Analysis Of Stiffened Plate

Nandish B[1], Abijith[2], Jayashankaramurthy R[3], Jubil C Govind[4], Muhammed Hafis[5]
[1, 2] Department of Mechanical Engineering
[1, 2, 3, 4, 5] Coorg Institute of Technology, Ponnampet, Kodagu, Karnataka, India

Abstract- *In the present world, the growing demand of architecturally efficient and significantly higher strength to weight ratio structures is mostly served by Stiffened plates. The main purpose of using stiffened plates is their significantly high stiffness to weight ratio compared to unstiffened plates.*

To increase the stability of the plates and to withstand the stress and deformations developed it is not possible to increase the thickness of the plates. Instead of increasing the thickness of plates and thereby increasing the cost of materials, stiffeners are provided. The advantage of stiffening lies in achieving an economical and lightweight design. By increasing the number of supporting grids, the stiffness of the plate can be increased.

Bending and buckling are possible only at the time on failure of supporting grids. Due to the increase in overall stiffness of the system, enhanced load carrying capacity and stability characteristics are obtained. Stiffened plates are used extensively for aerospace and marine constructions, highway bridges, railway wagons, and floor slab as in building structures etc.

I. INTRODUCTION

Rectangular plates subjected to predominantly compressive forces can be found in primary elements of ships structures and in other marine structures. This justifies the interest of having a complete understanding and a prediction capability for the compressive strength of unstiffened plate elements. Plate elements are generally part of stiffened panels which are in fact the structural components whose strength one is finally interested to predict. However, global failure of the stiffened panels is usually avoided by design so that the usual failure modes are interface failure of the plate stiffener assembly or failure of the plate elements between stiffeners.

The different prediction methods assess the strength of the plate and stiffener assembly by properly accounting for the contribution of each one. Therefore, in addition to the interest in its own right, the prediction of plate strength is also a prerequisite for assessing the strength of stiffened plates. Sea shells, leaves, trees, vegetables all of these are in fact, stiffened structures observations of structures created by nature indicate that in most cases strength and rigidity depend not only on the material but also upon its form. This fact was probably noticed long ago by some observers and resulted in the creation of artificial structural elements having high bearing capacity mainly due to their form, such as, girders, shells.

II. PROBLEM FORMULATION

Objectives

The main objectives of Modelling and Finite Element Analysis of Stiffened plate analysis are as follows.

- To analyse the load sustainability of the Stiffened plate at described loading and boundary conditions.
- To compare the Stiffened plates with unstiffened plates at same loading and boundary conditions of different materials (Steel and Aluminium).
- To prove the weight optimization.

Methodology

Following are the basic steps for the Analysis of Stiffened plates.

1. Modelling: Modelling of stiffened and unstiffened plate is done by CATIA V5 R21 as per the geometrical considerations.
2. Meshing: Meshing is done by using ANSYS R15.0 workbench. It is necessary to understand how the structure is likely to behave and how elements are able to behave.
3. Boundary conditions: Edges of the plates are fixed and applying the uniform pressure on top surface of the plate.
4. Analysis: Analysis is done by using ANSYS R15.0 workbench.
5. Comparison of results.

III. MODELING AND ANALYSIS

Modeling by CATIA V5 R21 Software

CATIA views for Computer Aided Three dimensional Supportive Application. CATIA offers a solution to shape design, styling, surfacing workflow and visualization to create, modify, and validate complex innovative shapes from industrial design to Class-A surfacing with the ICEM surfacing technologies. CATIA supports multiple stages of product design whether started from scratch or from 2D sketches. CATIA V5 is able to read and produce STEP format files for reverse engineering and surface reuse. In this analysis we are using CATIA V5 R21 for modeling of stiffened and unstiffened plates as per the geometrical considerations and the files are saved as .STEP format.

Modeling of Stiffened Plates

For analysis first step is to create a solid model as per the considered dimensions. Following are the steps used to create a solid model of stiffened plate in CATIA V5 R21 software.

- Open the software and select START then MECHANICAL DESIGN. In that select PART DESIGN option as shown in Fig. below.

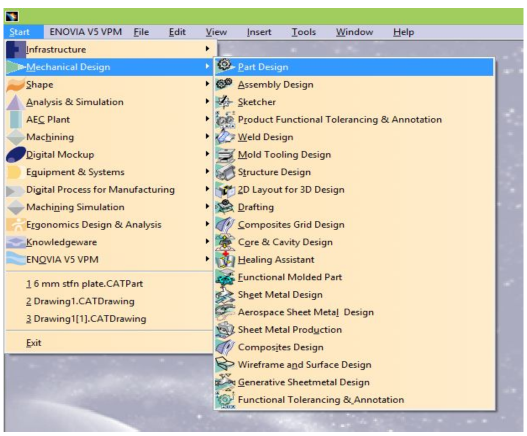


Figure 1. Selection of Design Type

- Enter the PART NAME and then click OK.
- Select the sketching plane as YZ or Front plane.
- Draw the sketch as per the dimensions considered by using commands such as LINE, RECTANGLE, CIRCLE, AXIS, POINT, TRIM, SPLIT, MIRROR, CONSTRAINTS as shown in Fig. below.

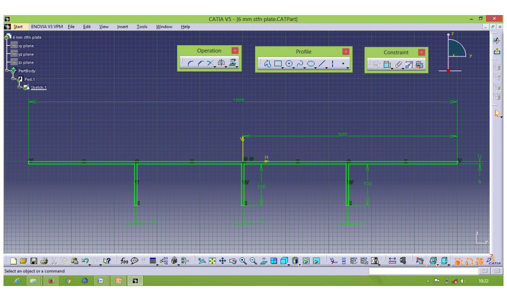


Figure 2. 2D Profile Sketch of Stiffened Plate

- Click on RETURN it will return to 3D mode part.
- Select the PAD from feature tool bar.
- Select the sketch to be protruded as shown in Fig. below

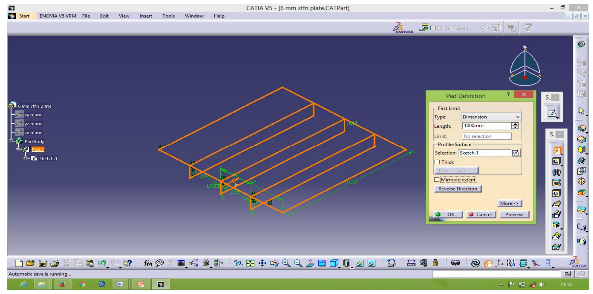


Figure 3. Protruded View of Stiffened plate

- Enter the length to be protruded as shown in Fig. below.

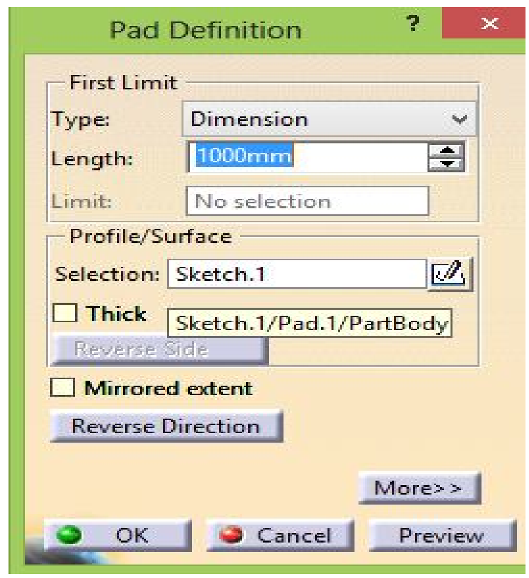


Figure 4. Pad Definition

- Click on OK and then click on FIT as shown in Fig. below.

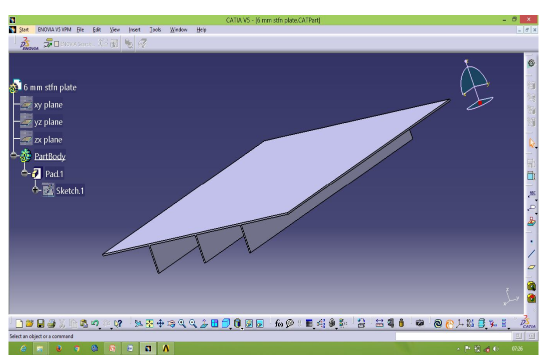


Figure 5. Modeling of Stiffened Plate in CATIA V5 R21

- Finally SAVE the part as .STEP format as shown in Fig. below.

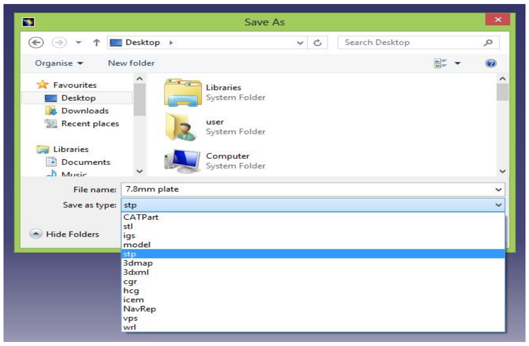


Figure 6. SAVE Format

Modeling of Unstiffened Plates

For analysis first step is to create a solid model as per the considered dimensions. Following are the steps used to create a solid model of unstiffened plate in CATIA V5 R21 software.

- Open the software and select START then MECHANICAL DESIGN. In that select PART DESIGN option as shown in Fig. below.

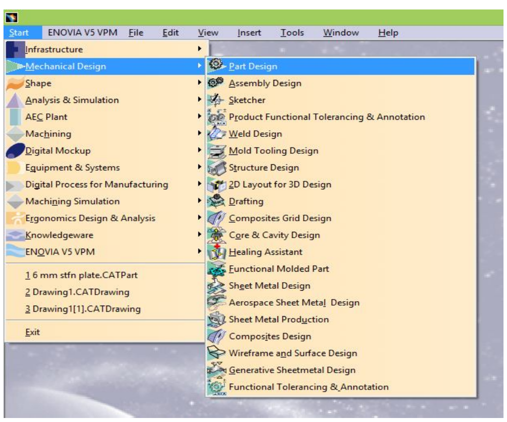


Figure 7. Selection of Design Type

- Enter the PART NAME and then click OK.
- Select the sketching plane as YZ or Front plane.
- Draw the sketch as per the dimensions considered by using commands such as LINE, RECTANGLE, CIRCLE, AXIS, POINT, TRIM, SPLIT, MIRROR, CONSTRAINTS as shown in Fig. below.

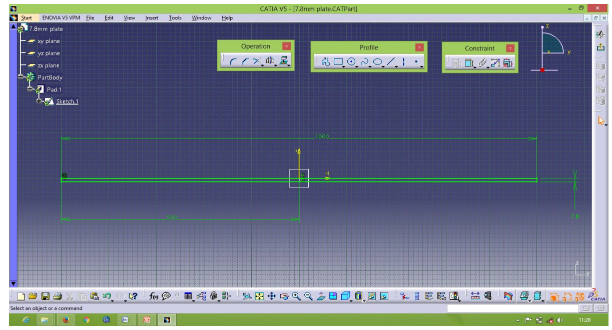


Figure 8. 2D Profile Sketch of Unstiffened Plate

- Click on RETURN it will return to 3D mode part.
- Select the PAD from feature tool bar.
- Select the sketch to be protruded as shown in Fig. below.

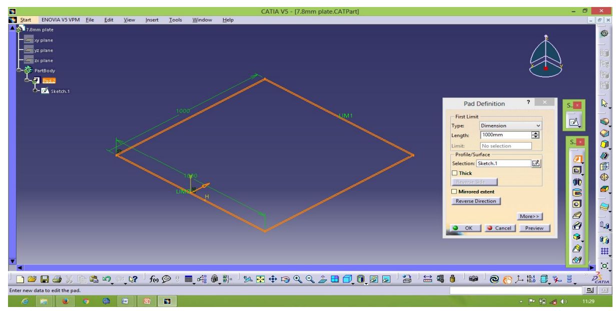


Figure 9. Protruded View of Unstiffened Plate

- Enter the length to be protruded as shown in Fig. below.

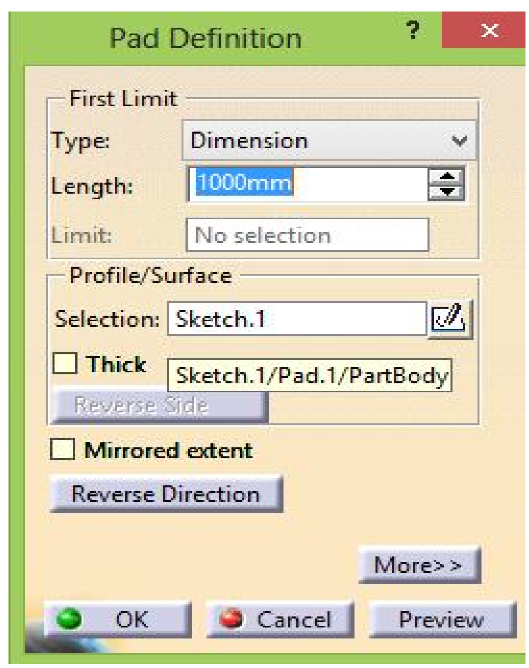


Figure 10. Pad Definition

- Click on OK and then click on FIT as shown in Fig. below.

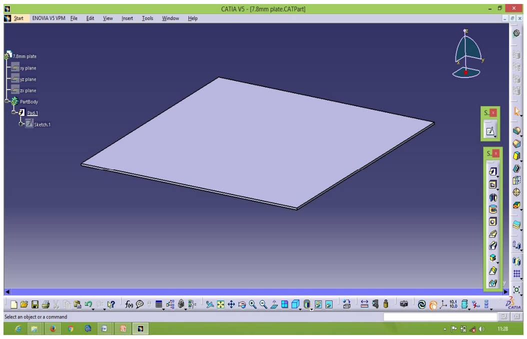


Figure 11. Modeling of Unstiffened Plate in CATIA V5 R21

- Finally SAVE the part as .STEP format as shown in Fig. below.

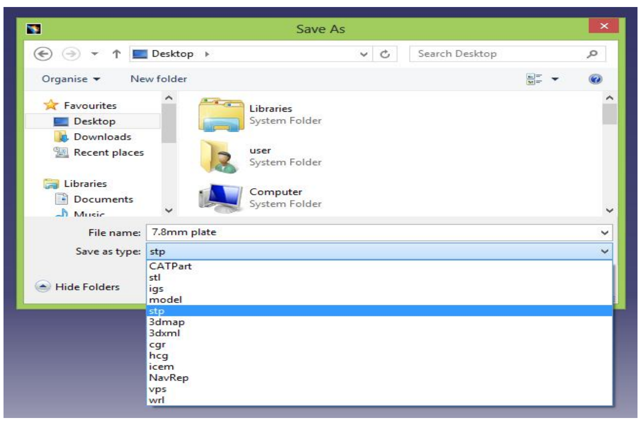
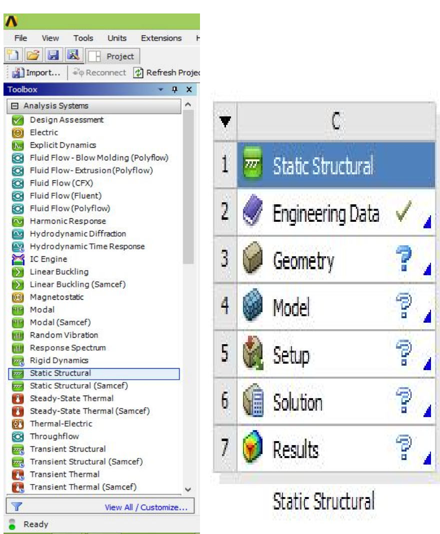


Figure 12. SAVE Format

- Open the ANSYS software and Select Static Structural from the tool box.



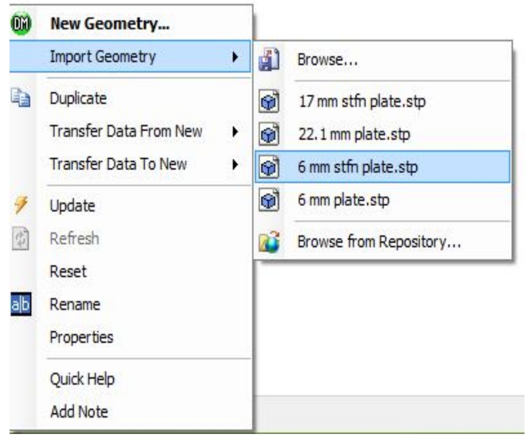
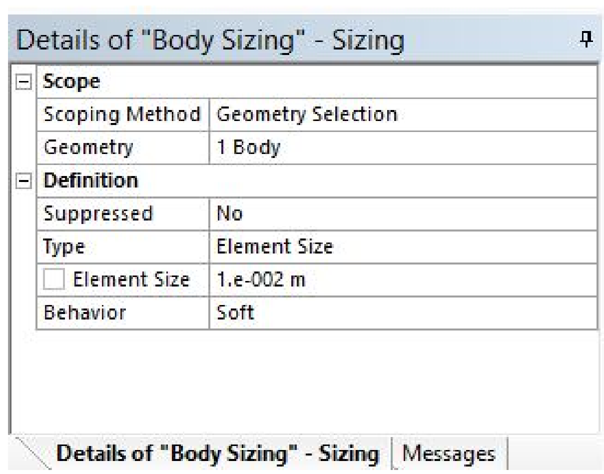


Figure 13. Selection of Analysis Type

- Import the geometry as in the form of .STEP format which is already saved as shown in Fig. below. And Click on MODEL then click on EDIT.
- Click on MESH in the OUTLINE tool bar. Right click on MESH option then click on BODY SIZING, the details of Body sizing window will be appeared in OUTLINE tool bar.



Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	1.e-002 m
Behavior	Soft

Details of "Body Sizing" - Sizing | Messages

Figure 14. Details of Meshing

- Select the mesh type and enter the value of element size as 0.01m as shown in the Fig. below.
- Select the geometry to be mesh.
- Click on APPLY then right click on MESH.
- Then click on GENERATE MESH.

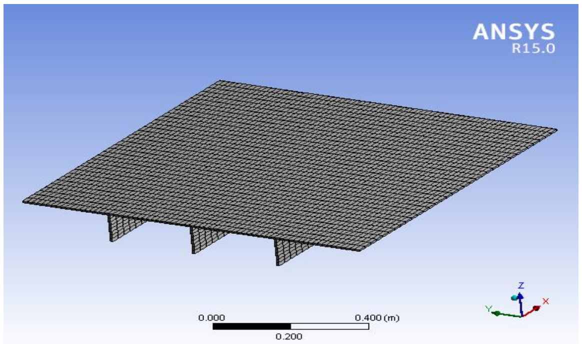
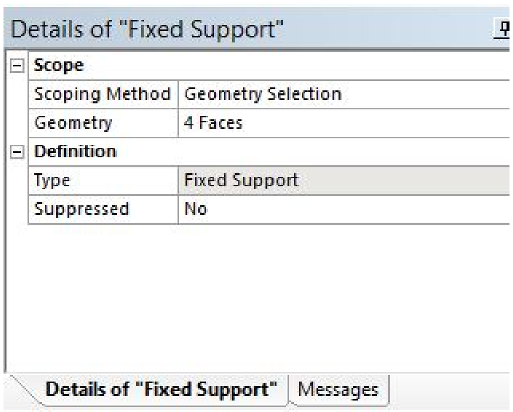


Figure 15. Meshing of Stiffened Plate in ANSYS R15.0

- After importing the geometry and completion of meshing as mentioned above, then click on STATIC STRUCTURAL in outline tool bar.
- Click on SUPPORTS then select the FIXED DISPLACEMENTS.



Details of "Fixed Support"

Scope	
Scoping Method	Geometry Selection
Geometry	4 Faces
Definition	
Type	Fixed Support
Suppressed	No

Details of "Fixed Support" | Messages

Figure 16. Details of Fixed Supports

- Select the all 4 faces of the plate as shown in Fig. below.
- Click on APPLY in Details of Fixed Support window.

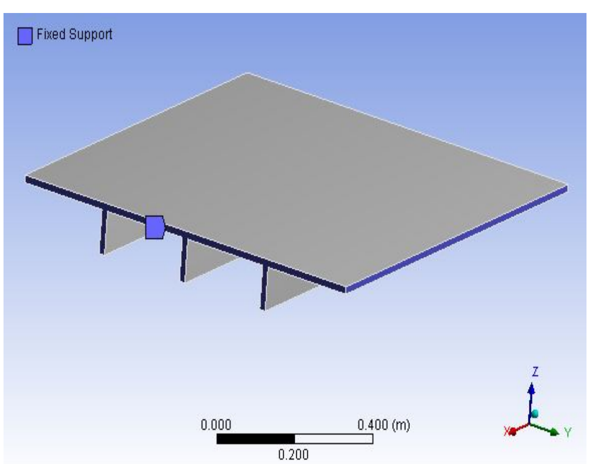
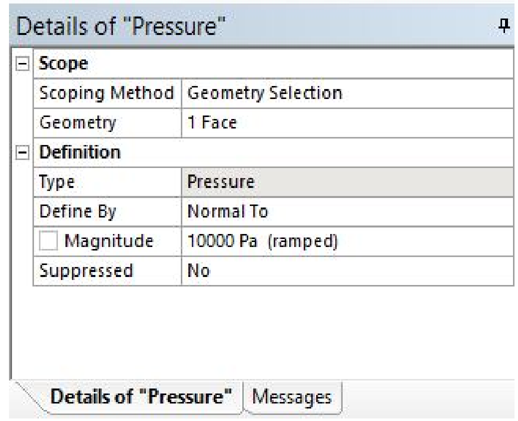


Figure 17.

- Again click on STATIC STRUCTURAL.
- Click on LOADS and select the PRESSURE option.
- Select the top surface of the plate, and define the MAGNITUDE and DIRECTION of pressure.



Details of "Pressure"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Pressure
Define By	Normal To
Magnitude	10000 Pa (ramped)
Suppressed	No

Details of "Pressure" | Messages

Figure 18. Details of Pressure

- Enter the value of pressure to be applied on surface in Details of Pressure window as shown in Fig. above.
- Click on APPLY and in outline tool box.

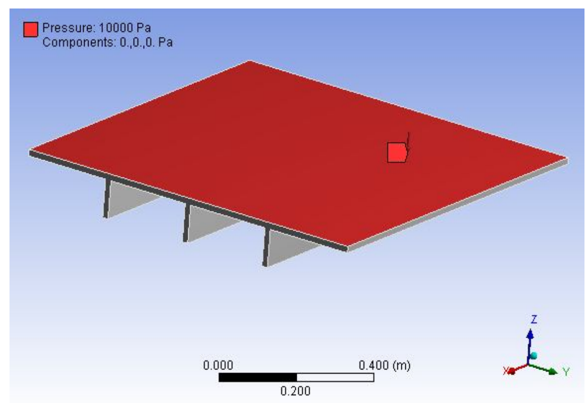


Figure 19. Pressure Applied on Stiffened Plate

IV. RESULTS AND COMPARISON

Analysis

ANSYS is a software tool for analysis of any mechanical components it allows us to compute Von-mises stress (Equivalent stress), Maximum Principal stress and Total deformation at any point with high accuracy, whether the component is on dynamic or static loading. For this study, analysis is done by using ANSYS R15.0. Following are the steps used for the analysis of stiffened plate and unstiffened plate.

- After importing the geometry and completion of meshing, applying the boundary and loading conditions as mentioned above, and then click on SOLUTION in outline tool bar.

- Then click on DEFORMATION and select TOTAL DEFORMATION.
- Click on STRESS and select VON-MISES STRESS (EQUIVALENT STRESS) and MAXIMUM PRINCIPAL STRESS.
- Finally click on SOLUTION and then click on SOLVE in the feature tool bar.

Results and Comparison of Total Deformation for 10000 Pa

Stiffened and Unstiffened Plate of STEEL (61.23 kg)

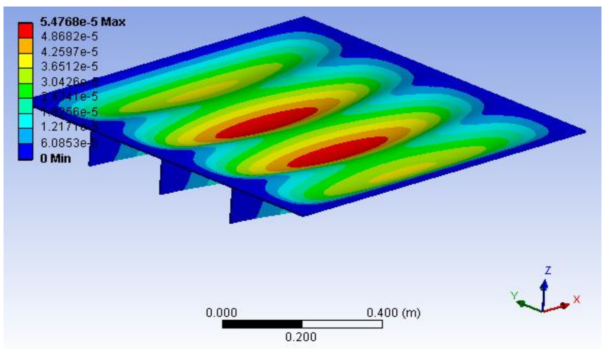


Figure 20. Total Deformation of STEEL Stiffened Plate

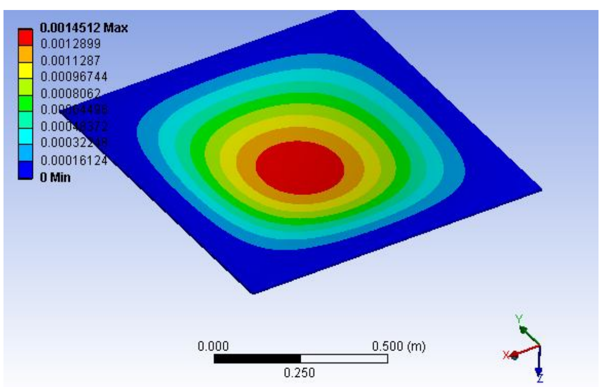


Figure 21. Total Deformation of STEEL Unstiffened Plate

Stiffened and Unstiffened Plate of ALUMINIUM (61.23 kg)

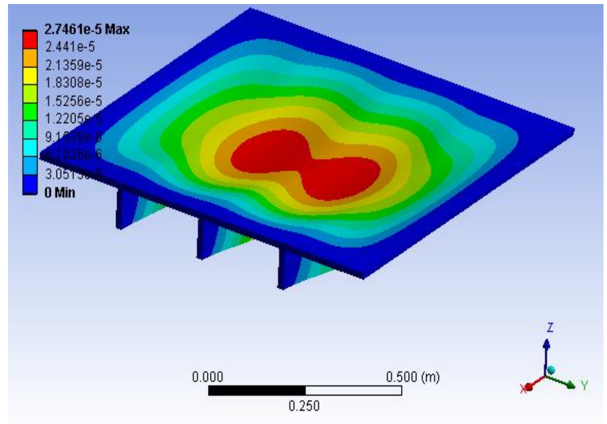


Figure 22. Total Deformation of ALUMINIUM Stiffened Plate

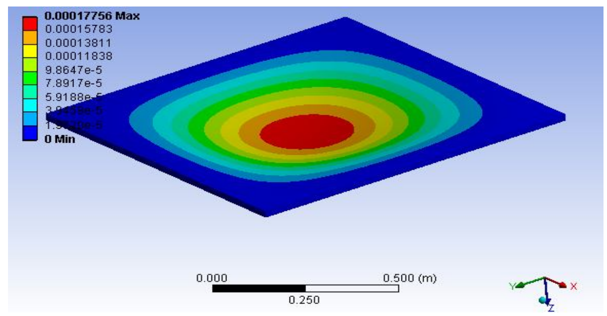


Figure 23. Total Deformation of ALUMINIUM Unstiffened Plate

The first two Fig. above show the total deformation for the steel stiffened and unstiffened plate respectively. The other two Fig. are also shows the total deformation for the aluminium stiffened and unstiffened plates respectively. From these results we can observe that the deformation is more at middle region of the plate.

Table 1. Comparison of Total Deformation Results

Type of Material	Type of Plate	Thickness in (mm)	Mass in (kg)	Total deformation in (m)	
				Analytical Results	Numerical Results
STEEL	Stiffened Plate	6 mm	61.23	0.0000547	0.0000514
	Unstiffened Plate	7.8 mm	61.23	0.001451	0.001457
ALUMINIUM	Stiffened Plate	17 mm	61.23	0.0000274	0.0000279
	Unstiffened Plate	22.1 mm	61.23	0.0001775	0.000171

The above Table show comparison between Stiffened and Unstiffened plate for Total deformation at the load of 10000 Pa. This shows that, the total deformation of the Stiffened plate of 61.23 kg is 27.3% less than the unstiffened plate of same weight at same load and boundary conditions. And also the load sustainability of Stiffened plate is more than the unstiffened plate of same weight. From the obtained result we can also observe the weight optimization.

REFERENCES

[1] Deepak Kumar Singh, S. K. Duggal, “Analysis of Stiffened Plates using FEM”, International Research Journal of Engineering and Technology, Volume 2, pp. 1650-165, July 2015.

[2] Dr. Alice Methai, Shiney Vaeghese, “Finite Element Buckling Analysis of Stiffened Plates”, International Journal of Engineering Research and Development, Volume 10, pp. 79-83, 2014.

[3] Ali Yeilaghi Tamijani, Rakesh K. Kapania, “Buckling and Static Analysis of Stiffened Plates”, American Institute of Aeronautics and Astronautics, Volume 17, pp. 1-24, 2009.

[4] Dr. Hathem H., Dr. Adil A. Moosawy, “Optimum Design of Stiffened Plates Subjected to Static Loading”, Al - Khawarizmi Engineering Journal, Volume 4, pp. 46-58, 2008.

[5] Gabor M. Voros, “Finite Element Analysis of Stiffened Plates”, Periodica Polytechnic, pp. 105-112, 2007.

[6] Khosrow Ghavami, Mohammad Reza, “Numerical and Experimental Investigations on the Compression Behaviour of Stiffened Plates”, Journal of Constructional Steel Research, pp. 1087-1100, 2006.

[7] Roslan Abdul Rahman, “Finite Element Modeling, Correlation and Model Updating of Stiffened Plate”, Mechanical Engineering Technology, University of Malaysia, Volume 16, pp. 91-106, 2003.

[8] Ravi Shankar Bellur Ramaswamy, “Optimal Design of Stiffened Plates”, Graduate Department of Aerospace Science and Engineering, University of Toronto, pp. 12-100, 1999.