International Journal of Engineering and Technical Research (IJETR)
ISSN: 2321-0869, Volume-2, Issue-12, December 2014

Finite Element Analysis and Optimization of Commercial Bus Body Structure

Mahesh Haldankar, A. M. Shirahatti

***Abstract*— Road transport is the most popular transport which mostly uses buses. Local and medium enterprises take the bus body building work. The smaller companies do not have much funding towards the FEA to develop the design. To remain competitive in the market, the continuous improvement of the design is needed and it is easier to do in finite element analysis. During the normal operation, the bus body is subjected to several loads, external loads from the road (i.e. crossing over a speed bump). Moreover, there is a substantial possibility that these loads may lead to a structural failure. Hence, it is necessary to determine stresses occurred in the bus body to ensure its integrity under these driving scenarios. This project deals with the modelling, analyzing and optimization of important section of the bus body for the standing gravity load and for the bump case and it is further tried to validate the results with the help of theoretical calculations.**

***Index Terms*—Structural optimization, bus body, gravity load, stress, bump case, weight reduction.**

I. INTRODUCTION

As there have been no legal requirements on bus body building in India, there are plethora of designs and variants of the same design in the country. This makes it difficult both for the manufacturers as well as the testing agencies to decides the family concepts and arrive at the worst case. For design safety assurance, design experts traditionally attempt to maintain all required structural parts and elements, which raise the production cost and the product price. Moreover, it increases the weight and the fuel consumption of the bus [1].

The static load response of simple structures, such as uniform beams, plates and cylindrical shells, may be obtained by solving their equations of motion. Practical structures consist of an assemblage of components of different types, namely beams, plates, shells and solids. In these situations it is impossible to obtain analytical solutions to the equations of motion. This difficulty is overcome by seeking some form of numerical solutions and finite element methods.

II. DIFFERENT TYPES OF BUS STRUCTURE

A. Single Decker

A single-decker bus or single-decker is a bus that has a single deck for passengers. Normally the use of the term single-decker refers to a standard two-axled rigid bus, in direct contrast to the use of the term double-decker bus, which is essentially a bus with two passenger's decks and a staircase.

B. Double Decker

A double-decker bus is a bus that has two storey's or decks. Double-decker buses are used for mass transit Almost all double-deckers have a single, rigid chassis [1], [2].

C. Limobus

A limobus (also known as a limousine bus or luxury bus) is a large motor vehicle usually based on a conventional bus or coach body style, augmented with a luxury interior. A limobus' passenger capacity is dictated by its size, with smaller buses able to carry up to 8 passengers, and larger buses being able to accommodate over 30.

D. Different parts of bus structure

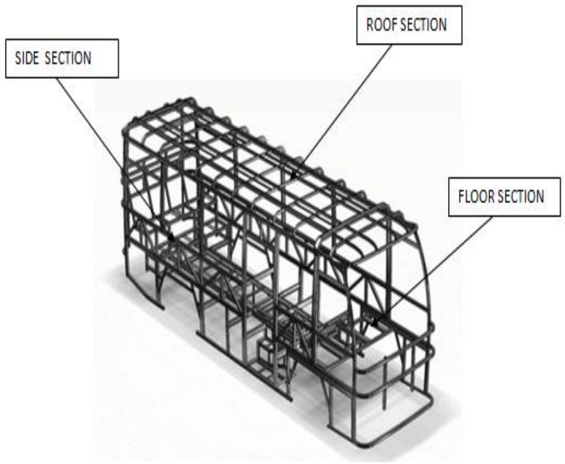

Fig. 1. Different section of bus

The bus body structure consists of six major parts. The front part supports the windshield, the console, main front lights, and the top part. The back part supports the rear window and the top part. The left part supports the front door, a middle door or/and a back door, windows, the left side panel made of sheet metal with primer coats and paints, and the top part. The right part supports the driver door, an emergency door, windows, the right side panel, and the top part. The top part supports both fixed and varying loads. Fixed loads consist of an air conditioner and stereo system and varying loads consist of passengers and luggage [2].

III. METHODOLOGY AND OBJECTIVE

A. Problem definition

The bus body is subjected to various loads such as standing

Manuscript received December 20, 2014.

Mahesh Haldankar, Department of Mechanical Engineering, Maratha Mandal Engineering College, Belgaum, India, Phone No. 0831-2925005,
Dr. A. M. Shirahatti, Department of Mechanical Engineering, Maratha Mandal Engineering College, Belgaum, India, Phone No. 0831-2925005,

and bump case. The current project work deals with the static load condition and for the floor section for 2.5g case when vehicle moves on a plain road surface and when the vehicle passes across the bump and also determining the stresses and deformation in the different cases with different length. Further in this project optimization is done by reducing the thickness.

B. *Objective*

To analyze and optimize the bus body for standing load and bump case and optimize the floor section.

C. *Methodology*

1. Determine the critical point having highest stress and modify the design of bus body to get an optimized in terms of reducing weight and reducing stresses.
2. Developing the mesh model using the Hypermesh-11 tool.
3. Checking the quality criteria of mesh model such as aspect ratio, Jacobian, skew-ness , warpage etc.
4. Analysis is carried out using the Nastran tool and proper cards are specified.
5. Static analysis is carried out on all the front side and bottom parts and to determine the stresses and deformation.
6. Optimization is carried out to reduce the thickness and determine whether the part is safe or not.

IV. HYPERMESH AND NASTRAN

A. *Problem definition*

Hyper mesh is a finite element modeler used to perform a variety of CAD/CAE tasks including modelling meshing, and post processing for FEM solvers LSDYNA, NASTRAN,ABAQUS Etc. Hyper mesh can be directly used to access the geometric models from leading design software's like CATIA, Pro-E etc. Hyper mesh helps to overcome the FEM challenges, including many topological irregularities, multi body contacts etc.

Multi-disciplinary analysis can be possible by using the open integrated CAE environment provided in Hyper mesh. This feature can be used to simulate manufacturing process and behaviour of the product in early stages of design-to-manufacture process. This has the ability to import geometry from any CAD system and various data exchange standards. Hyper mesh provided with rich set of tools made it possible to achieve a required mesh quality with the different meshing. The various visualization tools helps to find much critical information, including maximum and minimum values, contour plots which shows trends and correlations. It is also possible export result images and animation videos in many standard forms which help to present in reports [3]

B. *Problem definition*

Research laboratories and many other industries uses MSC-NASTRAN software for analysis. MSC-NASTRAN library contains more than 50 types of elements which include one, two and three dimensional elements, Scalar elements, mass element sand heat transfer elements

V. THEORETICAL CALCULATIONS

A. *Material specification*

The material assigned to the entire load body is mild steel and the grade is ST52 which is quite popularly in use in the automotive sector. Table I gives the material properties used for current analysis [4].

Table I. Material properties

Young's modulus	210 GPa
Density	7850 kg/m^3
Yield strength	355 MPa

B. *Calculation for existing C section*

Figure 2 shows the dimensions required for calculation and the loading condition.

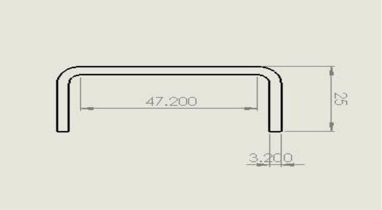

Fig. 2. C section

The C section is made up of mild steel grade ST52 and has available in two lengths i.e. 670 mm and 402 mm. First, moment of inertia is found for the section and then value of stress is calculated as tabulated in table II.

Table II. Details of C section calculation

Thickness (mm)	Y (mm)	I (mm^4)
2	18.08	1.166 x 10^4
2.5	17.96	1.425 x 10^4
3.2	17.79	1.769 x 10^4

C. *Standing gravity load validation in Nastron*

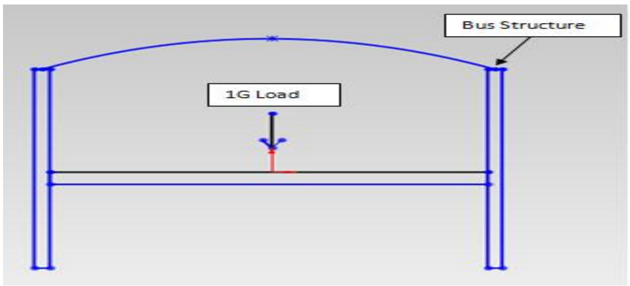

Fig. 3. Depicts the loading condition

During standing gravity load, F = mg
Where F = Force.
m = Total mass of the pay load.
g = Acceleration due to gravity.

In general two passengers sit on the seat, it is assumed each passenger weighs 70 kg and seat weighs 16 kg (information is provided by company)

The total mass of the pay load (156 kg) is carried by two C section and it is uniformly distributed.

Therefore,

$$F = 78 \times 9.81 = 765.18\ \text{N}$$

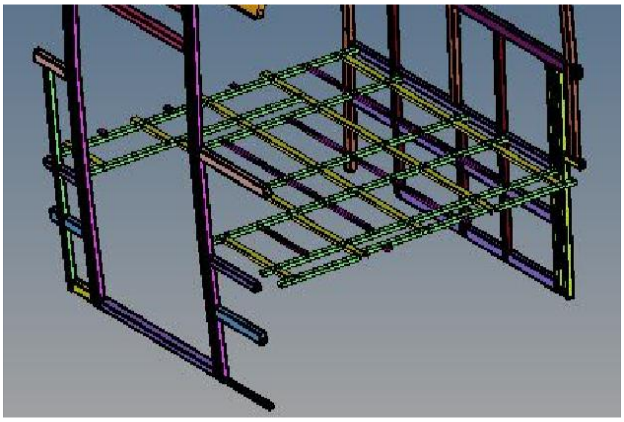

Fig. 4. Standing gravity loading condition

D. *For analysis (1g) for 2mm thickness*

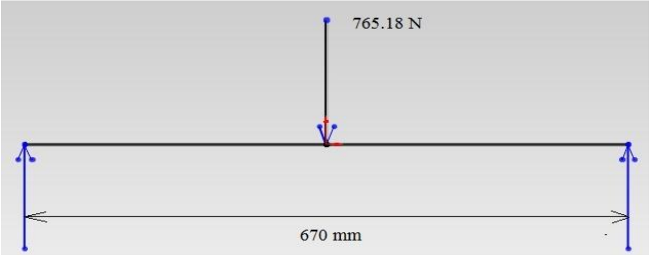

Fig. 5. C section loading (1g) for 2mm

For uniformly distributed load

$$p = \frac{765}{670} = 1.142\ N/mm$$

The maximum Displacement for the uniformly distributed load is.

$$W_{max} = \frac{5}{384}\frac{PL^4}{EI} = 1.2236\ mm$$

$$M_{max} = \frac{PL^4}{8} = 6.41\ x\ 10^4\ N - mm$$

The Max stress will be,

$$\sigma_{max} = \frac{M_{max}\ y}{F} = 99.37\ MPa$$

E. *Meshed model in HYPERMESH*

The C section is affected during static case and the displacement occurs at the C section and the deformation is around 1.32 mm as shown in Fig 5.1 for standing gravity (1g) case [5].

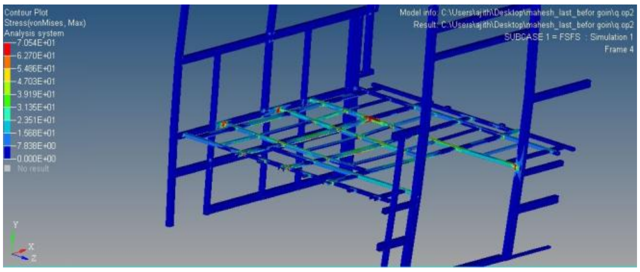

Fig. 5.1. Stresses in the section for 1g case

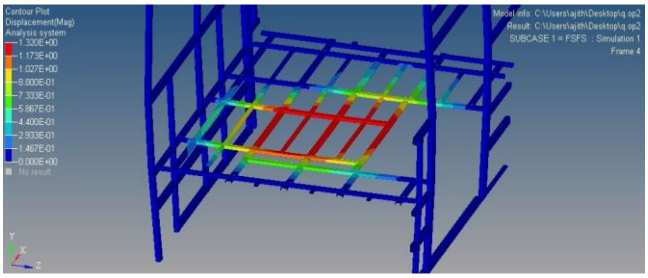

Fig. 5.2 Displacement in the section for 1g case

F. *Optimized results of C section*

Von misses stresses and displacement for 3 mm thickness The thickness of the C section is reduced from 3.2 mm to 2.5 mm. therefore The Fig 5.3 shows stresses generated in the front section due to static case which is 76.87 MPa.

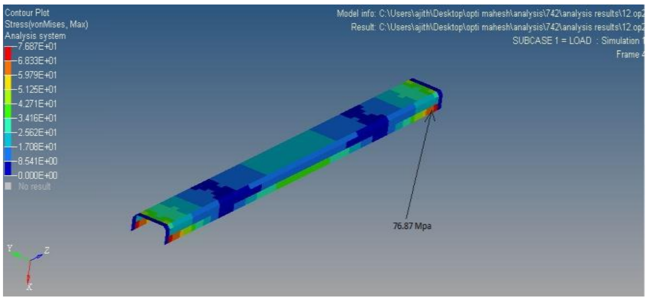

Fig.5.3 Stresses in the front section

The C section is affected during static or 1g case and the displacement occurs is around 1.04 mm as shown in Fig 5.4.

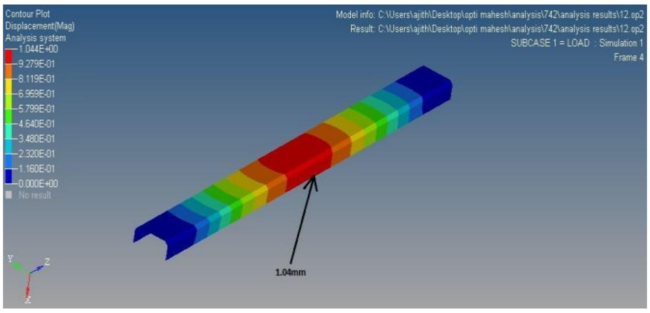

Fig.5.4 Displacement in the front section.

VI. RESULTS AND DISCUSSION

The table III gives the comparisons of the von misses stress and the displacements with the thickness and the trend that we see is that the von misses stresses increase with the decrease in the thickness. The displacements also follow a similar trend t.

Table. III Comparison of FEA and theoretical displacement of C Section

Factors	Theoretical	FEA	% Error
Stress (MPa)	149.6	153.9	4.3
Displacement (mm)	0.66	0.64	2

For (1G) case with length 670mm

The values of stresses vary from the 64 to 100 MPa and the displacements vary from 0.80 mm to 1.24 mm. Stress increases with decrease in thickness and same is the case with the displacements.

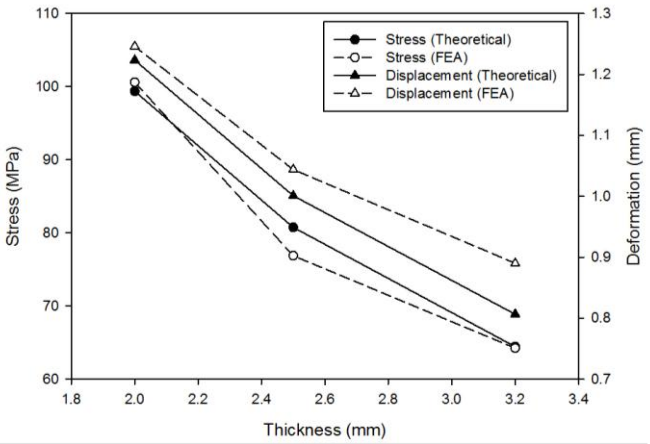

Fig. 6. Variation of stress and displacement with thickness

VII. CONCLUSION

The present work illustrates the numerical simulation of bus body section for static and bump case with different length as well as von misses stresses and deformation of each case is determined. The simulation results of stress and deformation verses thickness is in good correlation with that of the theoretical results.

The present study involves the study of the static and bump case study. In the static case study, the deformation of 0.8901 mm and stresses of 64.24 MPa is obtained for 3.2 mm thickness and after optimization the stress went up to 100 MPa and deformation of around 1.246 mm is observed.

In the bump case study, the deformation and stress of 2.106 mm and 170.5 MPa respectively was obtained for 3.2 mm thickness and after optimization the stress went up to 244.72 MPa and deformation to 2.958 mm. In both studies stresses and displacement were well below the safety level specified by the company.

Lastly the project work conclude that C section can be changed with rectangular section and the thickness can also be varied so that it can withstand the load and also can provide stiffness.

VIII. FUTURE SCOPE

In the current study the thickness optimization was carried out for the bus body structure after selecting certain major parts and identifying the load distribution.

In future, optimization can be carried out by changing the material properties and considering more vehicle parts/structure. Also, in the analysis fatigue behaviour of the structure can be included.

As the public transport is the major source of transport for majority of people in some part of the country. Many times it is found that public transport are overcrowded and carry more passengers than its specified capacity. Hence, overloading analysis can be done from safety point of view.

ACKNOWLEDGMENT

The authors are gratefully acknowledge Mr. Dilip Desai, General Manager Design Department, Automobile Co-operation of Goa Limited, Goa for providing technical data. The authors would also like to thank Mr. A. C. Mattikalli for his valuable suggestions.

REFERENCES

[1] S. Butdee, and F. Vignat “TRIZ method for light weight bus body structure design” Presented in International Journal in Achievements in Materials and Manufacturing Engineering, Volume 31, Issue 2, December 2008.

[2] Osvaldo Ruíz, Edgar I. Ramírez, Víctor. H. Jacobo, Rafael Schouwenaars, Armando Ortiz. “Efficient optimisation of the structure of a passenger bus by iterative finite element models with increasing degrees of complexity” 3rd International Conference on Engineering Optimization Rio de Janeiro, Brazil, 01 - 05 July 2012.

[3] Croccolo, D. – De Agostinis, M. – Vincenzi, N. and Dario Croccolo – Massimiliano De Agostinis – Nicolo Vincenzi “Structural Analysis of an Articulated Urban Bus Chassis via FEM: a Methodology Applied to a Case Study” International Journal of Mechanical Engineering 57(2011)11, 799-809, Itley.

[4] Manokruang S. and Butdee S.”Methodology of Bus-Body Structural Redesign for Lightweight Productivity Improvement” International journal published in *AIJSTPME (2009) 2(2): 79-87.*

[5] S. Butdee and S Manokrung “Knowledge capitalization to Bus body Light weight redesign and validated by FEM” Proceedings of IDMME – Virtual concept 2010 Bordeaux, France, October 20-22, 2010.

Mahesh Haldankar received his B.E degree in Mechanical Engineering from VTU, Belgaum. He is currently pursuing his M.Tech degree in Machine Design from Maratha Mandal Engineering College, Belgaum. His area of research interest include design and FEA.

Dr. A. M. Shirahatti has completed his Ph.D. in Mechanical Engineering from University of Bristol, United Kingdom. He has in total a decade of experience in teaching, software development and R & D. His area of interest include structural integrity, residual stress, vibration and composites. In his credit, there are 15 international and national publications.