Proc. of Int. Conf. on Current Trends in Eng., Science and Technology, ICCTEST

Finite Element Analysis of Front End Module for Fracture Load Case Condition

Yashvanth Kumar
S.J.B.I.T/ Mechanical Department/ Bangalore/ India
yashkstban@gmail.com

***Abstract*—Automotive front-end modules (FEMs) typically are multipieces assemblies that integrate a large number of components like: Engine Compartment, forward lighting, radiators and cooling fans, air conditioning (A/C) condensers, grille-opening reinforcement (GOR) panels, crumple zones, bumpers with decorative fascia, hood latches, washer bottles, plus electronics and wiring. Front End Module component consists of mainly four parts. They are: Frame, Polyamide Ribs, Front End Module Bracket, Z-Braces.**

The function of the frame is to carry the load coming from the hood latch. In the series, Frame is made of steel, since model weight is a major factor for such a huge component which also comes close to the engine compartment. It is highly recommended to optimize the frequency of Front Module by reducing the weight. Hence the project emphasize majorly on weight optimization of Front End Module assembly. Based on this fact, a proposal is made on changing the material of Front End Module Frame with composite. In this project frame of the Front End Module is made by carbon fiber reinforcement laminated composite. It is subjected to linear static analysis using ABAQUS solver software. From the results it is shown that frame made by laminated composite has 62 % reduction in mass when compared to steel. Also it is shown that the strength of frame made by steel and laminated composite is almost equal for 2KN load case and for 5KN load case there is drastic rise in strength of frame made by laminated composite. It is observed that carbon fiber material is the best replacement material for frame in terms of stiffness and mass only if cost is not a major issue

I. INTRODUCTION

Automotive front-end modules (FEMs) typically are multi-piece assemblies that integrate a large number of components: forward lighting, radiators and cooling fans, air conditioning (A/C) condensers, grille-opening reinforcement (GOR) panels, crumple zones, bumpers with decorative fascia, hood latches, washer bottles, plus electronics and wiring, although specific components can vary by tier supplier and OEM. Rather than using the traditional method of piece metal assembly on the OEM production line, FEMs, which are supplied by a tier integrator, provide a complete system for closing out a vehicle's front end at the assembly line. Because of this, they work best with so-called open-architecture builds that are reinforced to support the front-end module. To date, FEMs have been used on compact and midsize cars, and more recently on large sedans, all with monocoque/unibody architecture. They also have found application on SUVs and full-size pickups that feature body-on-frame constructions. While FEMs might not yet be appropriate for every vehicle, they bring many benefits to automakers in appropriate circumstances.

Grenze ID: 02.ICCTEST.2017.1.34

II. METHODOLOGY

Finite element analysis of front end module involves meshing of different components i.e., Metal frame, polyamide ribs, Z-brace, frame bracket using 2D shell elements. The components are integrated together using 1D elements and defining contact pair between the parts. The assembly is then fixed at the mounting location and a load of 2KN and 5KN is applied at room temperature at the latch point. A comparison study is made between series model and the proposed design.

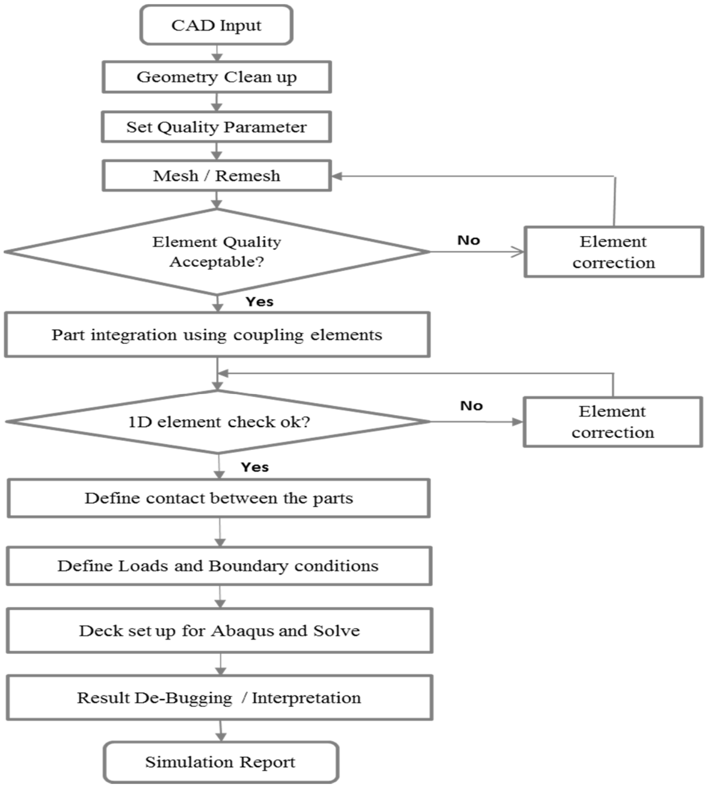


Figure 1. Flow Chart of the Project

III. PROJECT OVERVIEW

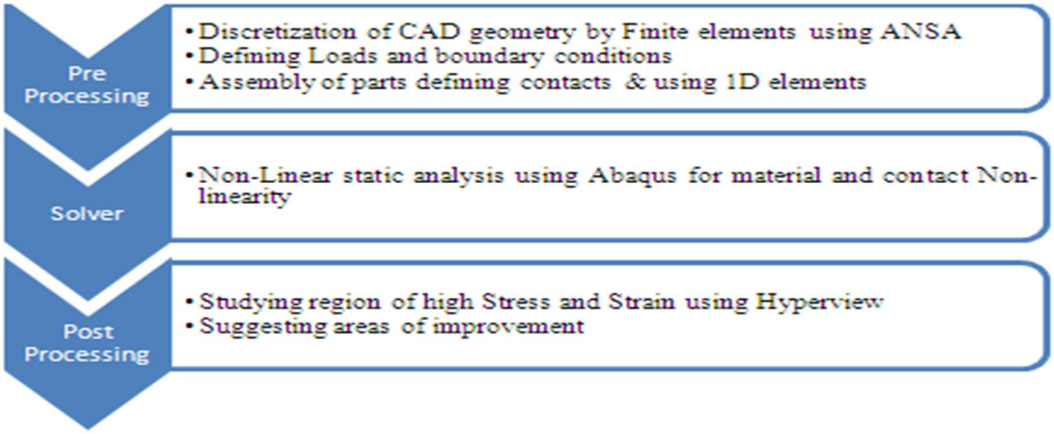


Figure 2. Project Overview

A. Parts of Front End Module

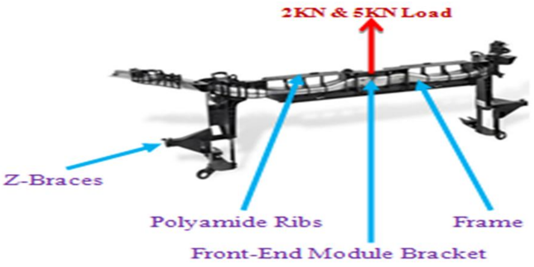


Figure 3.Parts of Front End Module

IV. LOAD AND BOUNDARY CONDITIONS

Strength analysis of Front end module is performed for Fracture load conditions, in which a load of 2KN and 5 KN is applied at the latch point (generally at the center lock where the hood connects the front module) at room temperature. The analysis is carried out for 2KN and 5KNloading at room temperature. Load vs. deflection at the loading point is extracted. Deflection target for 2KN calculated at 23°C in global "z" direction are lesser or equal to 8mm.

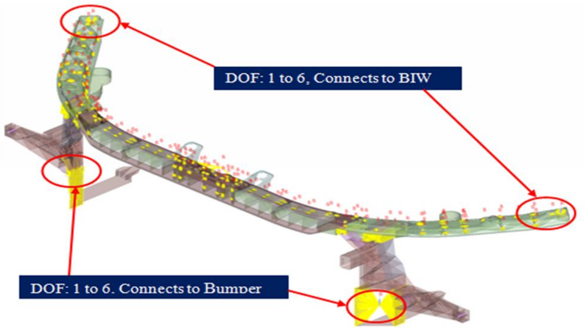
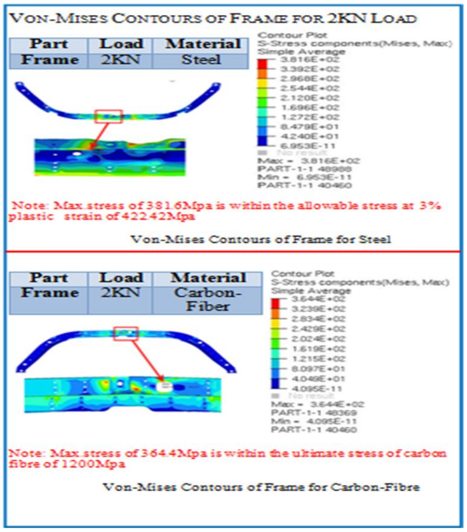


Figure 4.Boundary Condition

V. RESULTS AND DISCUSSION



VON-MISES CONTOURS OF FRAME FOR 2KN LOAD

Part	Load	Material
Frame	2KN	Steel

Note: Max stress of 381.6Mpa is within the allowable stress at 3% plastic strain of 422.42Mpa

Von-Mises Contours of Frame for Steel

Part	Load	Material
Frame	2KN	Carbon-Fiber

Note: Max stress of 364.4Mpa is within the ultimate stress of carbon fibre of 1200Mpa

Von-Mises Contours of Frame for Carbon-Fibre

Figure 5.Von-Mises Contours of Frame for 2KN Load Case

VI . LOAD VS. DISPLACEMENT FOR 2KN LOAD

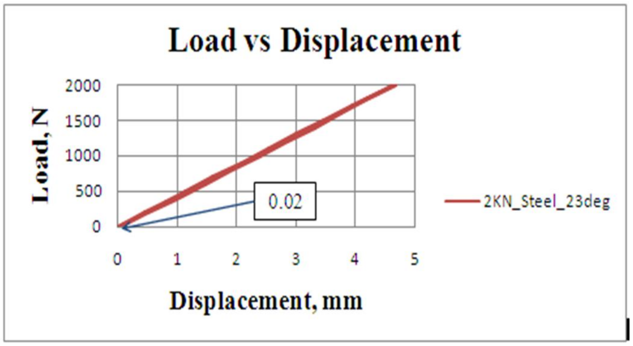


Figure 6. Load vs. Displacement of 2KN for Steel

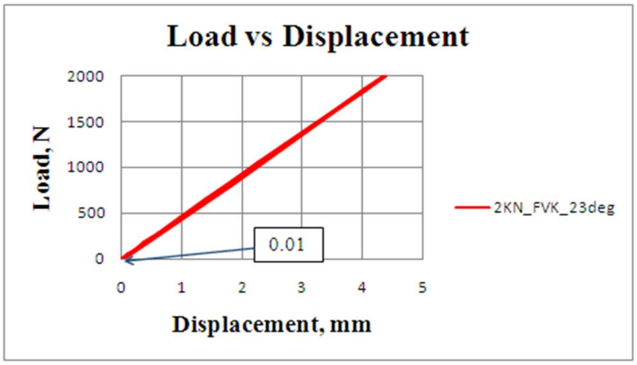
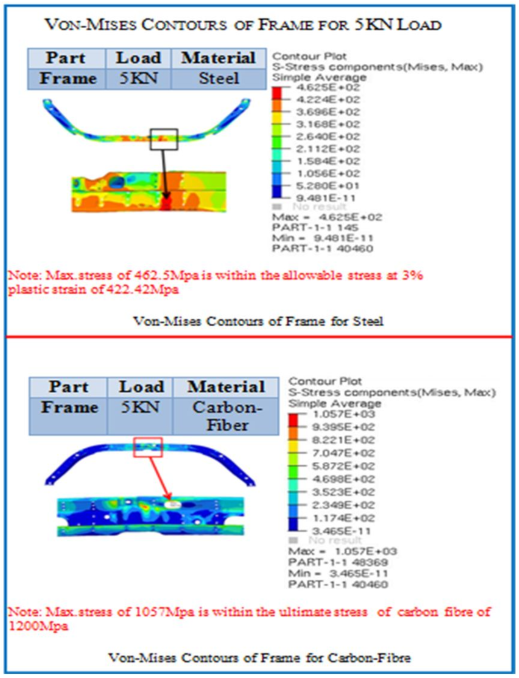


Figure 7. Load vs. Displacement of 2KN for Carbon Fibre

VII. VON-MISES CONTOURS OF FRAME FOR 5KN LOAD


VON-MISES CONTOURS OF FRAME FOR 5KN LOAD

Part	Load	Material
Frame	5KN	Steel

Note: Max.stress of 462.5Mpa is within the allowable stress at 3% plastic strain of 422.42Mpa

Von-Mises Contours of Frame for Steel

Part	Load	Material
Frame	5KN	Carbon-Fiber

Note: Max.stress of 1057Mpa is within the ultimate stress of carbon fibre of 1200Mpa

Von-Mises Contours of Frame for Carbon-Fibre

VIII. LOAD VS. DISPLACEMENT FOR 5KN LOAD

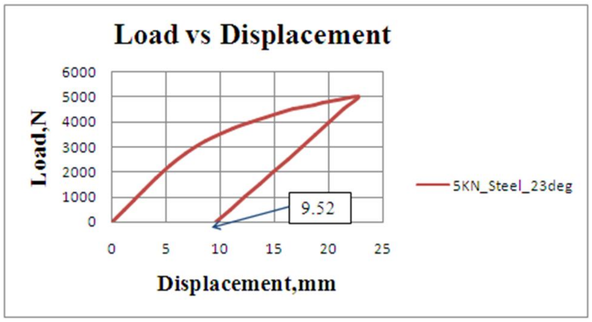


Figure 9. Load vs. Displacement of 5KN for Steel

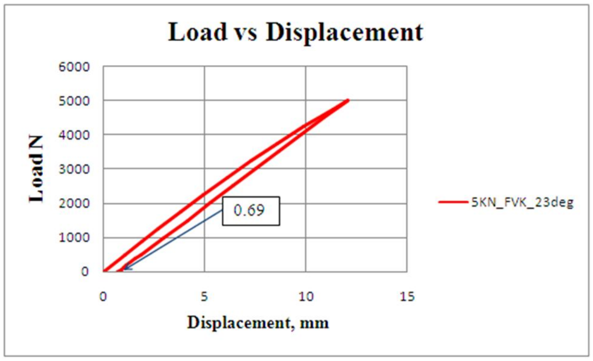
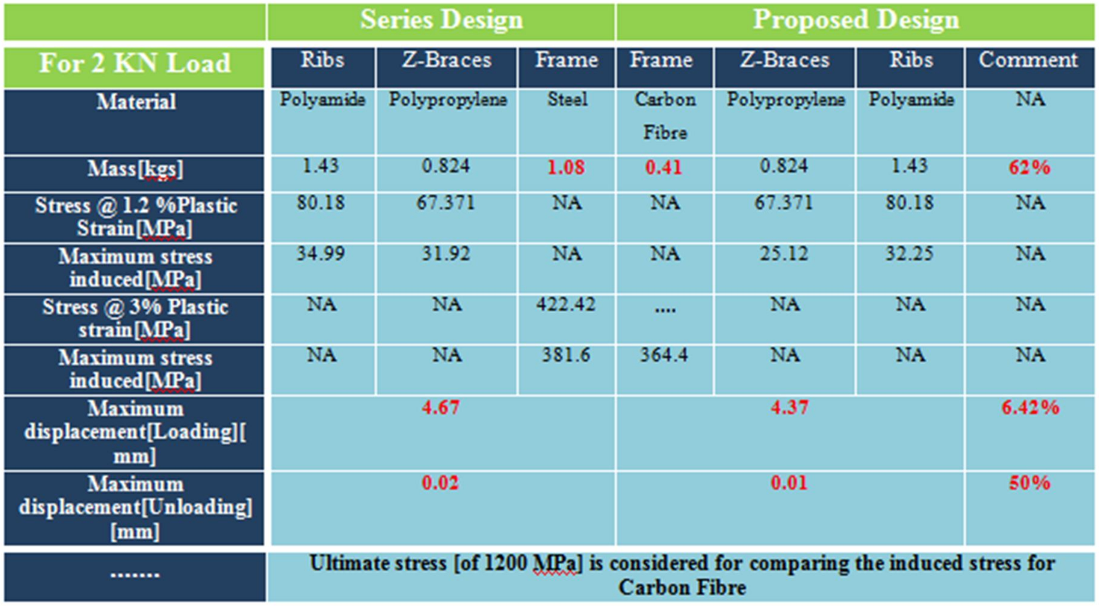


Figure 10. Load vs. Displacement of 5KN for Carbon Fibre

IX. RESULTS ANALYSIS FOR 2KN LOAD

TABLE I. RESULTS FOR 2KN OF STEEL & CARBON FIBER MATERIALS



	Series Design			Proposed Design			
For 2 KN Load	**Ribs**	**Z-Braces**	**Frame**	**Frame**	**Z-Braces**	**Ribs**	**Comment**
Material	Polyamide	Polypropylene	Steel	Carbon Fibre	Polypropylene	Polyamide	NA
Mass[kgs]	1.43	0.824	**1.08**	**0.41**	0.824	1.43	**62%**
Stress @ 1.2 %Plastic Strain[MPa]	80.18	67.371	NA	NA	67.371	80.18	NA
Maximum stress induced[MPa]	34.99	31.92	NA	NA	25.12	32.25	NA
Stress @ 3% Plastic strain[MPa]	NA	NA	422.42	....	NA	NA	NA
Maximum stress induced[MPa]	NA	NA	381.6	364.4	NA	NA	NA
Maximum displacement[Loading][mm]	**4.67**			**4.37**			**6.42%**
Maximum displacement[Unloading][mm]	**0.02**			**0.01**			**50%**
....	**Ultimate stress [of 1200 MPa] is considered for comparing the induced stress for Carbon Fibre**						

Calculation for 2KN load case

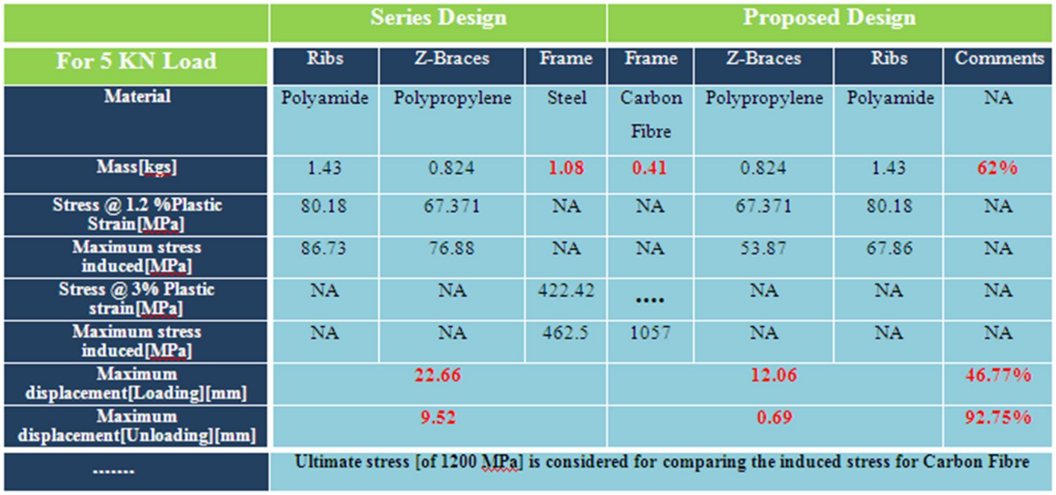
$$\frac{1.08 - 0.41}{1.08} * 100 = 62\ \%$$

$$\frac{4.67 - 4.37}{4.67} * 100 = 6.42\ \%$$

$$\frac{0.02 - 0.01}{0.02} * 100 = 50\ \%$$

X. RESULTS ANALYSIS FOR 5KN LOAD

TABLE II. RESULTS FOR 5KN OF STEEL & CARBON FIBER MATERIALS



	Series Design			Proposed Design			
For 5 KN Load	Ribs	Z-Braces	Frame	Frame	Z-Braces	Ribs	Comments
Material	Polyamide	Polypropylene	Steel	Carbon Fibre	Polypropylene	Polyamide	NA
Mass[kgs]	1.43	0.824	1.08	0.41	0.824	1.43	62%
Stress @ 1.2 %Plastic Strain[MPa]	80.18	67.371	NA	NA	67.371	80.18	NA
Maximum stress induced[MPa]	86.73	76.88	NA	NA	53.87	67.86	NA
Stress @ 3% Plastic strain[MPa]	NA	NA	422.42	....	NA	NA	NA
Maximum stress induced[MPa]	NA	NA	462.5	1057	NA	NA	NA
Maximum displacement[Loading][mm]	22.66			12.06			46.77%
Maximum displacement[Unloading][mm]	9.52			0.69			92.75%
....	Ultimate stress [of 1200 MPa] is considered for comparing the induced stress for Carbon Fibre						

Calculation for 5KN load case

$$\frac{1.08 - 0.41}{1.08} * 100 = 62\%$$

$$\frac{22.66 - 12.06}{22.66} * 100 = 46.77\%$$

$$\frac{9.52 - 0.69}{9.52} * 100 = 92.75\%$$

XI. CONCLUSION

- Based on the result summary it is observed that the proposed design has 62% saving in mass compared to the series model.
- Maximum displacement (loading) for the proposed design is 6.42% lesser than the series model for 2KN load case and 46.77% for 5KN load case.
- Maximum displacement (Unloading) for the proposed design is 50% lesser than the series model for 2KN load case and 92.72% for 5KN load case.
- Based on these observations, it is observed that carbon fibre design is the best replaceable material for the frame to achieve better performance in terms of stiffness and mass only if cost is not a major concern.

XII. SCOPE FOR FUTURE WORK

Based on the simulation result, it is evident that composite (carbon fibre) is the best replaceable material for steel frame if cost is not a major concern. Further iterations were carried out replacing steel frame by different materials like Aluminium, Organoblech (Short fibre Organic composite) and Polyamide, which exhibited lower performance compared to steel due to its lower modulus of rigidity in comparison with steel. However there is a considerable improvement in mass. Topology optimization can be carried out further to remove material at those regions which do not contribute for the overall performance.

ACKNOWLEDGMENT

Project work is the job of great enormity and it can’t be accomplished by an individual all by them self. Eventually I am grateful to a number of individuals whose professional guidance, assistance and encouragement have made it a pleasant endeavour to present this project.

I take this opportunity to express my profound gratitude to Dr.S.B.ANADINNI, Principal, VijayaVittala Institute of Technology, for his constant support and encouragement.

I express my deep gratitude to Dr.KRISHNAMURTHY.N, Head of the Department, Department Mechanical Engineering, for providing me an opportunity for fulfilling my goal.

I express my gratitude to the project coordinator, Prof.HARISH.H, Assistant Professor, Department of Mechanical Engg, for extending support for the successful completion of the project.

Guidance and deadlines play a very important role in successful project work. I express my warm regards to Dr.KRISHNAMURTHY.N Head of the Department, Department of Mechanical Engineering, Project guide, for his skilful guidance, constant supervision, timely suggestion and constructive criticism in successful completion of my project on time.

First and foremost I would like to thank my mentor Mr. Renil Padmanabhan, Functional Specialist, CAE-Durability, MBRDI, for constantly helping me through out with my project work. His persistence and knowledge are the two factors that have helped in the success of this project.

I also thank Mr. KeshavPrabhu, Senior Program Manager CAE-Durability, MBRDI, for guiding throughout the course of the project. It is his constant support that has helped me complete my work on time.

I specially thank Mr. DayanandaGowda, Deputy General Manager, MBRDI, for giving me a chance to carry out internship at a world-class working environment.

I would also like to thank the entire Durability team for their help and support for helping me with the assignment and the allotted project work.

I am deeply grateful to my Parents, Friends, teaching and Non-Teaching staff of the Department who have been supportive with their tolerance and genuine inspiration, in equal parts.

REFERENCES

[1] Research Analysis: A review of Front-End Modules by Matthew Beecham, 3November 2008.
[2] A survey of Front-End Modularity as an automotive architecture and its ability to deliver value by Mahe, Vincent R (Vincent Robert) ,Massachusetts Institute of Technology.2008.
[3] Current Trends in Bumper Design for Pedestrian Impact by Peter J. Schuster, California Polytechnic State University, 2006-01-0464
[4] Pedestrian Bumper Design Survey by Dr.Peter Schuster Mechanical Engineering Department, California Polytechnic State University.

APPENDIX

[A], [B] & [D] MATRICES

To find the [A], [B] & [D] matrices for six ply [0º/0º/0º/0º/0º/0º] of Carbon fiber laminate for unidirectional lamina for 2KN & 5KN load and also to find stresses and strains in each ply.

Calculations for 2oooN Load Case

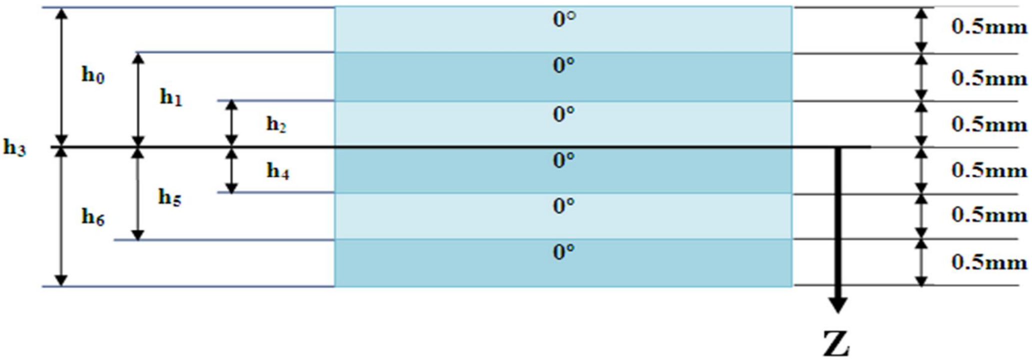


Figure 11.Thickness and co-ordinate locations of the six-ply laminate

Given

$E_1 = 19.300\,Gpa$ ⟹ Longitudinal Elastic Modulus

$E_2 = 18.525\,Gpa$ ⟹ Transverse Elastic Modulus

$G_{12} = 2293\,Gpa$ ⟹ Shear Modulus

$G_{13} = 2293\,Gpa$ ⟹ Shear Modulus

$G_{23} = 2293\,Gpa$ ⟹ Shear Modulus

Calculations

$$\frac{V_{12}}{E_1} = \frac{V_{21}}{E_2} \Rightarrow V_{21} = \frac{V_{12} \times E_2}{E_1} \Rightarrow V_{21} = 0.04799$$

To find the stiffness co-efficients [Qij]

- ❖ $Q_{11} = \frac{E_1}{1 - V_{21}V_{12}} = \frac{19.300 \times 10^9}{1 - 0.04799 \times 0.05} = 19.3464 \times 10^9\,pa$
- ❖ $Q_{12} = \frac{V_{12} \times E_2}{1 - V_{21}V_{12}} = \frac{0.05 \times 18.525 \times 10^9}{1 - 0.04799 \times 0.05} = 0.9284 \times 10^9\,pa$
- ❖ $Q_{22} = \frac{E_2}{1 - V_{21}V_{12}} = \frac{18.525 \times 10^9}{1 - 0.04799 \times 0.05} = 18.5695 \times 10^9\,pa$
- ❖ $Q_{66} = G_{12} = 2293 \times 10^9\,pa$

The transformed reduced stiffness matrix [Q] for each of the six plies is

$$\left[\overline{Q}\right]_0 = \begin{bmatrix} 19.3464 & 0.9284 & 0 \\ 0.9284 & 18.5695 & 0 \\ 0 & 0 & 2293 \end{bmatrix} [10^9]Pa$$

The total thickness of the laminate is h= [0.0005] [6] =0.003m.

The mid-plane is 0.0015m from the top and the bottom of the laminate.

$$\begin{bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} & B_{11} & B_{12} & B_{16} \\ A_{12} & A_{22} & A_{26} & B_{12} & B_{22} & B_{26} \\ A_{16} & A_{26} & A_{66} & B_{16} & B_{26} & B_{66} \\ B_{11} & B_{12} & B_{16} & D_{11} & D_{12} & D_{16} \\ B_{12} & B_{22} & B_{26} & D_{12} & D_{22} & D_{26} \\ B_{16} & B_{26} & B_{66} & D_{16} & D_{26} & D_{66} \end{bmatrix} \begin{bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \\ K_x \\ K_y \\ K_{xy} \end{bmatrix} m/m$$

$$\begin{bmatrix} 2000 \\ 2000 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} 58039200 & 2785200 & 000000 & 00000000 & 00000000 & 00000000000000 \\ 2785200 & 55708500 & 000000 & 0000000000 & 00000000 & 000000 \\ 00000000 & 0000000000 & 6879\times10^{6} & 0000000 & 0000000 & 00000 \\ 000000000 & 00000000 & 000000 & 43.5294 & 2.0889 & 00000000000 \\ 00000000 & 00000000 & 000000 & 2.0889 & 41.781375 & 0000000 \\ 00000000 & 00000000 & 00000000000 & 00000 & 00000000000 & 5159.25 \end{bmatrix} \begin{bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \\ K_x \\ K_y \\ K_{xy} \end{bmatrix} m/m$$

$$\begin{bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \\ K_x \\ K_y \\ K_{xy} \end{bmatrix} = \begin{bmatrix} 0.3282\times10^{-4} \\ 0.3426\times10^{-4} \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} m/m$$

The strains and stresses of the 0 degree are found as follows

$$\begin{bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{bmatrix}_0 = \begin{bmatrix} 0.3282\times10^{-4} \\ 0.3426\times10^{-4} \\ 0 \end{bmatrix} m/m$$

Using stress-strains Equations

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix}_0 = \begin{bmatrix} 0.9530\times10^{5} \\ 6.6666\times10^{5} \\ 0 \end{bmatrix} Pa$$