BUS BODY STRUCTURAL STRENGTH ANALYSIS THROUGH FEA

Guruprasad T[1], Dr. Satish Babu[2], Dr. Maruthi B H[3], Pramod K N[4], Manjunatha H G[5]
[1,4,5]Student, [2]Asso. Prof., [3]Prof. and Head, Dept. of Mechanical Engineering
East West Institute of Technology, Bengaluru, India.

Abstract: Buses are the foremost mode of road transportation. The design of the bus body depends mainly leading the performance constraint under various types of loading and operating circumstances besides those of the road conditions. the model analysis, linear static analysis and impact analysis of an articulated urban bus body, carried out with the Finite Elements Method. The purpose of this work is to simulate and forecast the structural response of the bus body in terms of stress, strain and displacement, under several loading and constraining conditions Sensitivity analyses about FEM parameters have been run, in order to achieve an adequate trade-off between computational time and results accuracy This project deals with the GFEM modeling, analyzing of important section of the bus body for the standing gravity load, acceleration, breaking load and for the impact case.

Keywords: GFEM Model, model analysis, linear static analysis, impact analysis

I. INTRODUCTION

The bus body structure must be balanced in order to obtain the safety when the bus is running body must be sufficiently strong both the situation of supporting normal loads and accident loads. The bus body can be divided into three parts; the chassis and engine, structural body, interior and exterior parts. The chassis and engine are quite important. They must pass the standard test by domestic and international organization. The chassis consists of frame, which is a box type section and varies longitudinally as per the load and strength required for Body. Numerous Stiffeners are also added at the locations where the effect of Bending is Maximum. The body comprises of six main components; the left frame side, the right frame side, the front frame side, the back frame side, the top frame side and the bottom frame side. The top frame side is sometime called “the roof frame side”. The bottom frame side is also called “the floor frame side”. The left and the right side are similar but the left side is normally composed of passenger door(s). On the other hand, the right side has two doors; the driver door and the emergency door. The sides are concerned to be critical parts and they must be strong. 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. The bus body manufacturing composes of several operation processes. In general, the first step is to prepare drawings after the design is already completely finished. Then, the production process is planned for how to build the bus body step by step, which machine and cutting tools are selected, how much materials are needed, how long time does it take and how much does it cost. Next, the chassis is selected and prepared. Normally, the chassis is combined with its engine.

A. Bus body design parameters:

The bus body design parameters consist of strength, light weight, manufacturability, adaptability, weld ability. Technical contradictions, the possible contradiction among the parameters have been identified. To accomplish this, the fact that improvement of one parameter can worsen another one has been taken into account.

- Weight of the moving object,
- Length of the moving object,
- Area of the moving object,
- Column of the moving object,
- Durability of the moving object
- Stability object, Strength

II. PROBLEM DESCRIPTION

Now days there is demand on buses, not only on the cost, weight and shape aspects but also on the improved entire vehicle features and overall work performance. In addition to this number of variants that are possible due to different types of designs and modulation, all for several design iterations to arrive at appropriate combination. For optimized bus body design, newly developed models are chosen whose specifications are taken from the local industry

A. Objective

The main objective of the work is

- To analyze different mode shape of the bus body structure by model analysis
- The project work concerned about the linear static analysis of bus body
- To analyze the bus body based on impact load case

B. Methodology

- Geometric Modeling: The three Dimensional created using CATIA or PRO-E
- Finite Element Analysis: The three Dimensional created using CATIA or PRO-E is imported to FE software ANSYS and is meshed and that model is called as Finite elemental model
- Suitable boundary condition: The meshed model is subjected to certain boundary condition and

sensitive analysis is done using Ansys14.5

C. Finite Element Approach

- An analysis of measured data is a process in which the measured frequency response functions are analyzed in order to find a theoretical model that most closely resembles the dynamic behavior of the structure under test. This part of the modal test is called experimental modal analysis, although this term is often incorrectly used for the entire modal test. The process of data analysis proceeds in two stages:
- Identifying the appropriate type of model (with viscous or structural damping). This choice is often in practice limited by software used for the modal analysis. Most of software packages work with one type of damping and give no choice to the user.
- Determining appropriate parameters of the chosen model. This stage, also called modal parameters extraction, is done by curve-fitting of the measured frequency response functions to the theoretical expressions.

D. Design parameter details

The parameters which considered are the dimensions of actual bus given below.

TABEL I: SPECIFICATION PARAMETER OF BUS ARE IN m

Specification parameter	Dimension (m)
Length	11.66
Width	2.47
Height	3.24
Wheel base	2.16

III. MODELLING AND SIMULATION OF BUS BODY

Bus body structure modeling process was carried out in CATIA V5. This chapter discusses the detailed three dimensional modeling of a bus body and simulation of the model. Before measuring of the structure, drawing of the bus structure is made. This helps for easy placement of the measured length on the corresponding members on the drawing. The bus structure is made with steel beams of rectangular hollow section with different size

A. Modeling

The geometric under consideration are generated in the CATIA V5 Modeling package. It is an authoritative program used to create complex designs with great precision. It has properties like Feature-based nature, Bidirectional associative property and parametric in nature. When designing a model using Solid Works, you can visualize it in three dimensions, the way the model exists once it is manufactured.

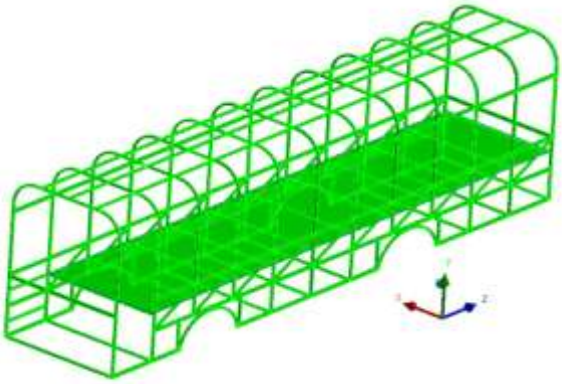

Fig.1 CATIA Model of Bus Body

B. Meshing

Finite element meshing is made with ANSYS 14.5 workbench. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time it takes to create mesh model is often a significant portion of the time it takes to acquire results from a CAE solution. Tetrahedral and quadrilateral mesh elements are used while meshing of the bus structure. In ANSYS 14.5 Workbench, Tetra/quad mesh method provides

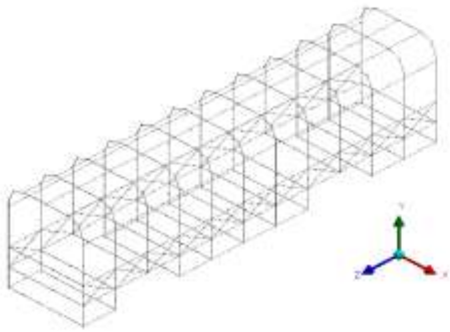

Fig.2 Mesh Model of Bus Body

C. Boundary and loading condition

The boundary condition used in the analysis is different according to the operating circumstances of the bus. During the static loading case the main loads that are considered are acceleration 60km/h, breaking load, 2g load and impact load

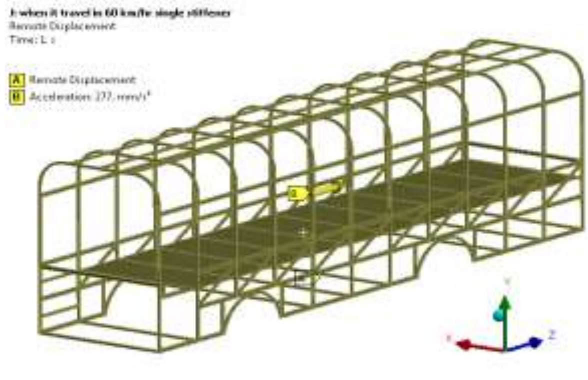

Fig.3 Analysis of structure when travel in 60km/hr:

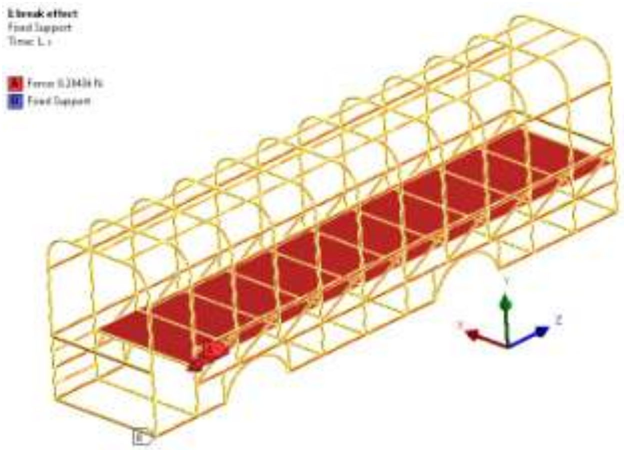

Fig.4. Analysis of structure for Sudden breaking effect

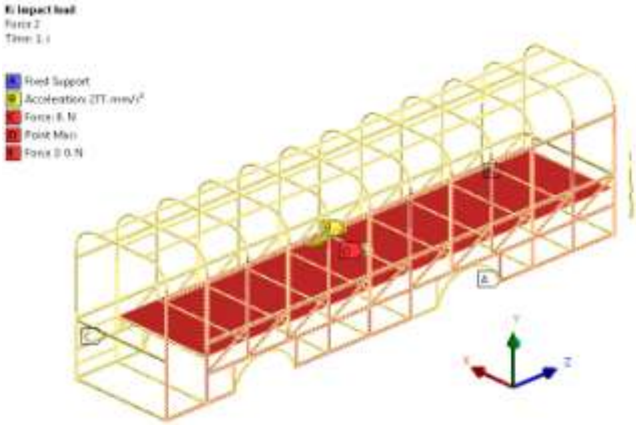

Fig. 5 Analysis of structure for Impact load condition

D. Post-processing

Post-processor contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution.

TABEL.II: MATERIAL PROPERTIES FOR STRUCTURAL STEEL

Material : structural steel	
Density (kg /m^3)	7860
Young's Modulus E (Mpa)	210
Poisson Ratio	0.3

IV. RESULTS AND DISCUSSIONS

A. Modal Analysis

Used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available.

TABEL.III MODEL ANALYSIS FOR SIX FREE-FREE MODES

Mode	Frequency (Hz)
1	0
2	0
3	2.5095e-003
4	4.1766e-003
5	5.7434e-003
6	9.8533e-003

Mode 1

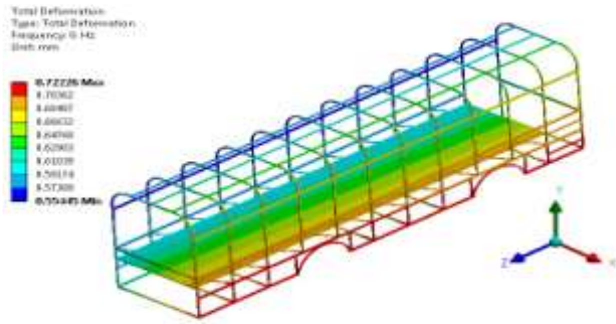

Fig.6. Zero frequency at Y translation

Mode 2

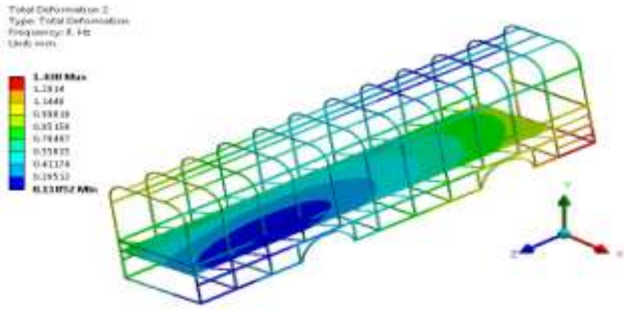

Fig .7 Zero frequency at Z rotation

Mode 3

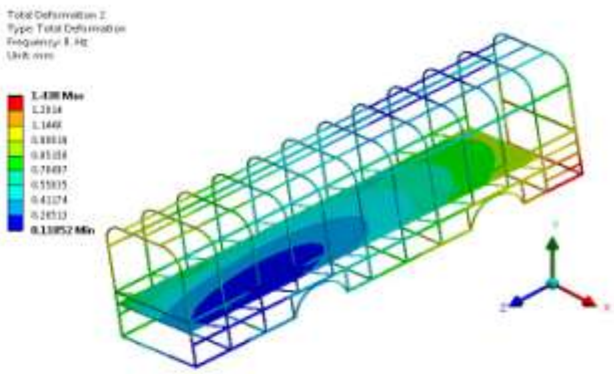

Fig.8. Directions at Z translation

Mode 4

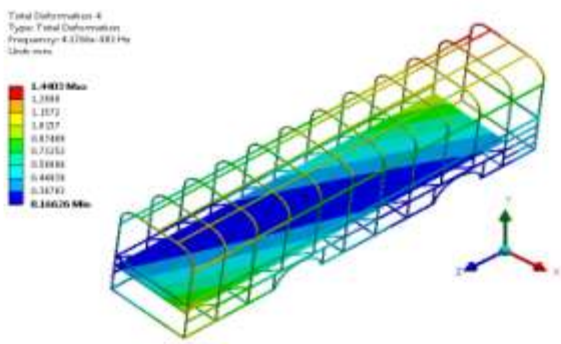

Fig.9. Directions at X rotation

Mode 5

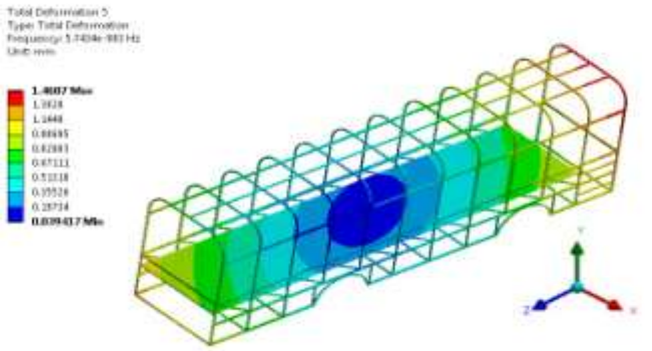

Fig.10. Directions at Y rotation

Mode 6

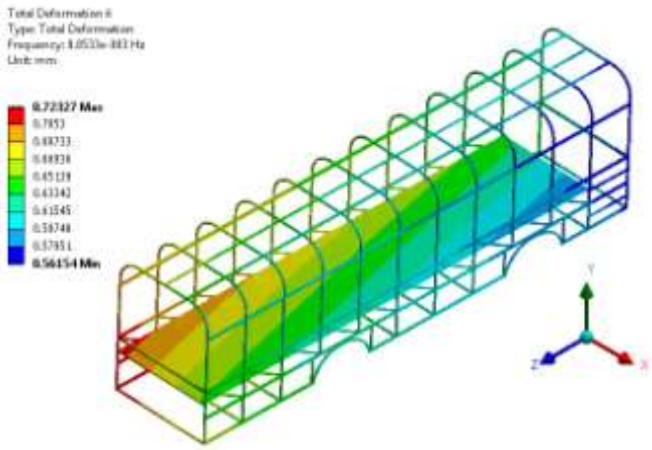

Fig.11 Directions at X translation

B. Static Structural Analysis

Used to determine displacements, stresses, etc. under static loading conditions, both linear and nonlinear static analyses. Non-linearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

Analysis of structure when travel in 60km/hr

1) Total deformation:

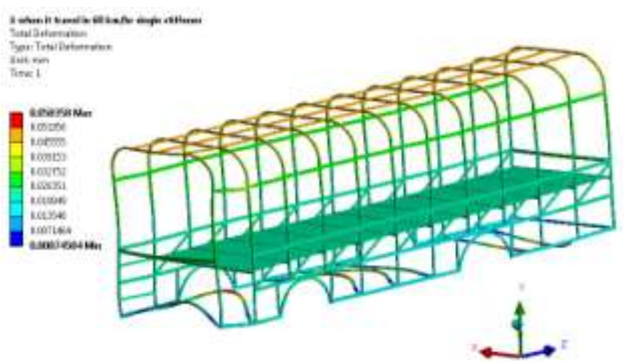

Fig 12 Displacement in the body when travel in 60km/h

2). *Equivalent stress:*

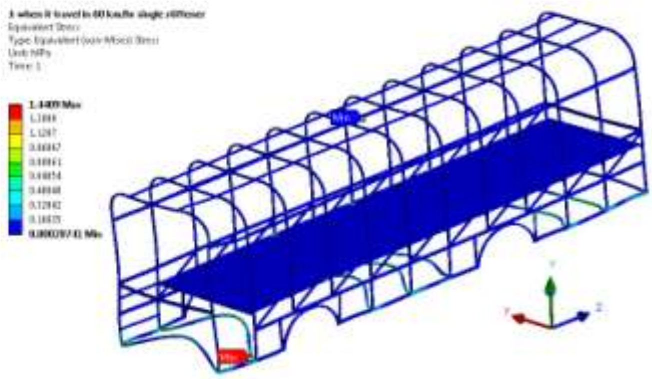

Fig.13 VonMises stress in body when travel in 60km/h

b). Analysis of structure when sudden breaking effect: 3g load act in the structure

1). *Total deformation:*

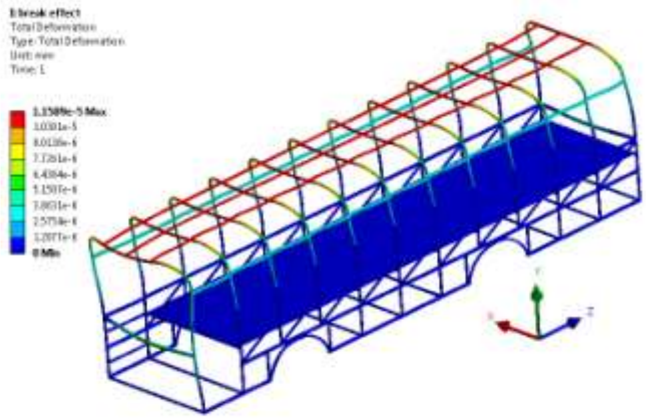

Fig. 14. Displacement in the body for breaking load effect

Equivalent stress:

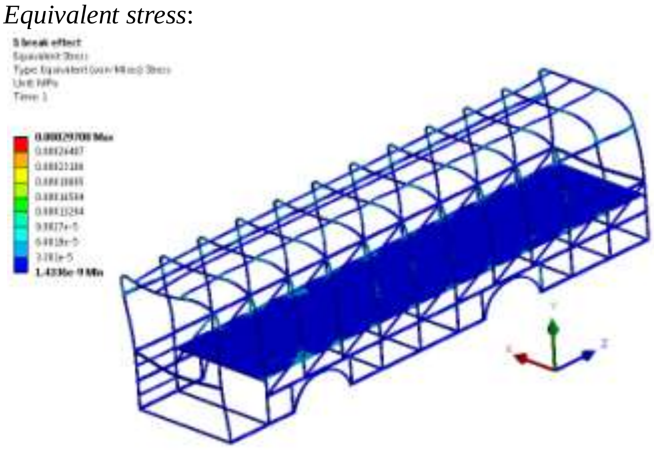

Fig.15 VonMises stress in body for breaking load effect

C. Impact analysis

Analysis of structure when impact load effect

1). *Total deformation*

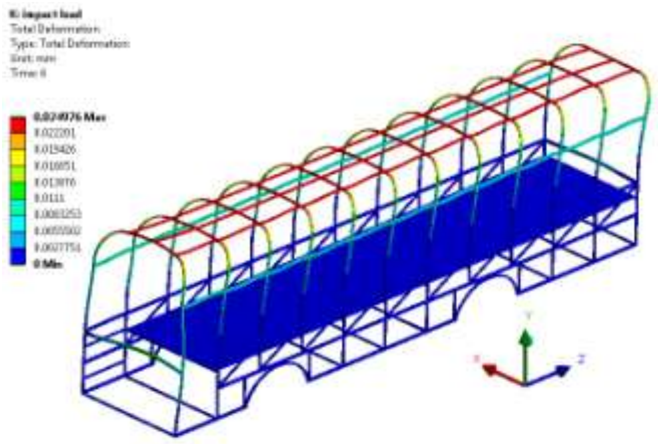

Fig.16 Displacement in the body for impact load case

2). Equivalent stress:

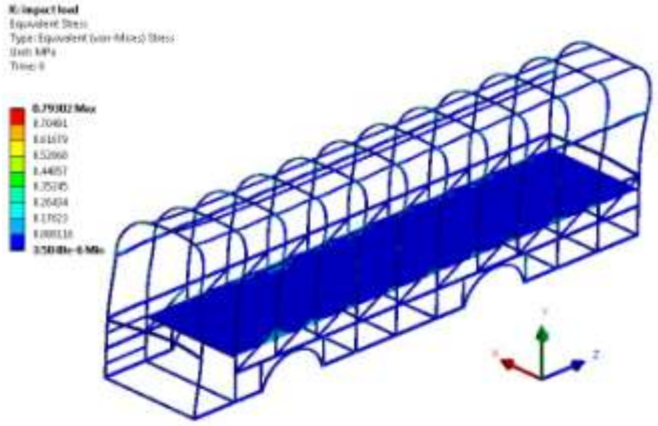

Fig. 17 VonMises stress in body for impact load case

V. CONCLUSION

The free free model analysis and static structural analysis of an articulated urban bus body, with a total length of 11.66 m, has been performed via Global Finite Elements Method. The structure behavior towards four different loading conditions, representative of its typical duty cycle, has been analyzed: the action of gravitational acceleration, the braking at the deceleration limit of the vehicle and impact load condition to obtained stress, strain and displacement. Sensitivity analyses in order to evaluate the bus body performances have been carried out in order to obtain reliable results in terms of stiffness and displacements of the bus body.

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] Manokruang S. and Butdee S. “Methodology of Bus- Body Structural Redesign for Lightweight Production improvement” International journal published in AIJSTPME(2009) 2(2):79-87.

[3] Feng, G. “Finite element analysis of bus body structure.”Mech.Engng, 1999, 35(1), 91-95.

[4] Kim, T. “Study on the stiffness improvement of bus structure.” SAE paper 931995, 1993

[5] Takamastu, M. “Development of lighter-weight, higher-stiffness body for new RX-7. SAE paper 920244, 1992.

