Mesh Sensitivity Study of Composite Cylindrical Shells

Muhammad Saqib

Table of Contents

Mesh Sensitivity Study of Composite Cylindrical Shells

To study the effect of element size on the estimated buckling load of composite cylindrical shells

Abstract

This report is about the impact of mesh size on the buckling behavior of composite shells using both linear and non-linear finite element analysis (FEA). The primary objective of this report is to determine the optimal mesh size. It is used to calculate for accurate buckling load predictions while evaluating the effectiveness of linear and non-linear models on two different models with different orientations. The methodology involves performing FEA simulations with various mesh sizes, ranging from 2.5 mm to 50 mm. It was used to observe the effect on buckling loads for two composite shell models with orientation 45 and -45, 0,45,0, and -45 for the 2nd model. Linear analysis is compared with non-linear analysis using Riks' method and eigenvalues, and errors are calculated by comparing the results with analytical solutions. The results show that smaller mesh with smaller sizes provide more accurate buckling load predictions, with the 2.5 mm mesh yielding errors below 1.4% for linear analysis and 0.16% for non-linear analysis in Model 1. As the mesh size increases, the error grows, reaching up to 33.9% for Model 2 with a 50 mm mesh size. In conclusion, non-linear analysis yields more accurate results compared to linear analysis, and mesh refinement plays a vital role in achieving reliable results. This study highlights the balance between computational efficiency and result precision in FEA simulations.

1 Introduction

To explore linear and nonlinear buckling in cylinders, the potential sensitivity of the buckling behavior of cylindrical shells is carried out. Carbon fiber-reinforced composite cylindrical shells have been used in modern structural engineering. Their application due to their structure is important in many fields like aerospace and others. The finite element analysis is used for analysis. For the cylindrical shell, FEA is used to analyze the linear or eigenvalue and non-linear Riks analysis. On the geometry, its imperfection effect and loading are under consideration.[1]. The use of these shells in different fields is due to their considerable strength-to-weight ratio. For anything to be used in the industry or somewhere, its strength, and buckling effect are important to know[2]. Carbon fiber-reinforced plastic composite materials have been used for buckling analysis of cylindrical shells. Numerical results are used to validate the experimental or analytical one for composite shells. So these numerical results or simulations can be more effective and time-saving for the shell than the expensive experimental methods[3]. For the verification of this simulation done in numerical analysis, some program or solution is required for the validation of the numerical model analysis. By using the proper tool for the analysis, it is then possible to perform the sensitivity mesh analysis on the cylindrical shell[4, 5]. Meshing on the structure is important for result accuracy. The asymmetric meshing technique was the first step in the numerical analysis of cylindrical shells. Irregular meshing is also used for the analysis of shells[6]. The concept of the Asymmetric Meshing Technique (AMT), introduced by Wardle, enables efficient detection of bifurcation points and post-buckling behaviour in a single FEM run, as opposed to traditional techniques that require multiple iterations.[7]. Buckling analysis of composite shells is essential for determining the structural stability and strength of the cylinder under critical loads. Finite Element Analysis (FEA) in Abaqus was used to enable both linear and then nonlinear buckling studies were done to compare the results. The linear analysis predicts the critical buckling load using eigenvalue analysis and then the nonlinear analysis captures post-buckling behaviour, accounting for large deformations and material nonlinearities. Composite shells exhibit complex buckling modes due to anisotropy and heterogeneity, making FEA indispensable for accurate prediction. Abaqus allows for detailed mesh refinement and the use of shell elements to achieve convergence. The results guide design improvements, ensuring optimal strength-to-weight ratios in aerospace, automotive, and marine applications. Various studies, including those by Kim, Baba, and Han, have also employed irregular meshing for composite plates and shell structures[8]. AMT is commonly implemented through "patch" and "band" meshing methods, where mesh asymmetry is introduced in localized regions, impacting post-buckling behaviour while having minimal effect on linear buckling [9]. Two types of finite element analysis were utilized: which is one as linear analysis to study the symmetric and asymmetric meshing and another one nonlinear analysis. In the linear analysis eigenvalue was done and in the nonlinear Riks was used to analyze. In both methods, the study is about the effect of symmetric and asymmetric meshing. It was about the buckling load and its behaviour on the cylindrical shell. Meshing was a critical step in finite element analysis. Results accuracy was dependent on the mesh sensitivity[10]. A very small element size or refined mess has more accurate results as compared to a large or coarse size mess element. In this lab report, the main purpose is mess sensitivity analysis. The different element-size mesh was used to analyze the cylindrical shell. When mesh size was decreased from large to small, it was understandable that mesh size improved the result abut the computation time also goes on increasing[11]. So, the sensitivity and accuracy of the result also improve by the mesh's accurate size and type. So, in industries like aerospace and others, high accuracy was required for the equipment analysis. For this, computers are used which are best for the finite element and also for better results. Material properties and boundary conditions were also considered to be very important for the results. High-strength materials undergo less buckling as compared to less strength[12]. The objective of the report was to study the mesh element size. Then mesh element size was analyzed on two different analyses linear and linear model[13]. Then after getting the results analytical results are the time compared with eh numerical ones. Then linear results of model one and model two are compared to each other. So the main purpose of this report was the element size effect on the mesh sensitivity on the cylindrical shell which was made of composite material. Blucking amalysis for the two model having different orientation was done in the Abaqus in this report. The results are then compared to each other and then understanding is developed about the buckling load. These analysis are best for the basic understanding of the buckling load in shells.

2 Methodology

The buckling analysis for the Mesh sensitivity of a cylindrical shell was done numerically to approach the accurate results. Numerical analysis was then compared to the analysis. For this, linear and non-linear analysis of the shell is done in Abaqus which was then compared to the numerical one. Abaqus was widely used for the analysis because of its ability to handle linear and nonlinear buckling scenarios. First of all, in

Abaqus, a work directory was added. Then model was saved in the required name buckled. The model was created. For this, part was created and then circumference and length were added. Circumference was about 520mm and the length of the cylinder was 700mm. The model was saved. Then material was assigned to the model. The material was carbon fiber-reinforced plastic. Modulus of elasticity and passion ratio were assigned. The section was created and then a section was assigned to the model. The model was assembled and orientation was applied. which was defined as 45,-45 and 0,45,0,-45. These were assigned to the model. The mesh was done on the model. In meshing element size was given as per the requirement. Linear buckling analysis is performed to determine the critical buckling load. Abaqus calculates eigenvalues representing the load factors that cause buckling. The first mode shape is extracted to understand the initial buckling behavior. This analysis helps identify the critical load at which buckling initiates. Similar way Riks's method was used for the nonlinear analysis. Meshing was done with the S4R element. First of all, the element size 40 was given and then part was meshed. After meshing mesh control was done and then mesh element type was done. Then step was created initial to static. Then after the step job was done. For this job was created and then it was submitted. Then the model was saved and then the result was visualized. Then the next step was to change the element size and analysis was done. The job was created. After the job, data was checked and if any error or step was missing, then that step was edited and then the job was submitted. Then a similar process was done for all the element sizes, and the mesh was again created and submitted. Then after the linear analysis of this, the model was copied, and then nonlinear was started. For this nonlinear analysis, the step changed and meshing was done. For this, all steps were similar to each other and the element size was given and the step was created and then the job was submitted. After submitting the job, results were visualized for the linear and nonlinear again. Then buckling load was checked for the both model's linear and nonlinear analysis. The result was copied and then compared.

3 Results

A numerical analysis of a composite shell made of carbon fiber plastic is done in the Abaqus. The main objective of this report is to do a mesh sensitivity analysis of a composite shell. For this, different element size was used to carry out the mess analysis. The results were obtained for the different models for the linear and nonlinear analysis and then the result was compared with the analytical one. The were two orientations of the model used for the mesh sensitivity analysis. The main purpose of the orientation was to check out the buckling analysis and loading factor in the circumference of the shell body. Then the result for these linear and nonlinear was done for the validation and accuracy of the analysis. Both analysis, linear and nonlinear was done in Abaqus software. The reason was its ability to solve the model in less computation time and better solver to solve and compute the results. Moreover, this analysis is about to contributes to the existing considering both linear and non-linear models with different mesh sizes. The inclusion of non-linear analysis allows for a more comprehensive understanding of the buckling behaviour under different loading conditions but this takes more time than the linear analysis. The reason is that the solver behind the nonlinear is using a different equation like Riks and solve the model.

3.1 Linear Analysis

The results of the linear analysis for the composite shell show that the buckling load for both models decreases as the mesh size decreases means more fine mesh. But it is to be understood that when the mesh size is decreased from 50mm to 2.5mm, then the computation time also increases and at 2.5 mm the solver takes about 20 to 25 minutes to solve the model and provide the result. This is for the linear model time but the nonlinear model takes more time than the linear. The accuracy of buckling load predictions in composite shells is highly dependent on the mesh size used in finite element analysis (FEA). The most accurate results are achieved with a mesh size of 2.5 mm, where the buckling load for Model 1 is 120.236 kN and for Model 2 is 239.080 kN. At this mesh size, the deviation from analytical results is less than 1.40%, indicating a high level of accuracy.

As the mesh size increases, the buckling load also increases, but at a slower rate. For a larger 50 mm mesh size, the buckling load for Model 1 rises to 133.00 kN and for Model 2 to 358.851 kN. This increase in mesh size introduces higher errors, with deviations of 12.2% for Model 1 and 49.5% for Model 2. These results demonstrate that finer mesh sizes, such as 2.5 mm, are essential for accurate buckling load predictions, while larger mesh sizes may compromise precision. Thus, for critical structural analysis, a finer mesh is preferred to ensure reliable and consistent results.

Table 1 Showing Buckling Load for Model 1 and Model 2 at different Element Size

	Buckling Load (kN)	
Element Size (mm)	**Model -1**	**Model- 2**
50	133	358.851
40	131.368	306.694
30	126.116	288.828
20	122.318	274.945
10	120.551	248.417
5	120.297	240.911
2.5	120.236	239.08

3.1.1 Model-1

Linear analysis of the composite shell was done in Abaqus. The buckling effect on the cylinder with the deformed shape. The buckling load calculated by the numerical method for the mesh size of 50 mm is 143.22 kN and it is very close to the analytical one. The deform figure due to buckling load shows that the red area is under more deflection as compared to the start and end of the cylinder. The center region is more under deflection. In a similar way to the above, when the mesh size was decreased and then analysis was done on the composite skell, the results were surprisingly different from the 50mm. The mesh size is now 40mm and the results shown are also that the buckling load found in this case is 131.37kN and these are quite different from the 50mm mess size. So, it is clear that when the mesh size is decreased or reduced the computation error is reduced and results become more accurate. The mesh size was again reduced to 30 mm and then analysis was done in the Abaqus. This was seen from the analysis that as the mess size becomes small the results become more accurate and close to the analytical line. So the sensitivity of the mess is very important for the buckling analysis. The buckling load calculated is 126.118kN. Similar way, the analysis is done for the 20 mm mesh size and then for the 10,5 and 2.5 mm mesh size for the linear model of 45,-45 orientation. So, analysis is best for the mesh sensitivity analysis. The linear analysis for the first orientation and blocking analysis of the composite cylinder. It was clear that when the mess size is kept small then the computation time required for the solver also increased and results became more accurate. The below analysis is done as the heme size of 10mm and the later on done in the 5mm. The computation time required for the 5 mm is about 1 minute to 1.5 and 10 mm is 45 seconds and the results are different in both ways. The result for the 10 mm mesh size is 120.55kN and the 5mm is 120.229kN and these are very close as the element size is very small.

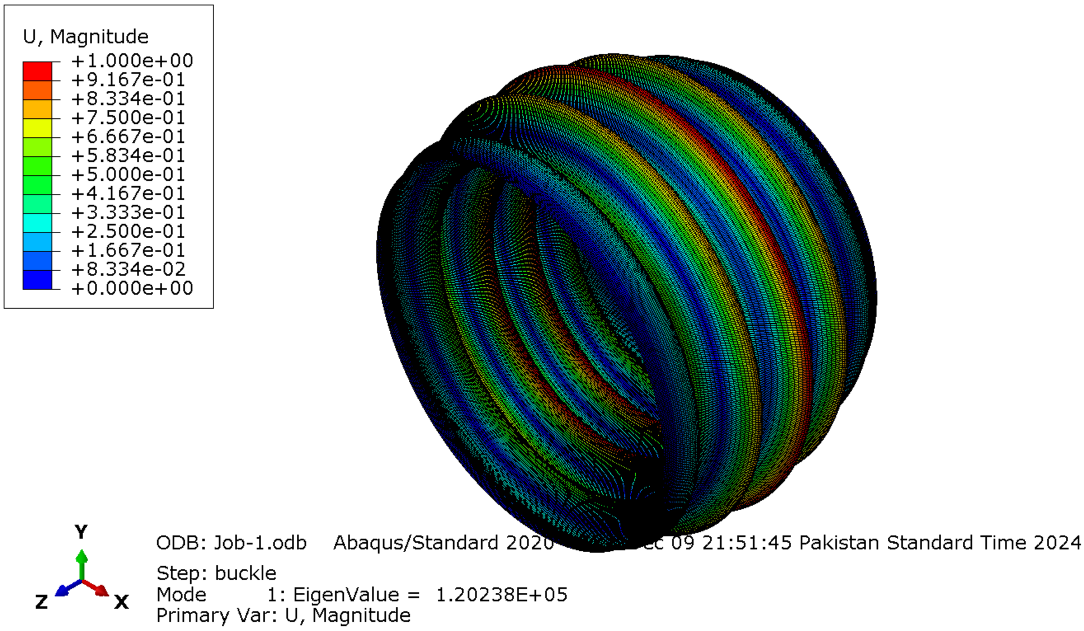


Figure 1 Linear analysis of model 1 at 2.5mm mesh size

3.1.2 Model-2

This analysis is done for model 2 having a different orientation and all the analysis is done for different mesh element sizes. For the first figure, it is about at 50mm mesh size. Then 40 mm, 30 mm, 20 mm, 10 mm, 5 mm, and 2.5 mm. The goal of this study is to know about the mesh sensitivity analysis of the composite skell and determine the buckling load for it.

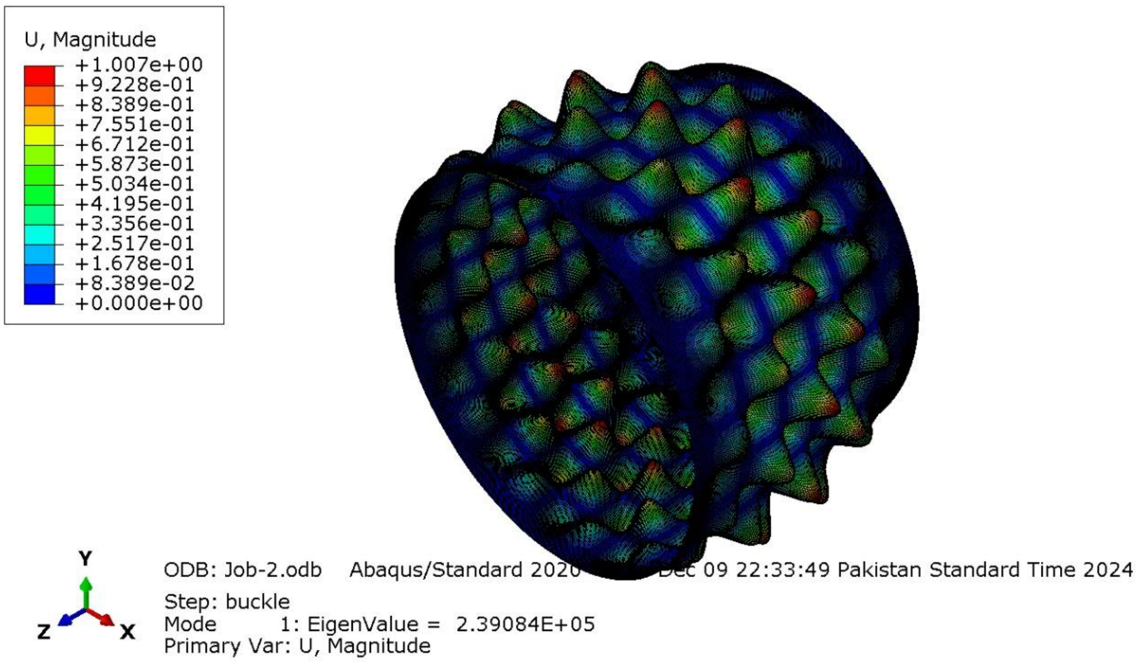
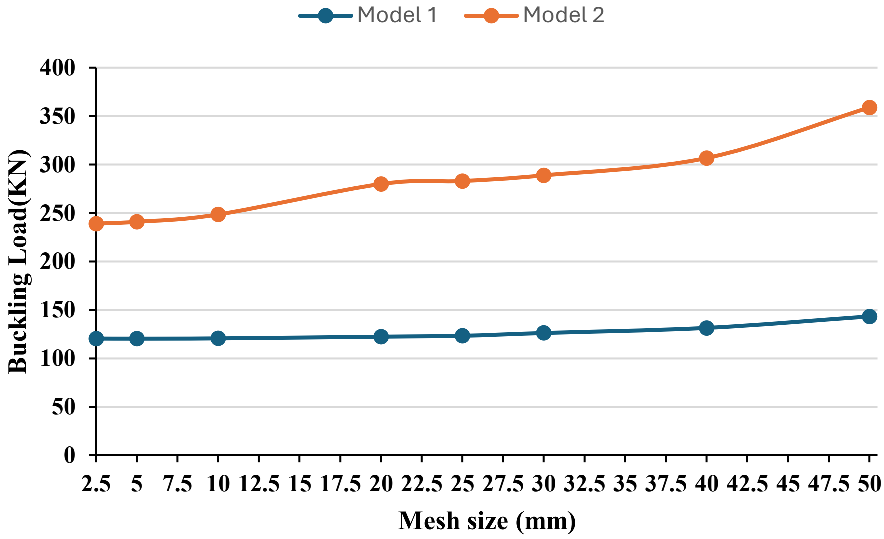


Figure 2 Linear analysis of model 2 at 2.5mm mesh size



Graph 1 linear analysis for model 1 and model 2

In all the figures of the buckling analysis, it is clear that as the mesh size becomes small the accuracy of the results becomes more accurate, and the computation time increases. For the 2.5 mm mesh size, the time for the linear first orientation is about 20 to 25 mint. While the second orientation is also about the same one. So first all these analyses were done in just 10 minutes, but the 2.5 mm takes a lot of time to converge.

3.2 Non-Linear Analysis

The results of the non-linear analysis show a similar trend to the linear analysis results. The buckling load for both models decreases as the mesh size decreases, and the optimal mesh size is found to be 10 mm. At this mesh size, the buckling load for model 1 is 119.269 kN and for model 2 is 274.945 kN. The percentage error compared to the analytical results is less than 0.58% for model 1 and 14.6% for model 2. As the mesh size increases, the buckling load also increases, but the rate of increase decreases. At a mesh size of 50 mm, the buckling load for model 1 is 140.753 kN and for model 2 is 321.466 kN. The percentage error compared to the analytical results is 18.7% for model 1 and 33.9% for model 2.

Table 2 Buckling Load Table for Model 1 and Model 2 at different Element Size

Element size(mm)	Buckling Load (kN)	
	Model 1	Model 2
50	140.753	321.466
40	131.368	306.694
30	126.116	288.828
20	122.318	274.945
10	120.551	248.417
2.5	120.236	239.80

3.2.1 Model-1

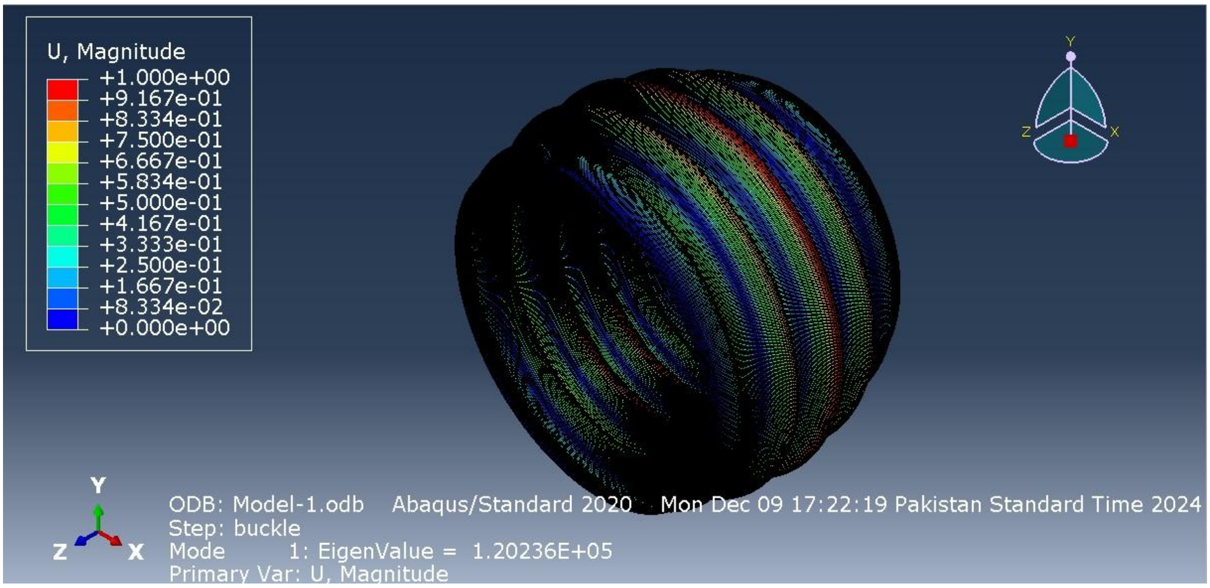


Figure 3 model 1 for nonlinear analysis

3.2.2 Model-2

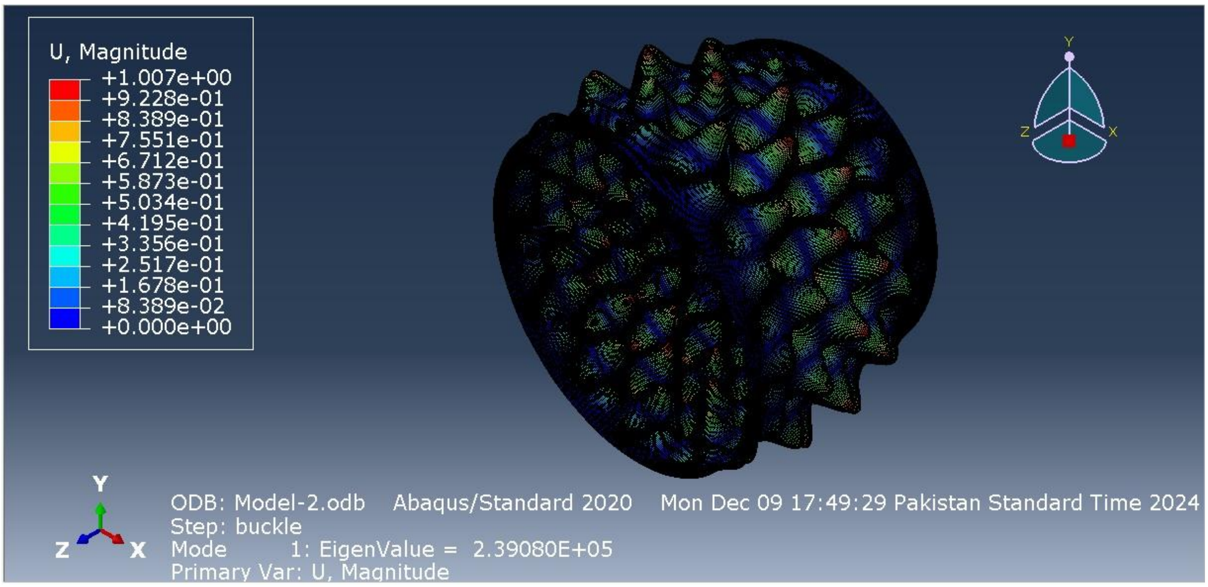
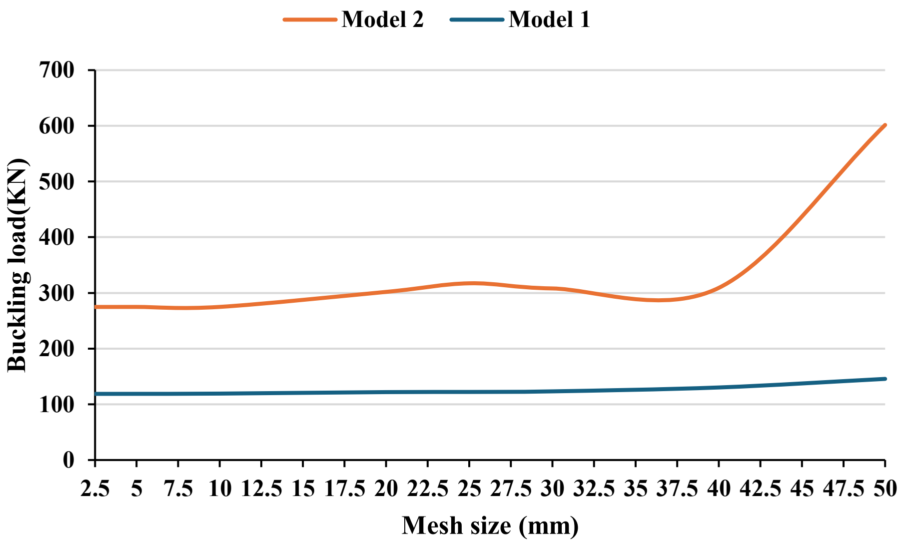


Figure 4 Model 2 for nonlinear analysis



Graph 2 linear analysis for model 1 and model 2

4 Discussion

In the buckling analysis of cylindrical shells, two different models of different orientations were under analysis. Both analyses are used for the understanding and numerical interpretation of the buckling load in the cylinder. In the buckling load, the main factor that is varying in this study is the mesh size. The mesh size controls the whole process of this analysis. The best numerical results that converge to the analytical one are at the mesh size of 2.5mm. However, it is so small that the computation time for the numerical results is too large to converge. Numerical results require better meshing means refined meshing for accurate results. In this study, 2.5,5,10 to 50 mm mesh size was used to develop the understanding and knowledge about meshing and how it affects the results. The main aim of this report is mesh sensitivity analysis. The below table summarizes the results for the model on and the model for linear analysis. Model 1 and model 2 for linear analysis but for the different orientations are compared with the numerical one and percentage errors are also calculated for the study.

Table 3 Buckling Load Table for Model 1 and Model 2 at different Element Sizes for comparison

Linear Analysis Results (kN)		**Analytical Results (kN)**		**Error (%)**	
Model 1	**Model 2**	**Model 1**	**Model 2**	**Model 1**	**Model 2**
120.236	239.08	118.58	240	1.4	0.38
120.297	240.911	118.58	240	1.45	0.38
120.551	248.417	118.58	240	1.66	3.51
120.551	279.925	118.58	240	3.15	16.6
120.551	282.976	118.58	240	3.93	17.9
126.116	288.828	118.58	240	6.36	20.3
131.368	306.694	118.58	240	10.8	27.8
133	358.851	118.58	240	12.2	49.5

The table is for the comparison of Numerical Results and analytical for different element sizes and percentage errors. In this table, it is clear from the results that the load for the buckling is high when the mesh size is very small. In nonlinear and linear analysis, the results are also shown for the nonlinear below table. Both tables summarize the best analysis for the buckling load for shell cylinders. In the case of non-linear analysis, the results are very close to the analytical one and more accurate and best for analysis. However, the computation time for the non-linear is about half more than the linear. A 2.5mm mesh size for linear and nonlinear was carried out and the results calculated are shown in the bale it is clear that the results obtained by nonlinear are better and close to the actual one.

The non-linear analysis of buckling behavior for composite shells made of carbon fiber reveals the significant influence of mesh size. It also affects the accuracy and computational time which is directly related to the results. As observed, the optimal mesh size for Model 1 is 2.5 mm, yielding a buckling load of 119.269 kN with a minimal error of 0.58% compared to the analytical result. For Model 2, the same mesh size produces a 14.6% error, indicating higher sensitivity in Model 2. The results show that as the increase in the size of the mesh for an element, the error also increases, with the maximum error reaching 33.9% at a 50 mm mesh size for Model 2 with a different orientation to Model 1. While smaller mesh sizes provide higher accuracy, they significantly increase computational time. Non-linear analysis demonstrated higher accuracy compared to linear analysis, especially for finer meshes and as element size becomes smaller the results become more accurate and precise. This study emphasizes the importance of mesh refinement to achieve accurate predictions of buckling loads and highlights the trade-off between computational efficiency and result precision in finite element analysis. So when highly accurate and precise results are required for the buckling analysis, high quality, and better mesh size are used.

Table 4 for nonlinear analysis of numerical with analytical and % error

Non-Linear Analysis Results (kN)		**Analytical Results**		**Percentage Error (%)**	
Model 1	**Model 2**	**Model 1**	**Model 2**	**Model 1**	**Model 2**
118.78	274.81	118.58	240	0.16	12.6
118.842	274.858	118.58	240	0.58	14.6
119.269	274.945	118.58	240	2.83	25.8
121.94	301.875	118.58	240	3.09	32.2
122.242	317.259	118.58	240	3.95	28.4
123.268	308.044	118.58	240	9.84	28.8
130.251	309.081	118.58	240	18.7	33.9

The above two tables compared the numerical results for the linear and non-linear analysis for composite shells. In this analysis, the mesh was refined to get better results. The error becomes small and near zero as the mesh size is decreased. However, the computation time increases as the mess size decreases. But for better results, the best accuracy is found at 2.5mm. The mesh size for non-linear analysis is 2.5 mm, with a satisfied. The results of this study are consistent with the findings of other studies on the buckling of thin cylindrical composite shells. These results align with previous studies on the buckling behavior of thin cylindrical composite shells, reinforcing their validity and consistency.

Table 5 for analytical, numerical, and experimental buckling load comparison

Solution	**Buckling Load**	
	Model-1	**Model-2**
Analytical	118.58	240
Linear Analysis	120.236	239.08
Non-Linear Analysis	118.78	274.81
Experimental	112.94	161.22

The above table is a comparison of numerical with the analytical one. Linear and non-linear analysis was done and results were compared for both of the models with experimental also. It is clear that numerical results are very close to the analytical ones and with a very small error. These results can be more refined by meshing making element size more small. But the required time will be too much and the system may not complete the task.

5 Conclusions

The analysis is about the critical role of mesh size in predicting the buckling behavior of composite shells. In this analysis, using linear and non-linear finite element analysis (FEA) for the shell for different mesh sizes was done. Both models were analyzed for the different smaller mesh sizes significantly and the results were improved but at the cost of higher computational effort. The optimal mesh size was identified as 2.5 mm, where the buckling loads for Model 1 and Model 2 using linear analysis were 120.236 kN and 239.08 kN, respectively. For non-linear analysis, Model 1 and Model 2 yielded buckling loads of 118.78 kN and 274.81 kN, with minimal errors of 0.16% and 12.6%, respectively, compared to analytical results. It is also clear that nonlinear is better than linear for load analysis. As the mesh size increased to 50 mm, the error rose to 18.7% for Model 1 and 33.9% for Model 2. The study confirms that non-linear analysis provides more accurate results than linear analysis, especially for finer meshes or smaller element sizes. This conclusion underscores the importance of mesh refinement to achieve accurate and reliable results in the FEA of composite shells. Future modeling and analysis of complex shell structures.

References

1. Bisagni, C., *Numerical analysis and experimental correlation of composite shell buckling and post-buckling.* Composites Part B: Engineering, 2000. **31**(8): p. 655-667.
2. Koiter, W.T., *On the stability of elastic equilibrium*. 1967: National Aeronautics and Space Administration.
3. Abramovich, H., R. Yaffe, and J. Singer, *Evaluation of stiffened shell characteristics from imperfection measurements.* The Journal of Strain Analysis for Engineering Design, 1987. **22**(1): p. 17-23.
4. Huyan, X., G.J. Simitses, and A. Tabiei, *Nonlinear analysis of imperfect metallic and laminated cylinders under bending loads.* AIAA journal, 1996. **34**(11): p. 2406-2413.
5. Bisagni, C., *Instabilità e comportamento post-critico di gusci in materiale composito*. 1997, PhD thesis (in Italian). Dipartimento di Ingegneria Aerospaziale ….
6. Kim, H. and K.T. Kedward, *A method for modeling the local and global buckling of delaminated composite plates.* Composite Structures, 1999. **44**(1): p. 43-53.
7. Wardle, B.L., *Solution to the incorrect benchmark shell-buckling problem.* AIAA journal, 2008. **46**(2): p. 381-387.
8. Han, H., et al., *Numerical and experimental investigations of the response of aluminum cylinders with a cutout subject to axial compression.* Thin-Walled Structures, 2006. **44**(2): p. 254-270.
9. Vaziri, A., *On the buckling of cracked composite cylindrical shells under axial compression.* Composite structures, 2007. **80**(1): p. 152-158.
10. Rahman, T., E. Jansen, and Z. Gürdal, *Dynamic buckling analysis of composite cylindrical shells using a finite element based perturbation method.* Nonlinear Dynamics, 2011. **66**: p. 389-401.
11. Huang, L., et al., *An efficient finite element model for buckling analysis of grid stiffened laminated composite plates.* Composite Structures, 2015. **122**: p. 41-50.
12. Wang, B., et al., *Numerical-based smeared stiffener method for global buckling analysis of grid-stiffened composite cylindrical shells.* Composite Structures, 2016. **152**: p. 807-815.
13. Imran, M., et al., *Design optimization and non-linear buckling analysis of spherical composite submersible pressure hull.* Materials, 2020. **13**(11): p. 2439.