université de BORDEAUX

BUCKLING OF A COMPOSITE STRUCTURE

TFMASA Course

Under Professor

Enrico PANETTIERI

By

GUPTA, Vivek Anantkumar

SANGHVI, Mahek Atul

NAGARAJ, Kushal

NAIK BURYE, Nishidh Shailesh

TABLE OF CONTENTS

1. INTRODUCTION

Buckling refers to the behaviour of a composite structure when it is subjected to compressive loads. Buckling is a type of failure that occurs when the structure becomes unstable and collapses under the applied load.

Post-buckling analysis is the study of the behaviour of a composite structure after it has undergone buckling. This type of analysis is used to determine the residual strength and stiffness of the structure, as well as to identify any potential areas of damage or weakness.

Composite structures are made up of two or more materials with different physical and mechanical properties. They are widely used in a variety of applications due to their high strength-to-weight ratio and ability to withstand high loads. However, composite structures are also prone to buckling under certain conditions, and it is important to understand their behaviour during and after buckling to design them effectively.

There are several factors that can affect the buckling behaviour of a structure, including its geometric configuration, material properties, and the type and distribution of loads applied to it. In order to perform a buckling analysis, engineers must consider these factors and use specialized software or mathematical techniques to model the behaviour of the structure under various loading conditions.

Buckling analysis is an important tool for engineers and designers, as it helps to ensure that structures are strong and stable enough to withstand the loads they will encounter in service. It is also used to optimize the design of structures in order to minimize the risk of failure and ensure the safety of the people who will be using them.

2. OBJECTIVES:

The objectives of the analysis include:

- To perform an analysis for a composite plate with analytical and numerical results.
- To use an appropriate FE model to numerically replicate the experiments performed on a single stringer-stiffened panel and to compare the outcomes.

- Additionally, to pre-process and post process for buckling of a composite structure.
- To provide a pre-process (Buckling) which is to build an analytical approach to assess the buckling load of a straightforward composite plate and to compare the outcomes with an Abaqus numerical model.

Here, composite structure with several stacking sequences is carried out, along with the application of various boundary conditions and loads in order to perform a numerical analysis. We may examine the body using Abaqus software based on the thickness, conditional layers, and various ply angle angles. In this project, the study of an object's buckling method is contrasted to a mathematical analysis of its buckling performed with the aid of a Python script and the displacement that an object can cause based on its eigen value.

Additionally, the goal of the post-buckling is to numerically replicate the experiments performed on a single stringer stiffened panel using an appropriate FE model and to compare the outcomes.

3. LITERATURE REVIEW

The current study performed post buckling analysis on a Single Stringer Composite (SSC) specimen built of carbon fibre-epoxy material using the research work described below as a reference. The linear buckling and post buckling analyses were carried out using the finite element method and ABAQUS 2020 software (FEM).

However, in our study specifically, we will assume that the stringer and the skin are properly connected in order to simplify the model and the analysis. The material used is IM7/8552. Using a single-stringer compression specimen, a method is suggested for evaluating the damage tolerance and collapse of stiffened composite panels. The specimen's dimensions are chosen so that its nonlinear response and collapse are comparable to those of a multi-stringer panel of the same size compressed. Both specimens with and without embedded delamination are tested experimentally. To anticipate the post-buckling reaction as well as the damage evolution from initiation to collapse, a shell-based finite element model with capabilities for intralaminar and interlaminar damage is constructed.

Aeronautical and aerospace uses for composite materials are mainly untapped. Metals have been replaced by composites in medium- to high-volume applications in recent years. For instance, the wing and fuselage are primarily constructed of composite materials. Composites have shown promise when used as structural elements. As a result, weight is reduced while maintaining the structural integrity. Stiffened structures are one example of how engineers have employed composites to construct such structures. The reason for such exceptional mechanical qualities is how such structures behave when they buckle. The construction can be extensively used in post-buckling areas where compressive pressure on a thin plate causes buckling load instability to occur before the point of failure. This phenomenon was thoroughly examined in the BISAGNI, C., & Vescovini, R. (2011) [1], which focused on the damage tolerance and failure of a stiffened single-stringer compression specimen. Using an MTS dynamic testing system with displacement as the control parameter, the specimen was evaluated under axial compression. The specimen was also given a 20 mm defect using a Teflon insert sandwiched between the skin and the co-cured hat-shaped stringer in order to study the impact of early faults on the total residual strength of the global structure.

Given the nonlinearity of the geometric response and its interaction with local damage modes, it is still challenging to estimate the durability and damage tolerance of post-buckled composite structures. A study was done to look into the advancement of quasi-static and fatigue damage in a single-stringer compression (SSC) specimen by experimentation. Three samples were created using a hat-stiffener, and a first flaw was created when a Teflon film was inserted between one of the stringer's flanges and the skin. The other two specimens were tested by cycling in post-buckling, while one specimen was evaluated under quasi-static compressive pressure.

The post-buckling behaviour predicted by computational analysis was compared to the experimental findings in terms of structural and damage response. The critical buckling loads that the specimens experienced were determined to be understated by the results of the computer study. The feature of damage progression under quasi-static and fatigue loading was also examined in the article by BISAGNI, C., D VILA, C. G., ROSE, C. A., and ZALAMEDA, J. N. [2]. The flaw was a 40 mm debonding between the skin and the hat-shaped stringer. Both the initial defect's development and the delamination effect's effects were monitored.

4. MODELLING AND METHODOLOGY

4.1 HYPOTHESIS

- There are no progressive damage models, such as skin-stringer separation or intralaminar damage, in the simulation used in this work.
- It is possible to utilize failure criteria to explain why the specimens are not unbreakable. In this instance, the longitudinal compressive strain criterion is taken into account.
- Modelling the structures based on dimensions from prior studies is done using shell elements.

4.2 ANALYTICAL MODELLING

4.2.1 COMPOSITE PLATE WITHOUT STIFFENER

To solve analytically we solve using the set of equation as shown in equations below. The code has been written and results have been evaluated.

```
###########
# Stacking sequence definition (write the entire stacking sequence)
# [02,902,02]s stacking sequence
SS=[0,0,90,90,0,0,0,0,90,90,0,0]
# [02,902,02,+-45]s stacking sequence
SS=[0,0,90,90,0,0,45,-45,0,0,90,90,0,0,+45,-45]
# [02,902,02,+-45]2s stacking sequence
SS=[0,0,90,90,0,0,+45,-45,0,0,90,90,0,0,+45,-45,-45,+45,0,0,90,90,0,0,-45,+45,0,0,90,90,0,0]

# For example for a composite plate clamped on all 4 edges
N_cr_clamp=np.pi**2/b**2*(4.6*np.sqrt(D_k[0,0]*D_k[1,1])+2.67*D_k[0,1]+5.33*D_k[2,2])
# For example for a composite plate simply supported on all 4 edges:
m = int((a/b)*(((D_k[1,1])/(D_k[0,0]))**0.25))
N_cr_simply=(((np.pi**2)*D_k[1,1])/(b**2))*(((m**2)*(D_k[0,0]/D_k[1,1])*((b/a)**2))+((2*(D_k[0,1]+2*D_k[2,2]))/D_k[1,1])+((1/(m**2))*((a/b)**2)))
```

These lines of code define the changes made to obtain the analytical results for different stacking sequence and boundary conditions for clamped and simply supported conditions. The stacking sequence needed to be analysed analytically are $[0_2,90_2,0_2]_s$ stacking sequence, $[0_2,90_2,0_2,+/-45]_s$ and stacking sequence $[0_2,90_2,0_2,+/-45]_{2s}$ stacking sequence. The boundary conditions analytically solved for clamped and simply supported conditions.

4.3 NUMERICAL MODELLING

4.3.1 COMPOSITE PLATE WITHOUT STIFFENER

Buckling analysis is used to evaluate the buckling value using eigen value extraction for first mode. For the initial buckling analysis, the plate without stiffener is used with two boundary conditions i.e., All clamped edges and simply supported boundary conditions with appropriate loading on one side. The dimensions used for simulation of a composite plate without and with stiffener has been shown in Fig.1.

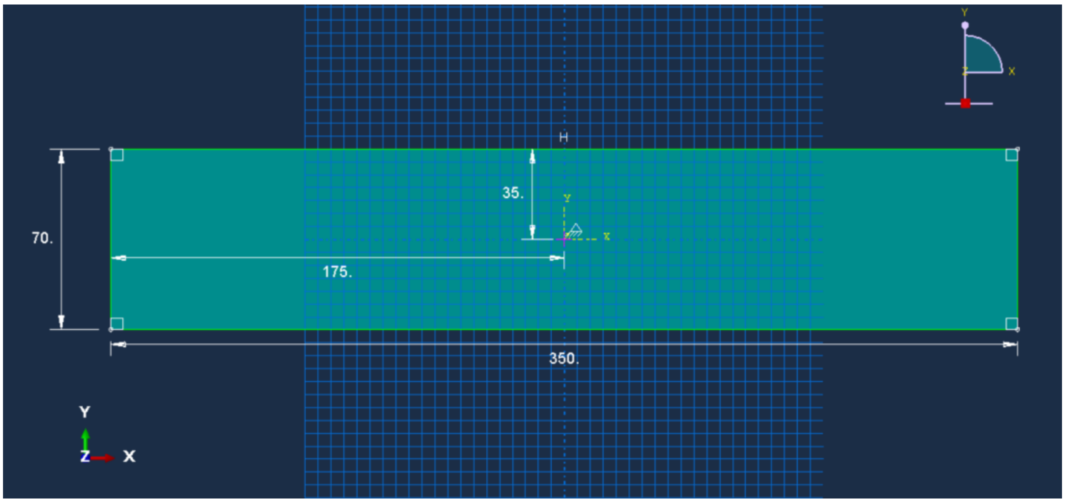


Fig.1 Design of composite plate without stiffener

MATERIAL PROPERTIES: Materials of composite plate for buckling evaluation has been tabulated in Table 1.

Table 1: Material Properties of Composite Plate

Material Constant	**Value**
E_1	180 GPa
E_2	10 GPa
v_{12}	0.3
G_{12}	3.5 GPa
G_{21}	3.5 GPa
G_{22}	5.3 GPa

Composite Stacking is performed as shown in Fig. 2 for $[0_2, 90_2, 0_2]_s$.

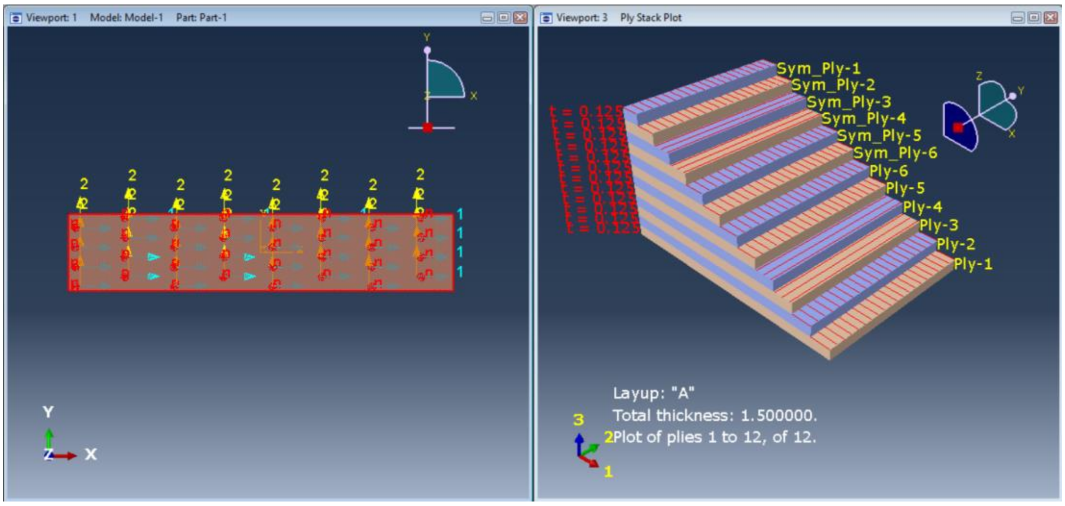


Fig. 2 Composite stack with $[0_2, 90_2, 0_2]_s$

Similarly for stacking sequence $[0_2,90_2,0_2,+/-45]_s$ and $[0_2,90_2,0_2,+/-45]_{2s}$ has been performed.

BOUNDARY CONDITIONS:

Condition 1: Here all the edges are clamped with null slope at the end points and a loaded edge A at constraint with a reference point. The edge opposite edge C to edge A, is constraint. Rest two edges B and D are unloaded edges.

Condition 2: Here all the edges are simply supported with slope at the end points and loading and constraints same as the clamped with similar reference point and edge A. B and D edges are unloaded here as well.

MESH:

The geometry used for composite plate is a simple rectangular 2D sheet thus simple Quad elements of size 35mm is used globally.

4.3.2 COMPOSITE PLATE WITH STIFFENER

GEOMETRY:

The skin and stinger are both modelled using shell elements in the finite element model of the SSC specimen used in this investigation. Both global and local buckling must be represented by elements that span at least two dimensions. The thicknesses of the parts do not require modelling because they are not taken into account or examined in this study and are unimportant in relation to their other dimensions. The SSC specimen is discretized in

ABAQUS utilizing traditional shell elements. While the skin is constructed of 3D planar shells, the stringer is made of 3D extruded shells as shown in Fig. 3.

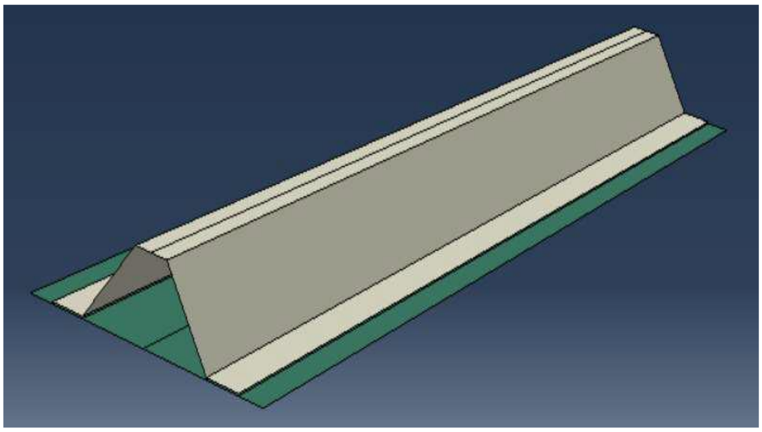


Fig.3 Model for Post Buckling analysis generated from the python script

MATERIAL PROPERTIES:

The skin and stringer are made of IM7/8552 graphite-epoxy for the material property settings, as shown in Table 1. The stacking sequence of 45,90, -45,0,0, -45,90,45 for skin and -45,0,45,0,45,0, -45 for stiffener and must be built for the composite layup settings.

INTERACTION:

Previous accounts state that a co-cured hat-stringer was used to create the specimens. As seen in above Figure, a Tie Constraint, a perfect interface, is used to express this interaction. The skin and stringer are used to confine the master and slave surfaces, respectively. In order to make it simpler to extract the force-displacement curve in the results, two reference points with Master Point Constraints were established, one at each end of the specimen. The MPCs are rigidly attached to the edge nodes in order to distribute the load evenly.

BOUNDARY CONDITIONS AND LOAD STEP:

According to the authors' analysis, the boundary requirements entail restricting one of the specimen's two ends while permitting free extension on the other side in the longitudinal direction—the fibre direction. The Linear Perturbation Buckle step, as opposed to the default Static General phase, is the load step that is taken into consideration while performing a buckling analysis. As stated in the linear buckling analysis, the unit value -1 was utilized for the load, and the buckling loads' amplitudes were given as eigenvalues.

MESHING:

To choose the ideal mesh size, a sensitivity analysis for the linear buckling analysis is carried out. Therefore, a coarse mesh with components of 12 mm is evaluated initially. When the difference in eigenvalues between two subsequent studies with different mesh sizes can be disregarded, the mesh element size is gradually decreased to an optimal mesh of size 4 and a fine mesh of size 2.4. The global seed size should be substantially larger than the element thickness because the model uses shell elements. As a result, the buckling may be represented more precisely. Meshes with element size of 0.01mm, 0.04mm and 0.08mm are compared for the post-buckling analysis results.

PRE & POST-PROCESSING:

In the pre-post buckling analysis, considering a particular stacking sequence we have done some modifications in the code related to change of dimensionality of the structure along with the material properties. Later on, we change the mesh seed parameters to get varying meshing from coarse to fine so that we conclude for the best mesh parameters.

For further analysis we have done changes to boundary conditions such as by modifying the displacement values to "SET" and "UNSET" which represent the following conditions, for "Encastred" one we define the value of reference point related to force as follows u1=SET, u2=SET, u3=UNSET, ur1=SET, ur2=SET, ur3=SET and for the other edge of RP u1=SET, u2=SET, u3=SET, ur1=SET, ur2=SET, ur3=SET. And, for "Simply Supported" condition, the value of reference point related to force are as follows u1=UNSET, u2=SET, u3=SET, ur1=UNSET, ur2=UNSET, ur3=UNSET and for the other edge of RP u1=SET, u2=UNSET, u3=SET, ur1=UNSET, ur2=UNSET, ur3=UNSET.

Same set of instruction of the code is further used for post-buckling analyses with changing the it to dynamic for results in post buckling.

5. RESULTS & DISCUSSION

5.1 ANALYTICAL RESULTS

5.1.1 COMPOSITE PLATE WITHOUT STIFFENER

The code for solving analytically with 3 different orientations and 2 cases which includes simply supported and encastered boundary condition has been run and the following results has been found as mentioned below.

1. **Stacking Sequence $[0_2, 90_2, 0_2]_{s.}$:**

Case: All Edges Clamped

The critical load, N_x, per unit length is 239.65956732826166 N/mm

The total load, N_x, per unit length is 16776.169712978317 N

Case: All Edges Simply Suppported

The critical load, N_x, per unit length is 124.7252057886434 N/mm

The total load, N_x, per unit length is 8730.764405205038 N

2. **Stacking Sequence $[0_2,90_2,0_2, +/-45]_s$:**

Case: All Edges Clamped

The critical load, N_x, per unit length is 591.1887798555658 N/mm

The total load, N_x, per unit length is 41383.214589889605 N

Case: All Edges Simply Suppported

The critical load, N_x, per unit length is 271.34132342460737 N/mm

The total load, N_x, per unit length is 18993.892639722515 N

3. Stacking Sequence $[0_2,90_2,0_2, +/-45]_{2s}$:

Case: All Edges Clamped

The critical load, N_x, per unit length is 4999.520708340615 N/mm

The total load, N_x, per unit length is 349966.44958384306 N

Case: All Edges Simply Suppported

The critical load, N_x, per unit length is 2424.0281689981703 N/mm

The total load, N_x, per unit length is 169681.9718298719 N

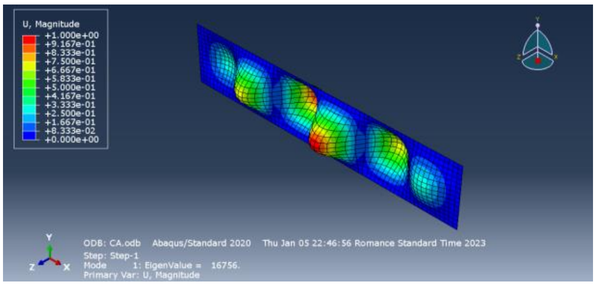
The results show that as number of plies and orientation changes it greatly impacts as the critical load on the structure drastically. The structure with more number plies and orientations has more critical load than the other which help to sustain buckling for a greater load. The encastered support has more support for critical load compared to the simply supported, which tells us that simply supported is more likely to form buckling at a less load application.

5.2 NUMERICAL RESULTS

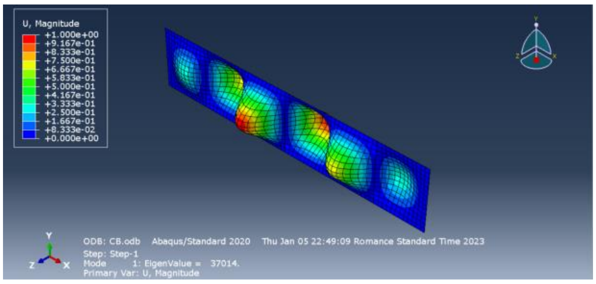
5.2.1 COMPOSITE PLATE WITHOUT STIFFENER

Initially considering non stiffened composite plate the buckling analysis is done to find the buckling load. When the load is applied on one edge with opposite edge is constraint the stacking sequence A gives bulking load of 16,756 N corresponding to clamped plate where as it is 7,500 N in the simply supported boundary condition. Similarly for the clamped with higher ply stack B have buckling load of 37,014 N and 15,915 N for clamped and simply supported plate respectively. The buckling load for both the conditions for a stack with 16 layers of plies increases for drastically with magnitude of 2,65,738 N and 1,32,882 N. This study shows that clamped requires higher buckling load for the composite plate to buckle. And as the number of plies increases its evident that the buckling load also increases. Fig. 4 shows a composite plate with a clamped boundary condition for stacking sequence of stack A as $[0_2, 90_2, 0_2]_s$, stack B as $[0_2,90_2,0_2, +/-45]_s$, and stack C as $[0_2,90_2,0_2, +/-45]_{2s}$. Fig. 5 shows a composite plate with a simply supported boundary condition for same stacking sequence.

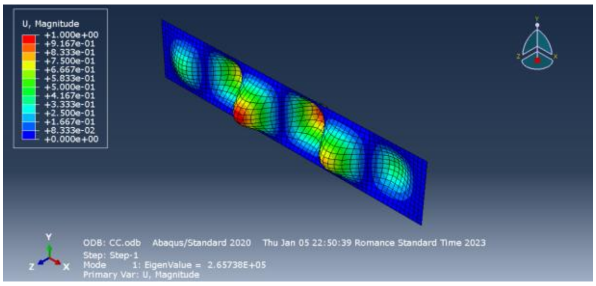
CLAMPED BOUNDARY CONDITION



Stack A



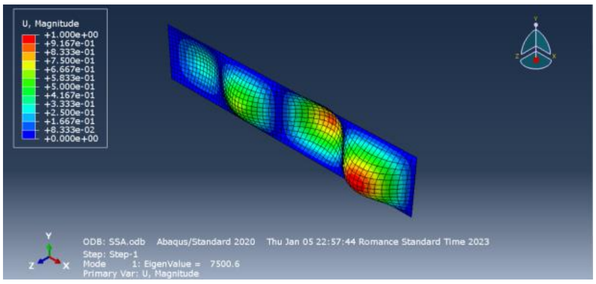
Stack B



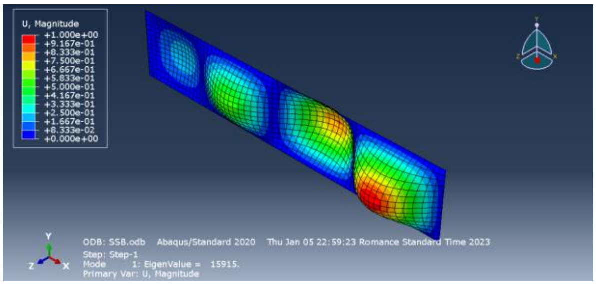
Stack C

Fig. 4: Deformation contour for mode 1 in clamped boundary condition for different stacks for a composite plated without a stiffener

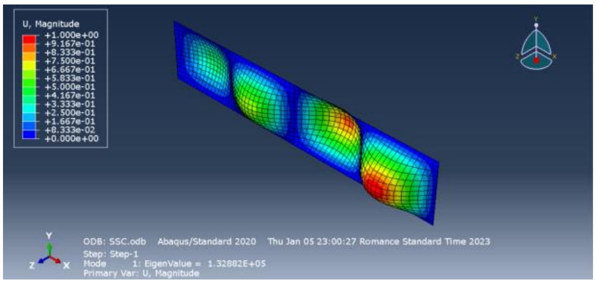
SIMPLY SUPPORTED BOUNDARY CONDITION



Stack A



Stack B



Stack C

Fig. 5: Deformation contour for mode 1 in simply supported boundary condition for different stacks for a composite plated without a stiffener

Comparison of Analytical and Numerical results for composite plate without stiffener for critical loads for 3 stacking sequence as mentioned before which is shown in Fig. 6.

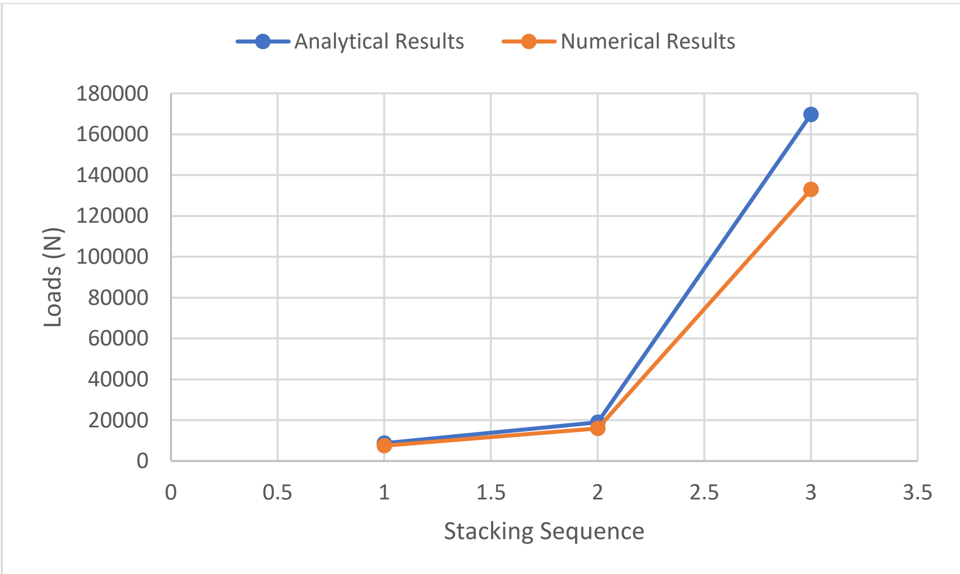


Fig. 6: Graph depicting critical loads solved analytically and numerically

5.2.2 PRE-BUCKLING OF COMPOSITE PLATE WITH STIFFENER

Numerical analysis is performed using a pre-defined python scrip with CAD mode designed with a stiffener and all boundary conditions and constrained stiffener-plate ends. The model is also meshed with mesh size defined in the script edge wise. To find an exact mesh size we go through various mesh size as mentioned before and finalize with coarse mesh size and run the entire analysis with the same mesh for the stiffened panel. And the stacking sequence has also been kept constant throughout.

Due to compressive load on the panel with stiffener the analysis is done using pre-defined python script. The geometry also has a length of 280mm, a stringer height of 30mm, a crown top width of 15mm, and a stiffener flange width of 15mm.The model's geometry was created by the Python script. The adjustments being made are to the composite panel's material description, which is IM7/8552 graphite-epoxy material.

When boundary condition of simply supported used the value of buckling load with the stiffener is 9214.9 N whereas for simply supported case is it 8594 N, which is almost same in terms of boundary condition variation the clamped required more load value for the structure to buckle as shown in Fig. 7 and Fig. 8 respectively. As we see the magnitude is almost similar but when stiffener is added the tendency for the composite panel to buckle decreases.

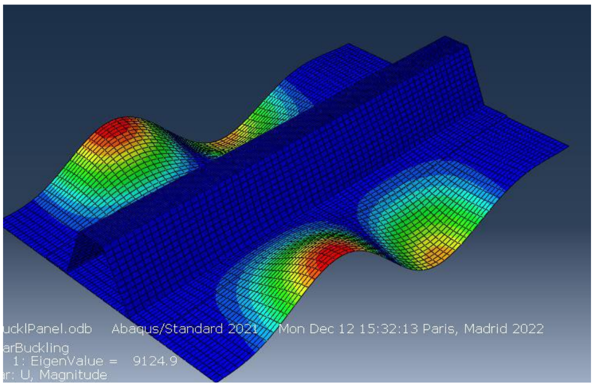


Fig. 7: Case 1 - Clamped Edges

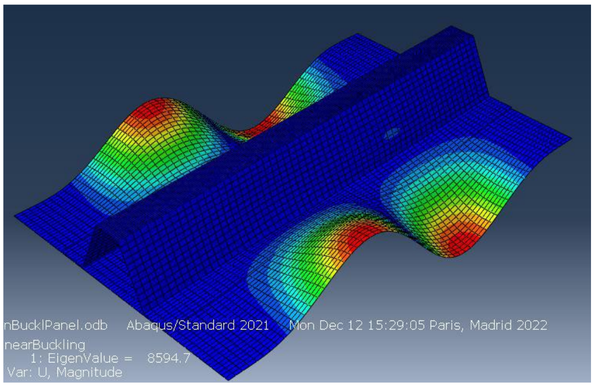


Fig. 8 Case 2 - Simply Supported Edges

5.2.3 POST BUCKLING OF STIFFENER

The model produced using the necessary boundary conditions and saved as a different input file. The number of stages that must come before processing is then added to the .inp file by editing. The assumed steps in this particular case are 10 and furthermore, the analysis process is made simpler by running the code through the Abaqus tool. It will create .fil file as the starting point for the post buckling analysis. We can get the scale factor by running the technique iteratively. The minute scale factor of 10% is used initially going further with an intermediate scale factor of 30% and then 80% scaling regarded as major scale factor. We can run the file numerous times by performing post buckling analysis on these parameters using the Abaqus command. As can be ween A stress value of 1001 MPa is generated shown in Fig. 9. As shown in the graph below the scenario for the initial post-buckling at 10% scale factor is run, we can observe a gradual growth. The graph likewise has a slight kink at about -0.65 displacement, but it gradually rises with each analysis run. Run the scale factor at a 30% setting at medium evaluation of the post-buckling mode. With a time, scale factor of 105, it reveals that the stress value at the stiffener is around 1002 MPa as shown in Fig. 10. Bu in this case the kink is significantly large ass compared to the 10% scale factor. For the major scale factor, the stress vale stained is 1002 MPa as shown in Fig. 11 with a graph of force-displacement with no kink variation. The variation is displacement of increases gradually as force acting on it increases with each scale factor showing almost same value of mises stress.

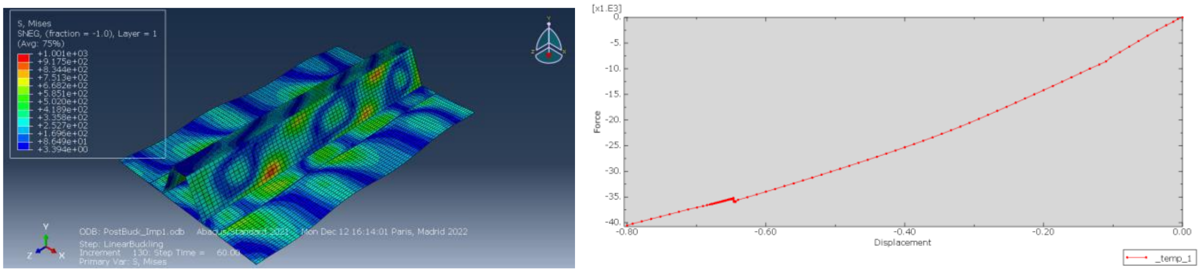


Fig. 9: Post buckling at scale factor of 10%

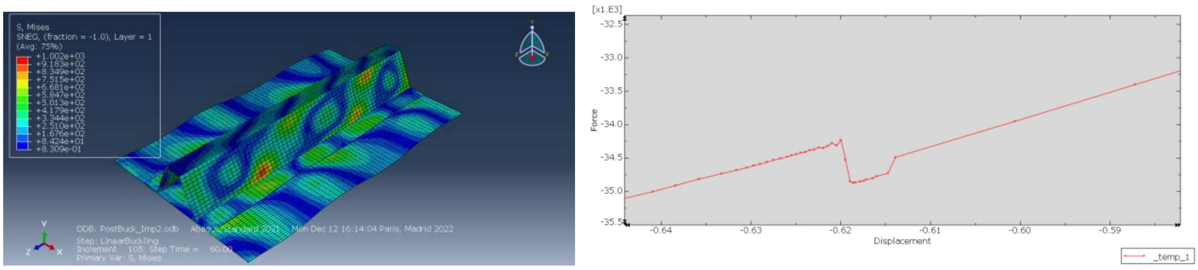


Fig. 10: Post buckling at scale factor of 10%

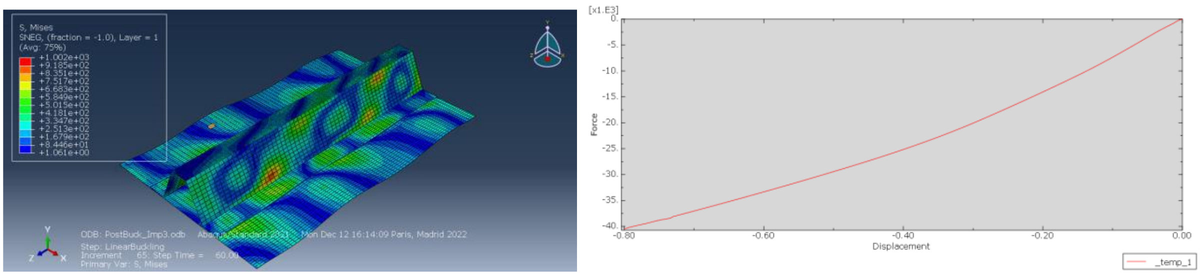


Fig. 11: Post buckling at scale factor of 10%

6. CONCLUSION

Here, we construct an analytical method in the pre-process (Buckling) to evaluate the buckling load of a simple composite plate and to contrast the results with an Abaqus numerical model. In order to do a numerical analysis, multiple stacking sequences for composite structures are used here, coupled with a variety of boundary conditions and loads. Using Abaqus software, we looked at the body's thickness, conditional layers, and different ply angle angles. The examination of an object's buckling technique in this project is compared to a mathematical analysis of its buckling carried out with the aid of a Python script and the displacement that an object can produce depending on its eigen value. Additionally, using a suitable FE model in the post-buckling, we numerically repeated the experiments conducted on a single stringer stiffened panel in order to compare the results. The involvement of stiffener does have magnitude reduction of buckling load required for the structure to deform. Also, material like composite with increment of stacking sequency i.e., number of plies also require overall larger buckling load to have deformation due to buckling failure.

The interval of kink between the force displacement curves the response do have a negative stiffness during which the stiffener with composite panel releases strain energy to maintain the equilibrium. To study experimentally and statistically the post buckling response of stiffened composite panels and to verify a progressive damage analysis model, a single-stringer specimen is presented. By spotting patterns in anticipated failure indices in a multistring panel, the specimen's size was determined. Trial and error were used to determine the size of a single-stringer specimen with a similar failure index pattern and post buckling reaction.

7. REFERENCES

[1] C. Bisagni, R. Vescovini, and C. G. Dávila, “Single-stringer compression specimen for the assessment of damage tolerance of postbuckled structures,” *J. Aircr.*, vol. 48, no. 2, pp. 495–502, 2011, doi: 10.2514/1.C031106.

[2] C. Bisagni, C. G. Dávila, C. A. Rose, and J. N. Zalameda, “Experimental evaluation of fatigue damage progression in postbuckled single stringer composite specimens,” *Proc. Am. Soc. Compos. - 29th Tech. Conf. ASC 2014; 16th US-Japan Conf. Compos. Mater. ASTM-D30 Meet.*, 2014.