STRUCTURAL ANALYSIS OF PASSENGER BUS BODY USING FEA BY CHANGING JOINT STRUCTURE FOR IMPROVING STRENGTH

Swapnil Khatavkar[1], Sanjay R. Kumbhar[2], Vaibhav V. Wakode[3]
[1]PG student, Automobile Engineering Department, RIT Islampur, Sangli, Maharashtra, India
[2]Assistant Professor, Automobile Engineering Department, RIT Islampur, Sangli, Maharashtra,India
[3]Superintendent Coach, MSRTC Central workshop, Dapodi, Pune.
Email: khatavkarswapnil9@gmail.com[1], sanjay.kumbhar@ritindia.edu[2]

Abstract

Buses are the major mode of public transportation. Design of passenger bus body becomes highly important to achieve minimum comfort and safety levels in public transport. A bus body in operating circumstances is exposed to various types of loading conditions which causes stresses, vibrations and noise in different components of the structure. But the performance of the structure can't be compromised in such kinds of operating situations. It requires appropriate strength, stiffness and fatigue properties to withstand with these loads. To evaluate this dynamic performance of structure, it needs to be tested. In order to validate the bus body structural design one must go for physical test by making the prototype of the conceptual design. Such test will help to understand the static and dynamical behaviour of the body. However, experimental testing of real bus structures is very expensive as it requires expensive resources and space. If testing is done by using Finite Element Analysis the previous required expenses are considerably reduced. In this study, the dynamic performance of the bus body structure in terms of frequency response is observed. A comparative study of Aluminium bus body structure before and after modifications in structure is done to troubleshoot the problem of superstructure strength. The 43.5% reduction in overall stress is achieved.

Key words: passenger bus body, dynamic performance, Finite element analysis, Frequency response.

I. INTRODUCTION

A bus in any country is a kind of industries which is connected directly to the prosperity and maintains the stability of this state. After two years of completion of the security and sociality stability of the welfare states, it stopped immediately in case of any defects in the stability. The design of the internal bus skeleton structure is the basis of various bus developments in the bus industries. It contains the framework of tubes with various cross sections are arranged within specified shapes based on the design. The bus body can be divided into three parts; the chassis and engine, structural body, interior and exterior parts. They must pass the standard test by domestic and international organization. In this study, the chassis and engine are bought from the well known automotive brand such as Benz, Volvo, Isuzu, Daewoo, Hino, Tata, Ashok Leyland and so on. [1]

Our society's increasing requirements for mobility with simultaneously growing environmental sensitivity is a big challenge for the traffic policy makers and the transport corporations including private fleet operators. Consequently, it is also indispensable for the manufacturers of light and heavy passenger vehicles and the body builders to adapt to the ecologically motivated requirements, which becomes more and more important without compromising on basic minimum requirements of safety and comfort. [2]

II. PROBLEM IDENTIFICATION AND METHODOLOGY

A. Problem Identification

The bus body is subjected to various loads such as standing, cornering and bump. The current work deals with the dynamic loading condition in terms of direct frequency response determination of the stresses and deformation in the different components. Joints form the weakest link in the structure and are practically observed to be critical region of failure.

B. Methodology

1. Developing 3D model of bus body structure using CATIA V5 tool.
2. Developing the mesh model using the Hypermesh-13 tool.
3. Analyzing the structure using the NASTRAN tool.
4. Modifying critical region of failure to improve strength of the structure.

The step wise flow of methodology is shown in fig. 1 below

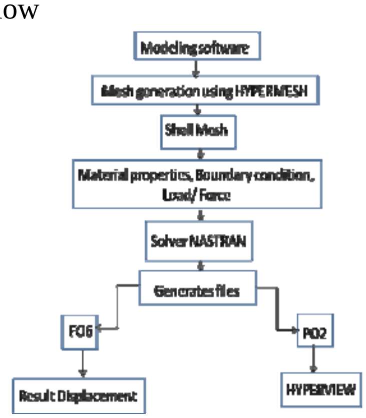

Fig. 1 Process flow chart

III. PREPARATION OF 3D MODEL OF BUS BODY STRUCTURE

The 3-D model of passenger bus body structure is prepared using CATIA V5. Table I below shows specifications of passenger bus body structure. Fig. 2 represents the 3D model used for bus body structure.

Table I Bus body specifications

Components		**Dimension (mm)**
Max Seating Capacity 44+11+2		
1	Overall width	2570
2	Overall length	10370
3	Overall height	3095
4	Wheel base	5895
5	Rear over hang	3285
6	Front over hang	1187

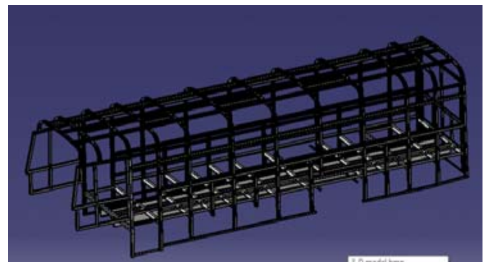

Fig. 2 3D Model of Bus Body Structure

A. Material

It is important to choose the correct material and section type to ensure that each part will be able to endure the load it will experience. The type of material to be used for chassis depends on its application requirements and operating conditions. The material is decided on the basis of its rigidity, strength, cost, durability and reliability. The material selected should also be easy to fabricate. Fabrication plays an important role in material selection. As strength of material increases, cost of fabrication increases. Table II shows that the material properties of material used for bus application

Table II Mechanical Properties of Materials Used in Bus Body Analysis [3]

Material	**Mild Steel**	**Aluminum**
Young's Modulus (GPa)	200	70
Poisson's Ratio	0.266	0.35
Density (kg/m^3)	7860	2800
Yield Strength (MPa)	250	150

IV. PREPARATION OF FINITE ELEMENT MODEL

Basic theme of FEA is to make calculations at only limited (Finite) number of points & to interpolate the results for entire domain (surface or volume). Any continuous object has infinite degrees of freedom & it's just not possible to solve the problem in this format. Finite Element Method reduces degrees of freedom from Infinite to Finite with the help of discretization i.e. meshing (nodes & elements). [4]

Here to generate the FE Model for Bus Body Hyper-Mesh v13.0 is used. Elements used for meshing is represented in Table III and Table IV represents the quality criteria used for this meshing.

Fig. 3 Mesh Model

This model shown in fig. 3 is prepared by using 2D-shell elements. Quad with four nodes and tria with three nodes at corners are used for meshing. The target size of quad element is kept to 15mm. And the quality criteria used are explained further in table IV.

Table III Mesh model details

Sr. No.	Type of element	No. of element
1	Quad4	2,89,495
2	Tria3	2,698
3	RBE-2	14
4	RBE-3	3,584
5	CBEAM	1,874
	Total	3,02,565

Table IV Quality Checks Details

Warpage angle	15^0
Aspect ratio	10.00
Skew angle	60^0
Jacobian	0.7

V. FREQUENCY RESPONSE ANALYSIS

There is no external force but the base where it is fixed itself is shaking. And the response of body is studied due to base excitation. [4] The fig. 4 below shows the input excitation to the system. The node where the structure is constrained, is shacked and displacement and equivalent stress plots are observed.

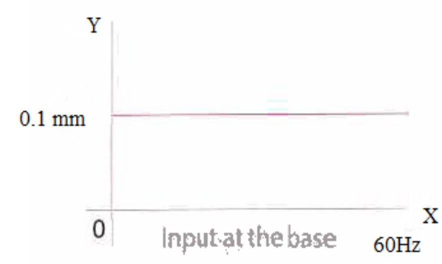

Fig. 4 Input Excitation

A. Boundary conditions

Fig. 5 below shows the boundary condition for Frequency response analysis. Here the lowest nodes of the body frame are constrained to a single central point through RBE-2. And excitation of 0.1 mm is provided at this point. The structure is allowed to vibrate with its natural frequencies. Finally this load is transferred to the body frame through this RBE-2 elements. The displacement and stresses induced in the structure due to this load are observed and discussed below.

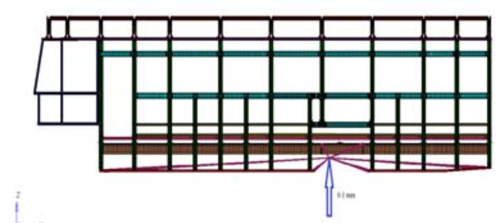

Fig. 5 Boundary conditions FRF

B. Results

The fig. 6 below shows the contour plot for displacement. From contour plot, it is observed that maximum displacement of the structure is 3.117 mm.

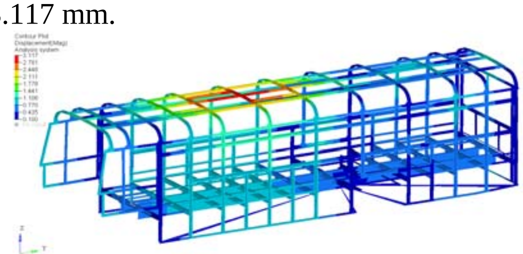

Fig. 6 Contour Plot-Displacement

The fig. 7 below shows the contour plot for equivalent stress. From contour plot, it is observed that the maximum stress induced in the structure is 63.244 MPa and the stress induced in the clits is 49.447 MPa. Corresponding natural frequency is 22.80 Hz.

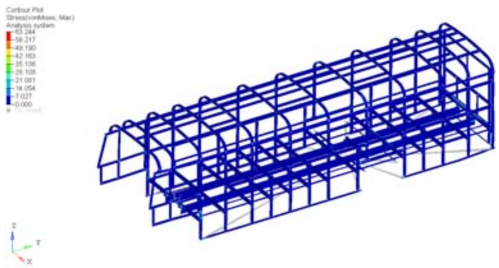

Fig. 7 Contour Plot-Stress

VI. DESIGN MODIFICATION OF JOINT

A. EXISTING JOINT (CLIT)

The fig. 8 below shows the design of clit used in MSRTC bus body structure. It is riveted to the waist rail through pop rivet of 6mm dia. and bolted to vertical body pillar through 8mm dia. bolt. Thus clit constrains the horizontal waist rail and vertical body pillar. The material used for clit is Aluminum alloy having the same material properties as that of body structure. The size of the clit is 38.1mmX38.1mmX4.76mm.

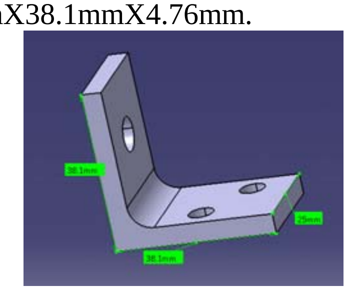

Fig. 8 Existing Clit

B. MODIFIED CLIT

The fig. 9 below shows new design of modified clit. Size of original clit is altered to 76.2mmX50.8mmX6mm. Original clit is having two rivet constrains to waist rail and only one bolting constrain to pillar. In modified clit there are same rivet constrain to waist rail as that of original clit but two bolting constrains are provided. The size of the constrains is same as that of original clit. The material used for this clit is same as that of existing clit. The whole bus body structure is modeled by using these clit and FEA analysis is carried out and results discussed below.

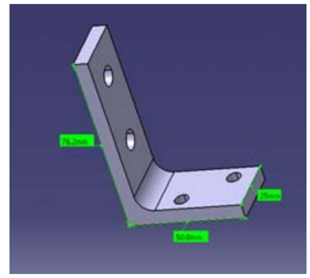

Fig. 9 Modified Clit

VII. FREQUENCY RESPONSE ANALYSIS

Frequency response analysis is carried out using same boundary condition as that of previous analysis and comparative study of result observed.

A. RESULTS

The fig. 10 below shows contour plot for displacement. The max. displacement observed is 1.603 mm.

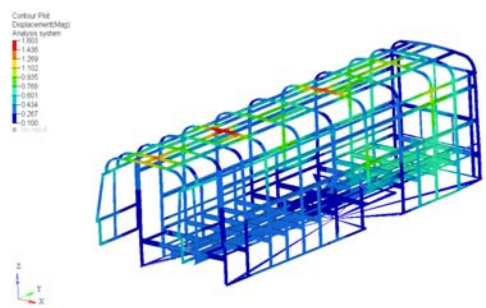

Fig. 10 Contour Plot-Displacement

The fig. 11 below shows contour plot for Stress.

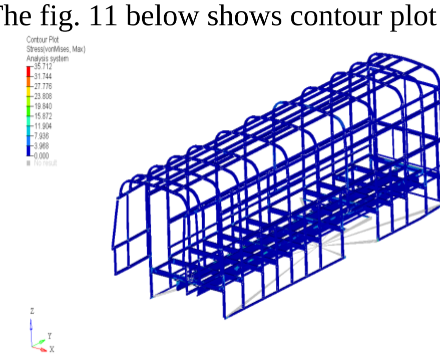

Fig. 11 Contour Plot-Stress

The max. Stress observed is 35.71MPa and that in clits is 26.025 MPa. And corresponding natural frequency observed is 26.11 Hz.

VIII. COMPARATIVE RESULTS AND DISCUSSION

The table V below shows the comparative results of Finite Element Analysis of original structure and modified structure.

Table V Comparative Results

	Original structure	**Modified structure**
Displacement (mm)	3.117	1.603
Von-mises stress (MPa)	63.244	35.712
Stress in Clits (MPa)	49.447	26.025
Natural Freq. (Hz)	22.80	26.11

From above results it is clearly seen that the displacement and overall stress in the structure after modification are reduced considerably. Here total reduction of 43.5% in overall stress is achieved. It means the structure which practically fails at 63.244 MPa stress, would not fail at 35.712 MPa stress.

CONCLUSION

In this project, a bus body structure is modeled in 3D modeling software Catia. The original body is analyzed to find out critical region of failure. Further the structure is modified and the analysis is carried out using same boundary and loading conditions as that of previous analysis. The results show good improvement in structural strength as compared with previous results.
Here total reduction of 43.5% in overall stress is achieved. Also critical natural frequency is shifted from 22.80 Hz to 26.11 Hz. Hence good improvement in fatigue strength can be achieved. Hence, It is advisable to acquire this modification in existing structure so as to improve its strength and thus service life.

ACKNOWLEDGMENT

I am obliged to staff members of Rajarambapu Institute of Technology as well as Maharashtra State Road Transport Corporation for the valuable information provided by them in their respective fields. I am grateful for their cooperation during the period of my dissertation.

REFERENCES

[1] P. Chinta, and L.V. Venugopal Rao, "A New Design and Analysis of Bus Body Structure," IOSR Journal of Mechanical and Civil Engineering, e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 11, Issue 5 Vol I, pp39-47, 2014.

[2] Automotive Industry Standard-052, "Code of Practice for Bus Body Design and Approval," Sept 2001.

[3] IS 733-1983 "Specification for wrought aluminum and aluminum alloy bars, rods and Sections," Nov 1983.

[4] Nitin Gokhale, "Practical Finite Element Analysis," pp-8,228,231, 2007.

[5] V. Ramaswamy, R. Pareek, A. Giri, and V. Srivastava, J. Nene, "Aluminum Bus Structure Joint Design Development", Simulation Driven Innovation, HTC, 2012.