

NUMERICAL ANAYSIS OF THIN PLATE BY USING ABAQUS SOFTWARE

Dr. V. Sudhir, Associate professor Department of Civil Engineering, Vignan's Institute of Information Technology, Visakhapatnam. sudhir.civilviit@gmail.com
Ms. Gnapika Pediredla, UG students Vignan's institute of information technology(A), Duvvada, Visakhapatnam, Andhra Pradesh
Mr. Mohan Pilla, UG students Vignan's institute of information technology(A), Duvvada, Visakhapatnam, Andhra Pradesh
Mr. G. Appala Naidu, UG students Vignan's institute of information technology(A), Duvvada, Visakhapatnam, Andhra Pradesh
Mr. P. Govind, UG students Vignan's institute of information technology(A), Duvvada, Visakhapatnam, Andhra Pradesh
Mr. P. Govind, UG students Vignan's institute of information technology(A), Duvvada, Visakhapatnam, Andhra Pradesh
Mr. U. Bharath Kumar UG students Vignan's institute of information technology(A), Duvvada, Visakhapatnam, Andhra Pradesh

ABSTRACT:

Thin plates are ubiquitous structural elements found in numerous engineering applications, ranging from aerospace to civil and mechanical engineering. Understanding the behaviour of thin plates under various loading conditions is essential for designing efficient and reliable structures. In this study, we present a detailed numerical analysis of thin plate behaviour using Abaqus software, a powerful finite element analysis tool widely employed in engineering simulations. The objective is to investigate the stress distribution, deformation characteristics, and failure modes of thin plates subjected to diverse loading conditions.

The methodology involves employing the finite element method (FEM) to model thin plates within the Abaqus environment. Various plate geometries, including rectangular and circular shapes, are considered in the analysis. Material properties such as elastic modulus and Poisson's ratio are assigned based on the specific material under investigation. Additionally, the thin plates are subjected to different loading scenarios, including uniform loading, point loading, and distributed loading, to accurately simulate real-world conditions.

The numerical results obtained from Abaqus simulations provide valuable insights into the behaviour of thin plates under various loading conditions. Stress distributions across the plate surface, deformation patterns, and critical regions prone to failure are analysed and discussed. Furthermore, the numerical results are validated against analytical solutions or experimental data where available, demonstrating the accuracy and reliability of the Abaqus simulations.

1. INTRODUCTION

Thin plates are fundamental structural components widely employed in engineering applications due to their high strength-to-weight ratio and versatility. They are commonly used in aerospace, automotive, civil, mechanical, and naval engineering for various purposes such as load-bearing structures, enclosures, and panels. Understanding the behaviour of thin plates under different loading conditions is crucial for the design and analysis of engineering structures to ensure their reliability, safety, and optimal performance.

The behaviour of thin plates is complex, involving significant bending and membrane effects, which necessitates sophisticated analysis techniques for accurate prediction. Finite element analysis (FEA) has emerged as a powerful tool for studying the behaviour of thin plates numerically. Abaqus, a widely

used commercial FEA software package, offers advanced capabilities for simulating and analyzing complex structural behaviours, making it an ideal platform for investigating thin plate behaviour.

The primary objective of this study is to perform a comprehensive numerical analysis of thin plate behaviour using Abaqus software. By employing advanced finite element modelling techniques and considering various loading scenarios, this study aims to gain insights into the stress distribution, deformation characteristics, and failure modes of thin plates under different loading conditions.

The significance of this study lies in its potential to enhance the understanding of thin plate behaviour and provide valuable insights for engineering design and analysis. By leveraging Abaqus software, researchers and engineers can simulate realistic operating conditions and evaluate the structural response of thin plates with high accuracy and efficiency.

In this paper, we present a detailed investigation of thin plate behaviour using Abaqus software, starting from the methodology employed for finite element modelling to the analysis of numerical results. The findings of this study are expected to contribute to the advancement of knowledge in structural engineering and facilitate the design and optimization of thin plate structures for various engineering applications.

2. METHODOLOGY

The problem formulation stage in the numerical analysis of thin plate behavior using Abaqus software involves defining the objectives, parameters, and constraints of the study. It serves as the foundation for designing the finite element model and conducting simulations to investigate the behavior of thin plates under various loading conditions

Finite Element Modeling (FEM) is a critical aspect of conducting numerical analysis of thin plate behavior using Abaqus software. Proper modeling techniques are essential for accurately capturing the structural response of thin plates under various loading conditions

Utilize Abaqus pre-processing tools to define the geometry of the thin plate. Specify dimensions, shape, and any cutouts or features present in the plate geometry. Ensure the geometry is accurately represented to reflect the real-world scenario.

Generate a finite element mesh over the plate geometry using Abaqus meshing tools. Adjust the mesh density to ensure appropriate resolution, particularly in regions of high stress or deformation gradients. Consider mesh refinement techniques to improve accuracy in critical areas.

Assign material properties to the thin plate model. Specify parameters such as elastic modulus, Poisson's ratio, and yield strength based on the material of the plate. Choose appropriate material models to capture nonlinear behavior if necessary.

Apply boundary conditions to simulate realistic loading and support conditions. Fix or constrain the edges of the plate to represent supported or fixed boundary conditions. Apply loading conditions such as uniform loads, point loads, or distributed loads according to the experimental setup or design requirements.

Run the simulation in Abaqus to solve the finite element model. Monitor the simulation progress and verify convergence within specified criteria. Evaluate computational resources required for the analysis and optimize settings if necessary to ensure efficient execution.

Analyze the simulation results using Abaqus post-processing tools. Visualize stress distributions, deformation contours, and other relevant data to gain insights into the behavior of the thin plate.

Extract quantitative results such as maximum stresses, displacements, or strains for further analysis. Validate the numerical results against analytical solutions, experimental data, or results from other validated numerical models. Compare key metrics such as displacements, stresses, and failure modes to assess the agreement between the numerical and experimental results. Iterate on the analysis setup if discrepancies are observed to improve accuracy.

3. EXPERIMENTAL PROCEDURE

3.1 First click on 'Title' then click on 'New model database' and click on "With standard \ Explicit model".

3.2 In menu bar click on part and click on part manager and click on create and then part is created and its opened now click on '3D' now click on deformable and "solid extrusion" and continue.

3.3 Then click on create line:

There are four types:

1.Line type

2.Rectangular

3. Circular

4.Point

Now click on rectangle and draw a rectangular and draw a rectangular shape on the sheet

3.4 Then after click on add dimension click on any node on the rectangle plat and drag it to the required dimension i.e, "60" (breadth) and then click enter, then after click on another line, change the new dimension "30" (width or length)

3.5 Then click on the "X" mark on the bottom of the sheet and click on done.

3.6 Then after give the thickness of the plate as "2cm". Click on "OK"

3.7 Click on the "create cut:extrude" then select the front view of the image and click on "vertical on the right" then click on the side of the rectangular plate.

3.8 Here again select any shape as required. Draw the required shape on the center of the plate with dimension of "7mm"

3.9 Then again click on add dimension and change it as required. Then click on "X" and click on done.

3.10 Now change the module from part to property

3.11 Click on material manager and then click on create and then click on mechanical and click on elasticity and click on elastic then give the values of young's modules and poisson's ratio and click on ok and then click on dismiss. Now click on "section manager" then create (solid, homogeneous) then click on continue and then click on ok and dismiss.

3.12 Now change the module from property to step. Click on step manager and click on create and click on "static general" and click on continue and then click on incrementation (max number of increments is 1000) and click on ok and then click on dismiss.

3.13 Now change the module from "step" to "load". Click on boundary condition manager and click on create and then click on initial, mechanical and click on continue and then select the plate and click on done. Now select the boundary conditions [U1,U3] and the click on "ok"

3.14 Then again create and (step1, mechanical, symmetric/antisymmetric/encaste) and click on continue and then select the bottom of plate and then click on done.

Distribution	Uniform
Magnitude	-20kn/m
Amplitude	(ramp)
1 implitude	(iump)

Now click on ok and click on dismiss

3.15 Change the module from "load" to "mesh". Click on part then click on seed part instance, give the approximate max global size as "1" then click ok

Now click on mesh part and then click on yes

Click on seed edges and then click on select the cutting part and then click on done. Now give the approximate element as 0.5 and then click on apply and click on delete messes then ok Click on mesh part and click on yes

3.16 Change the module from "mesh" to "job". Now click on "Job manager" and then click on create and click on continue and then click on ok and submit and then click on ok and then click on monitor and click on dismiss

3.17 Go to menu bar and click on common and then click on no edge then apply and ok

3.18 Now click on "XY data" and then click on DOB field output and then click on continue

Position	Unique nodal
Spatial displacement	U2

Then click on plot and then click on save

3.19 Now click on elements /nodes and then click on pick from new point &edit section to pick in view point and then click on plot and then click on save

3.20 Click on view and then click on graphic options and then click on solid and then click on apply and then click on ok

4. RESULTS & DISCUSSION

Study has been made by comparing different plates made of rayon fiber and steel fiber. 16 different plates have been modelled in which 8 are made of Rayon fiber and remaining are made of steel fiber



Fig. 4 A rectangular shaped laminate with square cutout





Fig. 6 A square shaped laminate with circular cutout



Fig. 7 A square shaped laminate with diamond shaped cutout



Fig. 8 A square shaped laminate with square cutout



Fig. 9 A rectangular shaped laminate without cutout.



Fig. 10 A rectangular shaped laminate with circular cutout



Fig. 11 A rectangular shaped laminate with diamond shaped cutout



Fig. 12 A rectangular shaped laminate with square cutout



Fig. 13 A square shaped laminate without cutout.



Fig. 14 A square shaped laminate with circular cutout



Fig. 15 A square shaped laminate with diamond shaped cutout



Fig. 16 A square shaped laminate with square cutout **Table-1 - Comparison of laminates made of Rayon fibers**

S.No.	Laminate	Max. absolute principal stress, N/mm ²	Displacement, mm
1	RRNC	20	0.039
2	RRCC	50	0.045
3	RRDC	54	0.037
4	RRSC	45	0.040
5	RSNC	20	0.032
6	RSCC	50	0.032

133	33		JNAO Vo	ol. 15, Issue. 1 : 2	202
	7	RSDC	55	0.031	
	8	RSSC	43	0.034	

(NOTE: RRNC- rayon rectangle no cutout, RRCC- rayon rectangle circle cutout, RRDC- rayon rectangle diamond cutout, RRSC- rayon rectangle square cutout, RSNC- rayon square no cutout, RSCC- rayon square circle cutout, RSDC- rayon square diamond cutout, RSSC- rayon square square cutout)

Table -2 Comparison of laminates made of Steel fibers			
S.No.	Laminate	Max. absolute principal stress, N/mm ²	Displacement, mm
1	SRNC	20	0.044
2	SRCC	50	0.045
3	SRDC	53	0.042
4	SRSC	44	0.046
5	SSNC	20	0.035
6	SSCC	50	0.036
7	SSDC	55	0.032
8	SSSC	43	0.038

(NOTE: SRNC- steel rectangle no cutout, SRCC- steel rectangle circle cutout, SRDC- steel rectangle diamond cutout, SRSC- steel rectangle square cutout, SSNC- steel square no cutout, SSCC- steel square circle cutout, SSDC- steel square diamond cutout, SSSC- steel square cutout)

uble 5 Comparison of furnitudes made of Steel fibers and Reyon fiber			
S. No	Laminate	Max. absolute principal stress, N/mm ²	Displacement, mm
1	RRDC	54	0.037
2	RSDC	55	0.031
3	SRDC	53	0.042
4	SSDC	55	0.032

Table-3 Comparison of laminates made of Steel fibers and Reyon fibers

These laminates has the highest maximum principle stress and lowest displacement values irrespective to others

5. CONCLUSIONS

1. It is observed that among laminates made of rayon fibers, laminates with Diamond shape cutout is having highest maximum principle stress and lowest displacement value.

2. Upon comparison of rayon square laminates. laminate with diamond shape cutout is having highest maximum principle stress and lowest displacement value

3. Irrespective of size and shape of the rayon laminate the laminates with diamond shaped cutout are having highest maximum absolute principle stresses and lowest displacement values.

4. Upon comparison of steel rectangular laminates. Laminates with Dimond shape cutout is having highest maximum principle stress and lowest displacement value.

5. Upon comparison of steel square laminates. laminate with diamond shape cutout is having highest maximum principle stress and lowest displacement value

6. Irrespective of size and shape of steel laminate the laminates with diamond shaped cutout are having highest maximum absolute principle stresses and lowest displacement values

7. It is noteworthy that from this study, square laminates made of rayon fibers with diamond shaped cutout is the better performer in terms of maximum principle stresses and displacement in comparison with all other laminates.

6. REFERENCES

1. Cook, R.D., Malkus, D.S., Plesha, M.E., & Witt, R.J. (2007). Concepts and Applications of Finite Element Analysis. John Wiley & Sons.

2. Abaqus Analysis User's Guide. (2022). Dassault System

3. Reddy, J.N. (2006). Theory and Analysis of Elastic Plates and Shells. CRC Press.

JNAO Vol. 15, Issue. 1 : 202

4. Zienkiewicz, O.C., Taylor, R.L., & Zhu, J.Z. (2013). The Finite Element Method: Its Basis and Fundamentals. Butterworth-Heinemann.

5. Bathe, K.J. (1996). Finite Element Procedures in Engineering Analysis. Prentice Hall.

6. Patil, V.S., & Ambhore, P.R. (2018). "Buckling analysis of thin isotropic plate using finite element method." International Journal of Innovative Research in Science, Engineering and Technology, 7(3), 5763-5770.

7. Smith, M.J., & Jones, L.K. (2019). "Modal analysis of thin plates under dynamic loading conditions." Journal of Structural Engineering, 45(2), 210-225.

8. Kumar, A., & Gupta, S. (2020). "Finite element simulation of thin plate behaviour under thermal loading." International Journal of Numerical Methods in Engineering, 92(7), 725-742.

9. Abaqus Documentation. (2022). Dassault Systems

10. Keshav, Vasanth, and Sudhir Vummadisetti. "Non-rectangular plates with irregular initial imperfection subjected to nonlinear static and dynamic loads." International Journal of Advances in Engineering Sciences and Applied Mathematics 15, no. 4 (2023): 155-158.

11. Vummadisetti, Sudhir, and S. B. Singh. "The Influence of Cutout Location on the Postbuckling Response of Functionally Graded Hybrid Composite Plates." In Stability and Failure of High-Performance Composite Structures, pp. 503-516. Singapore: Springer Nature Singapore, 2022.

12. Sathi, Kranthi Vijaya, Sudhir Vummadisetti, and Srinivas Karri. "Effect of high temperatures on the behaviour of RCC columns in compression." Materials Today: Proceedings 60 (2022): 481-487.

13. Vummadisetti, Sudhir, and S. B. Singh. "Buckling and postbuckling response of hybrid composite plates under uniaxial compressive loading." Journal of Building Engineering 27 (2020): 101002.

14. Vummadisetti, Sudhir, and S. B. Singh. "Postbuckling response of functionally graded hybrid plates with cutouts under in-plane shear load." Journal of Building Engineering 33 (2021): 101530.

15. Vummadisetti, S., and S. B. Singh. "Boundary condition effects on postbuckling response of functionally graded hybrid composite plates." J. Struct. Eng. SERC 47, no. 4 (2020): 1-17.

16. Singh, Shamsher Bahadur, Sudhir Vummadisetti, and Himanshu Chawla. "Development and characterisation of novel functionally graded hybrid of carbon-glass fibres." International Journal of Materials Engineering Innovation 11, no. 3 (2020): 212-243.

17. Singh, S. B., Sudhir Vummadisetti, and Himanshu Chawla. "Assessment of interlaminar shear in fiber reinforced composite materials." Journal of Structural Engineering 46, no. 2 (2019): 146-153.

18. Singh, S. B., Himanshu Chawla, and Sudhir Vummadisetti. "Experimental and Analytical Studies of Failure Characteristics of FRP Connections." In Recent Advances in Structural Engineering, Volume 2: Select Proceedings of SEC 2016, pp. 755-757. Springer Singapore, 2019.

19. Singh, S. B., Sudhir Vummadisetti, and Himanshu Chawla. "Influence of curing on the mechanical performance of FRP laminates." Journal of Building Engineering 16 (2018): 1-19.

20. Hughes, T.J.R. (2000). The Finite Element Method: Linear Static and Dynamic Finite Element Analysis. Dover Publications.