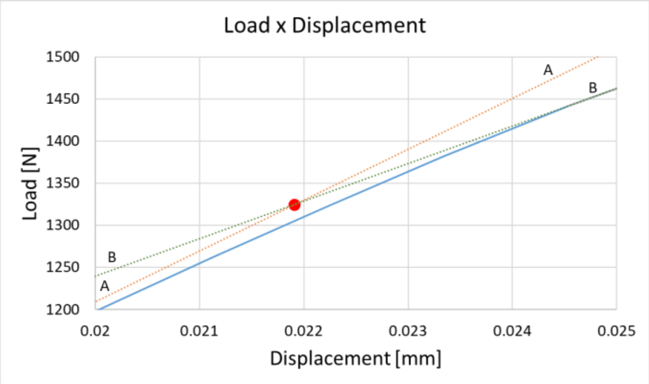

Figure 5: Non-linear analysis load-displacement graph (zoom in the region where lines A-A and B-B cross)

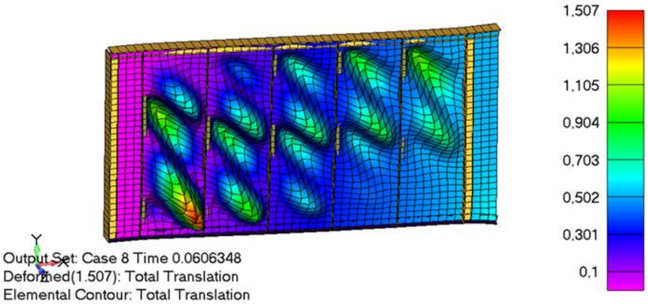

Figure 6: Non-linear analysis – complete diagonal tension (deformations are plotted to scale at 150 times their actual value)

The comparison between buckling loads obtained from the linear buckling (SOL105) and non-linear analysis (SOL106) is presented in tab. 2.

Table 2: Comparison between buckling loads from linear buckling analysis and non-linear analysis

Linear buckling analysis	Non-linear analysis	Difference
1365.85 N	1324.36 N	3.13%

The results from comparison between FEM and hand calculation for diagonal tension using NACA TN2661 [4] method, (tab. 1), and the comparison between linear buckling and non-linear analysis, (tab. 2), have shown that the greatest difference was 3.2%. These encouraging results create the necessary confidence to use the FEM model in the following analyses.

The reinforced panel post-buckling behaviour was also analysed from the non-linear simulation (SOL106). In this analysis, the panel total load (60075 N) was divided into 100 increments of 600.75 N, that were successively applied in the model. Therefore, at each 0.01 increment that occurs in the model, a load of 600.75 N is added to the reinforced panel [10].

The complete diagonal tension phenomenon was considered to occur when diagonal folders were visually formed in the panel's web. Figure 6 shows the load step where it occurred.

The complete diagonal tension load occurs when the panel's web redistributes the compressive load to the stiffeners. And this load is obtained by multiplying the step where first occurs diagonal tension by the total load applied in the reinforced panel. Consequently, this load is calculated in eq. (2):

$$0.0606348 * 60075N = 3642.64N \tag{2}$$

In order to check if this result was correct, two other verifications were performed. The first one was an analysis of the stress state of one element in the buckling region, using the non-linear solution (SOL106). Minimum Principal Stress (Min Prc), Maximum Principal Stress (Max Prc), and Maximum Shear Stress (Max Shear) were considered. Figure 7 shows these stresses plotted as functions of the applied load.

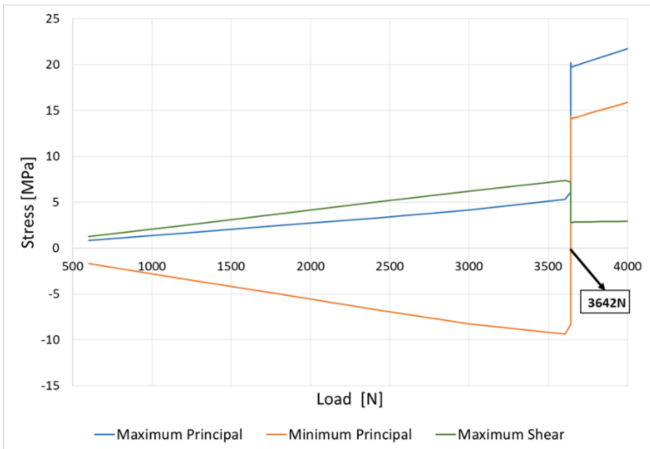

Figure 7: Stresses state during load increment

In fig. 7, all these stresses (Max Prc, Min Prc and Max Shear) experience sharp variations on their values at the load 3642 N. The explanation for these changes is the diagonal tension field. When the panel's web buckles and it cannot hold compression stress anymore, diagonal folds are formed in the web, which is further loaded only in tension. The shear stress is distributed for the stiffeners [3].

The second verification was an evaluation of the load that the horizontal stiffeners were carrying during load application. It was chosen four elements, two from the upper horizontal stiffener and the others from the lower horizontal stiffener, and the results were plotted in a graph. For the linear static analysis, the axial stresses in the linear element representing the stiffeners in the end of the analysis were taken. These values were plotted in the graph and a line was traced until the zero (at the load step zero the axial stress in the stiffener is also zero). For the non-linear analysis the axial stresses in the linear element representing the stiffeners were taken for each step of the analysis and these values were plotted against the load. The final result can be seen in fig. 8.

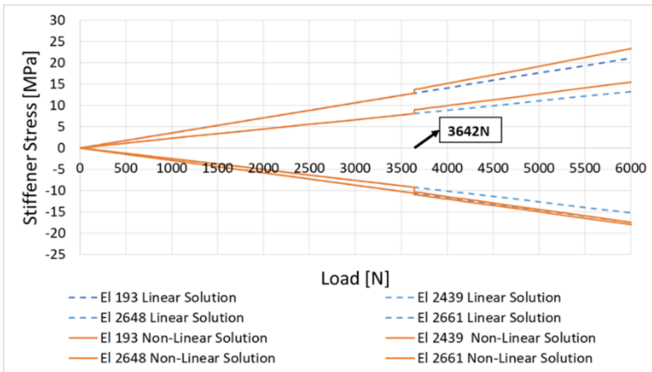

Figure 8: Stress on the stiffener with the increase of the load

The dashed line represents the axial stresses for the linear analysis, and the solid line represents the axial stresses for the non-linear analysis. It can be seen that for the load of 3642 N there is a discontinuity in the stress for the non-linear analysis, which means that at this load step the stiffeners start carrying more load than they would in the linear analysis. Therefore, this is the load step when the diagonal tension is complete, and the panel's web redistribute the compressive load to the stiffeners [3].

2.2 Composite Material Reinforced Panel versus Isotropic Reinforced Panel

The validated mesh aforementioned was used to build two FEM (with the same geometry) for the composite reinforced panel, model 1: modelling each layer and entering the elastic material properties of the tape, and model 2: with the equivalent properties for the layup selected. It was chosen a carbon/epoxy tape, $[45/-45/0/90]_S$, balanced and symmetric. The choice of using 0º, +45º, -45º and 90º fibre orientation is due to the recommendation made by [11]. They affirm that the laminate will be fibre dominant with the use of at least 10% of its plies in 0º, +45º, -45º and 90º. The balanced layups were chosen to remove the membrane coupling between in-plane normal and shear behaviour, and the choice of use symmetric laminate is to uncouple bending and membrane response, and to prevent warping under thermal loading [11]. They also affirm that laminates should be symmetric and balanced to maximize buckling strengths.

The mechanical and allowable properties of the carbon/epoxy tape used to build the layup came from [12]. The FEM material direction was aligned with the +X direction of global coordinate system, which means the tapes oriented in 0º were aligned with the +X direction.

The equivalent properties for the stacking sequence and tape material above-mentioned, is shown in tab. 3. Because the value of Ex is equal to Ey, an isotropic material was used in the FEM with the laminate equivalent properties.

Table 3: Equivalent property - Tape carbon/epoxy $[45/-45/0/90]_S$

Equivalent Properties	
Ex = Ey	56675.5 [MPa]
Gxy	22039.8 [MPa]
νxy = νyx	0.286

The materials for the reinforced panels can be seen in tab. 4. The material for the stiffeners were changed from aluminum to steel, and their thickness (web and flange) were doubled, and the load applied in the reinforced panel was altered from 60075 N to 143802 N. These modifications were necessary to guarantee that a post-buckling behavior would occur in the reinforced panel even with the increase in web's thickness from 0.635 mm to 1.52 mm. All the other panel's dimension (height, width, etc.) were kept the same.

Table 4: Reinforced panels materials

Material	
Web (model 1)	Tape carbon/epoxy $[45/-45/0/90]_S$
Web (model 2)	NA (it was used the Young's modulus equal to the composite model) E = 56675.5 MPa
Horizontal Upper Stiffener	Steel 4043
Horizontal Lower Stiffener	Steel 4043
Vertical Stiffener	Steel 4043

The results from the two models were compared, the first eigenvalue from the linear buckling analysis, and the load step when the diagonal tension is complete, and the panel's web redistribute the compressive load to the stiffeners from the non-linear analysis. The FEM plots are presented in figs. 8 and 9, and the comparison in tab. 5.

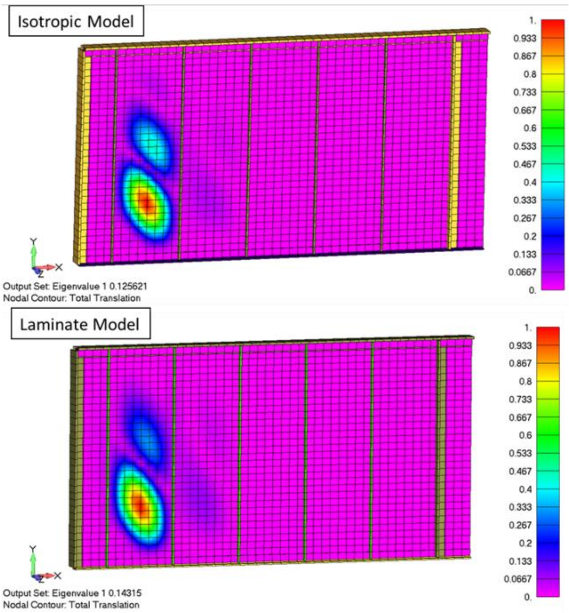

Figure 8: Isotropic and composite models: first eigenvalue (linear buckling analysis) comparison

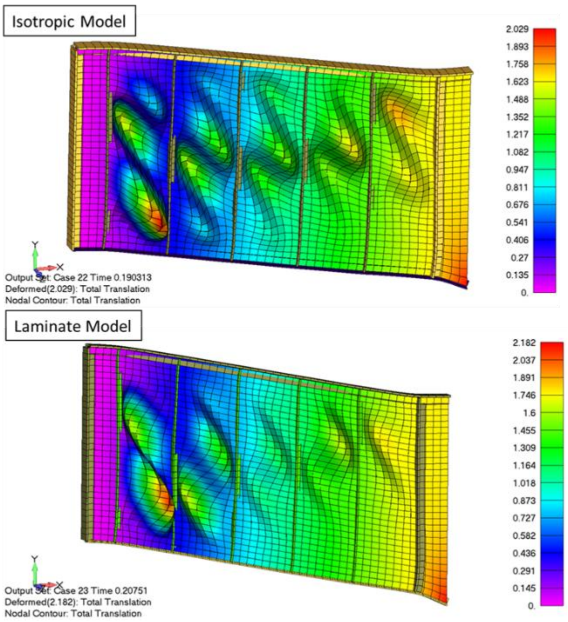

Figure 9: Isotropic and composite models: complete diagonal tension first occurrence (non-linear analysis) comparison (deformations are plotted to scale at 150 times their actual value)

Table 5: Comparison between Isotropic and Composite models.

	First Eigenvalue	First occurrence of Diagonal Tension
Isotropic Model	0.125621	0.190313
Composite Model	0.14315	0.20751
Difference	1.75%	1.72%

The results from comparison between Isotropic and Composite models, presented in tab. 5, show that the greatest difference is 1.75%. These encouraging results create the necessary confidence to use the FEM model in the following analyses.

2.3 Composite Reinforced Panel

Knowing that the FEM can be used to model the composite reinforced panel and to predict the post-buckling behavior, the model was used to choose the best laminate and compared it with the metallic reinforced panel. Six different laminates were selected to be compared, using carbon/epoxy (AS4/APC2) tape (same properties and allowable from [12]), all having 0º, +45º, -45º and 90º, being balanced and symmetric (the reasons for these choices were presented in Section 2.1). It was chosen to use the same material and fiber orientations and just change the stacking sequence because as mention by [13], the buckling modes are more sensitive to this type of variation. All the different stacking sequence are shown in tab. 6.

Table 6: Selected stacking sequence to be compared.

Model 1	$[0/90/45/-45]_S$
Model 2	$[45/-45/0/90]_S$
Model 3	$[45/0/90/-45]_S$
Model 4	$[0/45/-45/90]_S$
Model 5	$[0/45/90/-45]_S$
Model 6	$[45/0/-45/90]_S$

The reinforced panel dimensions and their vertical and horizontal stiffeners material and dimensions were the same as used in Section 2.2.

For each model it was applied seven loads, in order to verify the laminate behavior with different load magnitudes. For all models, it was evaluated the Laminate Max Failure Index using the Tsai-Wu criterion and Maximum Strain criterion, calculated by the software FEMAP 11.2®. The failure index was used to classify the stress state of the laminate and not to determine the failure of a lamina, therefore the failure indices values presented are not close to one (that indicates failure).

Because of convergence problems in the models, not all the loads used for one criterion could be used for the other. The loads applied in the model for each criterion are presented in tab. 7.

Table 7: FEM applied loads for each criterion

	Tsai-Wu	Maximum Strain
Condition 1	5500 N	5500 N
Condition 2	100000 N	100000 N
Condition 3	160000 N	160000 N
Condition 4	310000 N	310000 N
Condition 5	500000 N	480000 N
Condition 6	700000 N	700000 N
Condition 7	800000 N	900000 N

The results are presented in figs. 10 and 11.

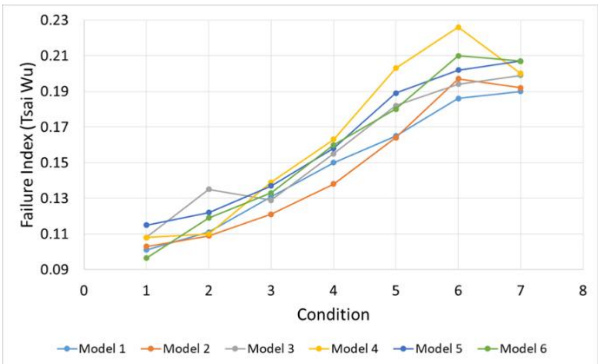

Figure 10: Comparison of the Tsai-Wu Failure Index for all the models

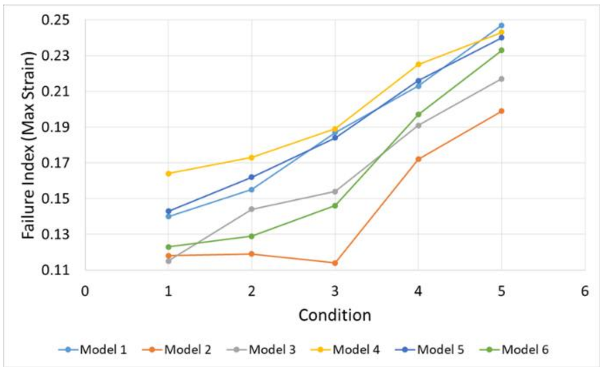

Figure 11: Comparison of the Max Strain Failure Index for all the models

Analyzing figs. 10 and 11 it is possible to conclude that the model that have the best behavior in the post-buckling is Model 2, because it presents the lowest failure index for the two criteria compared to all the other models. This result is corroborated by [11] that affirm for stability is better to use the +45º and -45º plies on the outer surfaces.

2.4 Composite Reinforced Panel versus Metallic Reinforced Panel

In order to verify which reinforced panel, metallic or composite, has the best behavior for the post-buckling analysis, the composite reinforced model chosen before (Model 2) was compared with a metallic reinforced model. Both models had the same mass and a load of 180000 N was applied in both panels.

The two reinforced panels use the same vertical and horizontal stiffeners, have the same boundary conditions applied, the web's thickness was the only parameter changed in the metallic one. Using the Aluminum 2524 T3 and Tape carbon/epoxy (AS4/APC2) densities, it was possible to calculate the web's thickness for the metallic reinforced panel which gives the same weight as the composite reinforced panel. Table 8 shows the thickness and mass for the two models.

Table 8: Thickness and mass for the composite and metallic models

	Composite Reinforced Panel	Metallic Reinforced Panel	Difference
Web's thickness [mm]	1.520	0.701	-53.87%
Mass [kg]	14.330	14.330	0.00%

The results from the two model were compared, the first eigenvalue from the linear buckling analysis, and the load step when the diagonal tension is complete, and the panel's web redistribute the compressive load to the stiffeners from the non-linear analysis. The FEM plots are presented in figs. 12 and 13, and the comparison in tab. 9.

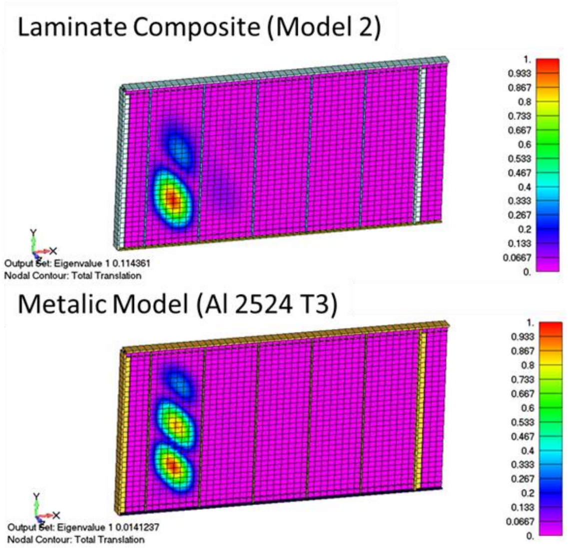

Figure 12: Composite and metallic models: first eigenvalue (linear buckling analysis) comparison

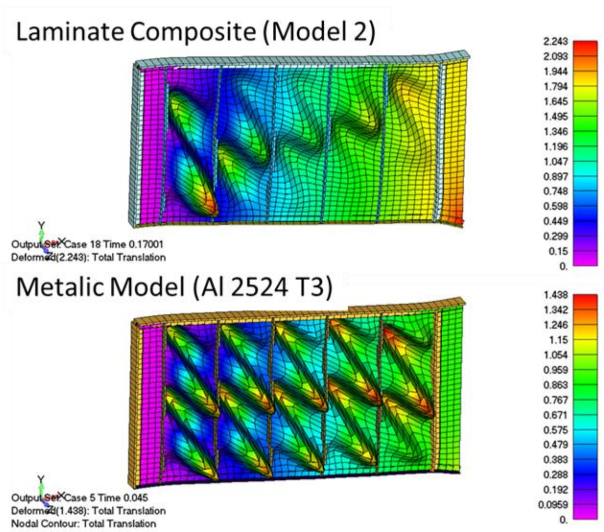

Figure 13: Composite and metallic models: complete diagonal tension first occurrence (non-linear analysis) comparison (deformation scaled in 150 times)

Table 9: Comparison between results from Composite and Metallic models

	First Eigenvalue	First occurrence of complete Diagonal Tension
Composite Model	0.114361	0.17001
Metallic Model	0.014124	0.04500

The comparison of the results of the two reinforced panels with the same mass, shows that the metallic reinforced panel supports lower loads before the buckling than the composite reinforced panel. This means that the web of the composite reinforced panel withstands to greater loads than the metallic one, consequently redistributes less load for the stiffeners. This result is corroborated by the analyses of fig. 14, which shows the comparison of stiffeners axial stress for both of the reinforced panels.

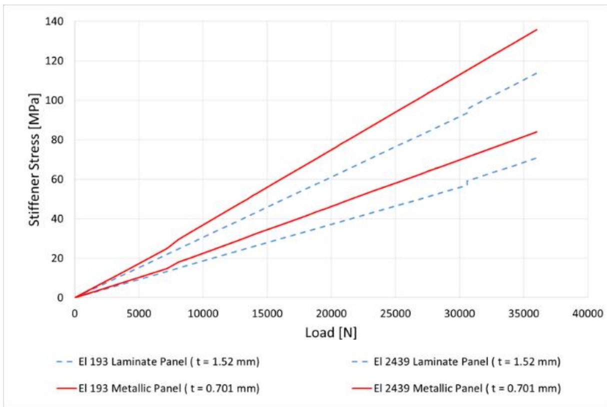

Figure 14: Stress on the stiffener for the composite and metallic models

Therefore, it is possible to conclude that the composite reinforced panel presents the best behavior for the post-buckling, in the conditions analyzed in this study (FEM boundary conditions, materials, etc.).

In order to determine the mass of the metallic reinforced panel for its behavior becomes similar to the composite reinforced panel, a new metallic reinforced panel was built. The web's thickness for the its panel was increased until its first eigenvalue and its load step when the diagonal tension is complete, and the panel's web redistribute the compressive load to the stiffeners got closer to the values for the composite reinforced panel. Table 10 shows the thickness and weight for the two models.

Table 10: Thickness and mass for the composite and new metallic models

	Composite Reinforced Panel	New Metallic Reinforced Panel	Difference
Web's thickness [mm]	1.52	1.51	-0.82%
Mass [kg]	14.330	17.368	21.20%

The FEM results for the two models are presented in figs. 15 and 16, and the comparison is shown tab. 11.

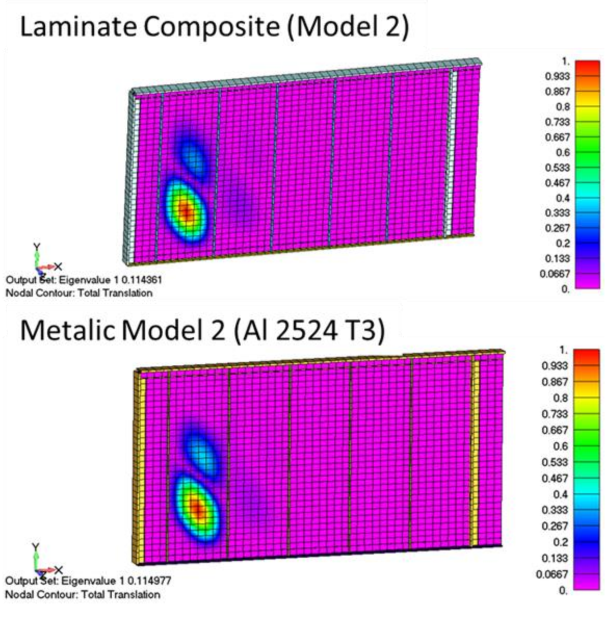

Figure 15: Composite and new metallic models: first eigenvalue (linear buckling analysis) comparison

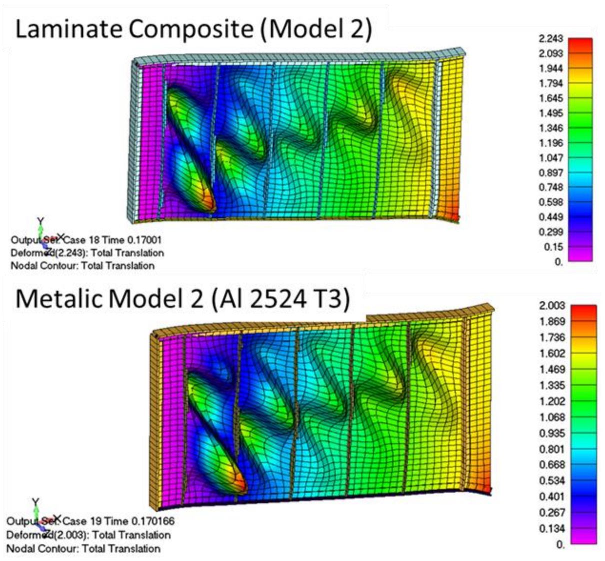

Figure 16: Composite and new metallic models: complete diagonal tension first occurrence (non-linear analysis) comparison (deformation scaled in 150 times)

Table 11: Comparison between composite and new metallic models

	First Eigenvalue	First occurrence of complete Diagonal Tension
Composite Model	0.114361	0.170010
New Metallic Model	0.114977	0.170166

3 Conclusion

The purpose of this study was to improve the understanding of the post-buckling behaviour in composite reinforced panels, by verifying the influence of stacking sequence in this performance. Moreover, evaluate if the composite reinforced panel have a better behaviour in the post-buckling analysis than the metallic one was also a purposed.

A Literature review revealed that most studies are trying to adapt the method NACA TN2661 [4], used to design metallic reinforced panels, for the composite ones, taking into account the material's anisotropy. But they do not consider the fact that the anisotropy is not the only characteristic of the material that affects the post-buckling behaviour. Stacking sequence, and fibres orientation have influence in buckling analysis, therefore they have effects in the post-buckling behaviour.

Therefore, the main goals of this study were to develop a method to build a FEM to represents the post-buckling behaviour of the composite reinforced panel to avoid having to use experimental results in future projects; study the influence of stacking sequence in post-buckling behaviour; choose the material to build a reinforced panel that have the best behaviour during the post-buckling analysis: metal or composite.

The methodology adopted was to build a FEM for the metallic reinforced panel, and to compare its results with the method NACA TN2661 [4]. The model was modified until the differences reached a value smaller than the error previous defined. Using the validated model, two FEM were developed for the composite reinforced panel, an isotropic one, using equivalent properties, and the other modelling each layer. The results for both models were compared to verify if the model developed in this study could be used to model a composite reinforced panel modelling each layer. Then it was compared the failure index for six models of composite reinforced panel, having the same material, but varying the stacking sequence. The one having the lower value of the failure index was considered to have the best post-buckling behaviour. This model was then compared with the metallic reinforced panel, having the same mass, in order to verify which panel has the best behaviour during the post-buckling event.

The results from this study have shown that it was possible to represent the behaviour of the composite reinforced panel during post-buckling using the FEM developed. For this FEM it was modelled each layer of the $[45/-45/0/90]_S$ laminate, which was balanced and symmetric. The boundary conditions and load used were described in this study.

Then six models with different layups were modelled, using the same material, the same total number of layers and the same number of layers for each orientation. The order of the layers was changed to evaluate the influence of the stacking sequence in the post-buckling behaviour. Comparing the failure indices for all the models in the moment when the diagonal tension is complete and the panel's web redistribute the compressive load to the stiffeners, it was concluded that the model that has better results, or lower failure index, is the one with +45 and -45 at the outside layers.

Two FEM were built for the metallic and composite reinforced panels, having the same mass, using the same stiffeners, only varying the web thickness. The comparison between buckling analysis and post-buckling behaviour shown the composite panel supports more load before the buckling than the metallic one. This means that the web of the composite reinforced panel withstands to greater loads than the metallic one, and consequently redistributes less load for the stiffeners. Therefore, it is possible to conclude that the composite reinforced panel presents the best behaviour during the post-buckling event.

References

[1] ARAKAKI, F. K.; FARIA, A. R. Composite-stiffened panel design under shear post-buckling behavior. Journal of Composite Materials, v. 50, n. 26, p. 3643-3662, 2016.

[2] JODOIN, A. et al. Diagonal tension in fibre-metal laminates. In: ICAS 2002, 23, 2002, Toronto. p. 7-8.

[3] NIU, M. C. Airframe Stress Analysis and Sizing. 2nd ed. Hong Kong: Conmilit Press Ltd., 2005. 810 p.

[4] KUHN, P; PETERSON, J. P.; LEVIN L. R. A Summary of Diagonal Tension: Part I – Methods of Analysis. NACA Tech. Note 2661: may, 1952.

[5] WITTENBERG, T. C.; VAN BATEN, T. J.; DE BOER, A. Design of fiber metal laminate shear panels for ultra-high capacity aircraft. Aircraft Design, v. 4, n. 2-3, p. 99-113, 2001.

[6] AGARWAL, B. L. Post-buckling behavior of composite shear webs. AIAA JOURNAL, Seattle, v. 19, n. 7, p. 933-939, 1981.

[7] BRUHN, E. F. Analysis and design of flight vehicle structures. U.S.A.: Tri-State Offset Co., 1973. 996 p.

[8] NASTRAN, M. S. C. Quick reference guide. MSC. SOFTWARE, v. 1, 2012a.

[9] BRAZ, T. B. C. Comportamento em flambagem e pós-flambagem de painel aeronáutico reforçado com laminado de camada fina versus camada grossa. 2018. 49 f. Monografia (graduação em engenharia mecânica) – Escola de engenharia, Universidade Federal de Minas Gerais, Belo Horizonte, 2018.

[10] NASTRAN, M. S. C. Implicit Nonlinear (SOL 600) User's Guide. MSC. SOFTWARE, v. 1, 2012b.

[11] BAILIE, J. A.; LEY, R. P.; PASRICHA, A. A summary and review of composite laminate design guidelines. Langley RC, Hampton, 1997.

[12] DANIEL, I. M.; ISHAI, O. Engineering mechanics of composite materials. New York: Oxford university press, 1994. 410 p.

[13] KASSAPOGLOU, C. Design and analysis of composite structures: with applications to aerospace structures. Chichester: John Wiley & Sons, 2013. 318 p.