# Finite Element based Static Analysis of a Bus Superstructure

Lakhwinder Singh
M.E Student
Mechanical Engg. Department
UIET, Panjab University
Chandigarh, India

Sanjay Vohra
Associate Professor
Mechanical Engg. Department
UIET, Panjab University
Chandigarh, India

Manu Sharma
Associate Professor
Mechanical Engg. Department
UIET, Panjab University
Chandigarh, India

## ABSTRACT
This paper describes the finite element analysis of a bus superstructure subjected to static loading. Safety of a bus structure and its occupants during rollover accident is a major concern for the designer. Rollover safety of a bus depends upon both the weight and strength of the superstructure in such a way that weight should be less along with a high strength. In this work static analysis of the FE model of a bus superstructure is performed in ANSYS 16.2 (ANSYS) by applying a static load at the top edge of a side wall with a constraint of fixed bottom edge. Two analyses are performed for the same loading and boundary conditions. In the first analysis, the material used for bus model is structural steel, while the second analysis is done by replacing some members of floor segment by the carbon epoxy composite members and comparison of results is performed. The superstructure with composite members has given a weight saving of around 13% as compared to pure steel model.

## Keywords
Superstructure, rollover, finite element, survival space, simulation, epoxy, shear, transverse.

## 1. INTRODUCTION
Buses are the most convenient mode of transportation. Safety of a bus structure and its occupants in an event of accident is a major concern for the designer. Rollover accident is very dangerous for the vehicles and thus the vehicle design should meet the safety requirements of a rollover event [1]. In Europe ECE R66 and in India AIS-031 are the standards used to ensure the strength requirements of a vehicle for rollover safety. According to these standards, the vehicle during rollover is considered safe if the survival space remains undeformed [2]. It is found from the literature that with increase in the mass, the survival space of bus superstructure gets harmed and with strength increase it remains unharmed which ensures the safety of the superstructure [3]. Thus for the rollover safety, the vehicle should have less weight and high strength.

For the testing of rollover of a vehicle, finite element programs can be used for the simulation in accordance to ECE R66 and AIS-031 [4]. Equivalent body sections of a bus superstructure can also be used for the simulation of rollover [5]. Use of composite materials in different sections of bus body like side panel [6], frame [7], roof door [8] and the floor segment [9] results into an excellent saving of weight. For the vehicle with less mass, the survival space is found to remain safe during rollover [10]. In this work, instead of simulating the rollover of whole FE model of bus, a static load is applied on the top edge of the side wall of the bus with the bottom edge fixed. Two studies have been carried out for the analysis. In first, the analysis of FE model of structural steel is carried out and resulting deformation and stresses are obtained. In the second one, the members of first FE model of structural steel having high stresses are replaced with the carbon epoxy composite and then static analysis is performed. Finite element modelling and static analysis of the bus superstructure are discussed in the next sections.

## 2. FINITE ELEMENT MODELLING
Finite element (FEM) analysis of the bus superstructure has been performed in ANSYS. The geometric model prepared by using CAD is imported into ANSYS and meshing is done by using mesh generator. In the first analysis the material assigned to model is structural steel while in the second analysis carbon epoxy composite material is used along with the steel.

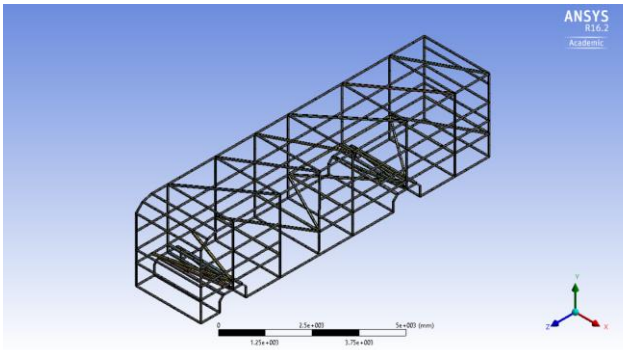


**Figure 1. Meshed FE model of bus superstructure**

Mass of the FE model of bus superstructure with steel as material is 1691.8 kg. Properties of structural used in modelling are: Density of structural steel is 7850 kg/m$^3$, elastic modulus 200 GPa and Poisson's ratio of 0.30. Meshing is done by using 646 elements and 1190 nodes. Element used is beam element with hollow rectangular cross section. Length, width and height of the FE model of bus are 10854 mm, 2440 mm and 2700 mm respectively.

## 3. STATIC ANALYSIS OF FE MODEL
### 3.1 Analysis with Structural Steel model
First the analysis of FE model of bus with structural steel as material is performed. A static load of 90000 N is applied at the top edge of the side wall of the bus with the bottom edge fixed as shown in figure 2.

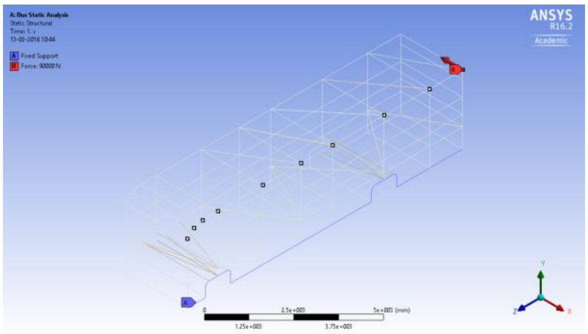


**Figure 2. Boundary and loading conditions**

The solution is obtained in the form of total deformation and the stresses resulting due to the static loading. The maximum deformation is found to be 363.31 mm as shown below in figure 3.

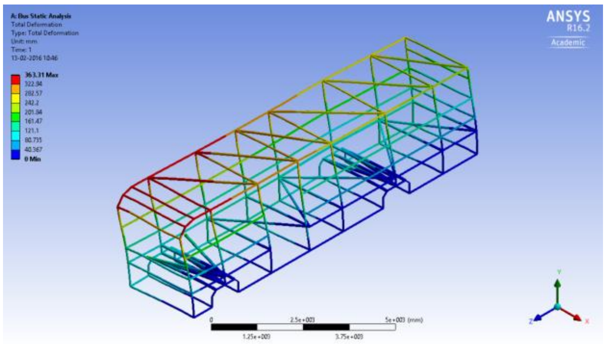


**Figure 3. Total deformation of the superstructure**

The maximum value of direct stress due is 59.814 MPa and maximum combined stress obtained is 1014.9 MPa.

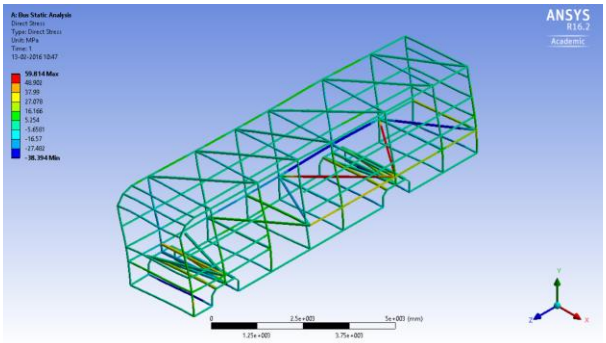


**Figure 4. Direct stress distribution due to applied load**

Now for optimum design of a bus superstructure, it should have less weight along with the required strength.

To reduce the weight, use of composite materials is the best solution as they have low density and high strength. In the second analysis, some members of floor segment which are highly stressed as found from above results are replaced by the carbon epoxy composite members. The minimum values of direct and combined stresses are -38.394 MPa and -24.215 MPa respectively.

## 3.2 Analysis of model with carbon epoxy composite

From the analysis performed on structural steel model, the elements bearing maximum stress are identified. In the second analysis, these elements of the floor section are replaced with the composite ones using the unidirectional carbon epoxy material. The Properties of unidirectional carbon epoxy composite material are given below [2].

**Table 1. Properties of carbon epoxy composite material**

| Property | Symbol | Value |
|---|---|---|
| Density | $\rho$ | 1.6 g/cm$^3$ |
| Elastic modulus (fiber direction) | $E_1$ | 141 GPa |
| Elastic modulus (transverse direction) | $E_2$ | 8.7 GPa |
| Shear modulus | $G_{12}$ | 5.6 GPa |
| Poisson ratio | $\nu_{12}$ | 0.30 |
| Tensile strength longitudinal | $X_T$ | 1925 MPa |
| Tensile strength (transverse) | $Y_T$ | 76 MPa |
| Compressive strength longitudinal | $X_C$ | 1725 MPa |
| Compressive strength (transverse) | $Y_C$ | 228 MPa |
| Shear strength | $S_{12}$ | 72 MPa |

The tensile and compressive stresses of carbon epoxy composite in longitudinal direction are very high. Also it has a very low density as compared to conventional steel which is the factor to achieve weight saving although cost of the carbon epoxy is very high as compared to steel and aluminium [11]. By replacing the floor members of the FE model of bus superstructure with composite carbon epoxy material and subjected to same loading conditions, the resulting deformation is slightly higher than the structure with steel. The maximum deformation is found 366.46 mm which is slightly higher than the analysis with steel model. The deformation plot obtained from the analysis is shown below in figure 5.

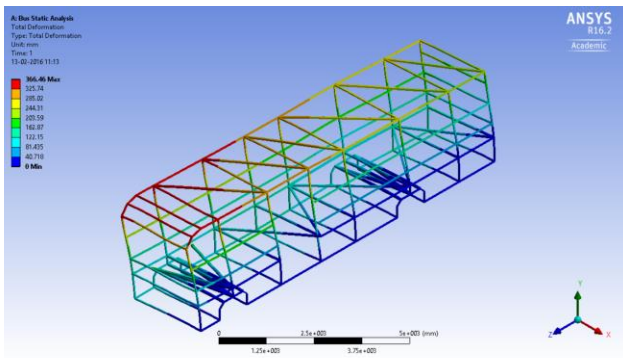


**Figure 5. Total deformation for model with composite material**

The values of maximum direct stress and combined stress due to applied load are 58.483 MPa and 1042.6 MPa respectively.

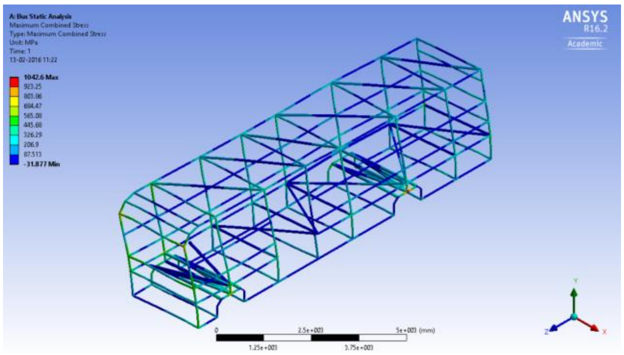


**Figure 6. Maximum combined stress distribution**

The mass of FE model of bus superstructure with carbon epoxy material along with steel is calculated as 1473.7 kg.

## 4. COMPARISON OF RESULTS

In the second analysis performed by using carbon epoxy composite along with steel in bus superstructure, mass of the bus is less than the pure steel model.

**Table 2. Comparison of results of the two studies**

| Property | Pure steel model | Steel and carbon epoxy composite model |
|---|---|---|
| Maximum direct stress | 59.814 MPa | 58.483 MPa |
| Maximum combined stress | 1014.9 MPa | 1042.6 MPa |
| Maximum Deformation | 363.31 mm | 366.46 mm |
| Weight | 1691.8 kg | 1473.7 kg |

Maximum direct stress in the elements has reduced to 58.483 MPa in case of analysis with composite material along with a weight saving of around 13% than the pure structural steel model.

## 5. CONCLUSIONS

Carbon fiber reinforced polymer (CFRP) composites are light in weight and stronger than the steel. The use of carbon epoxy composite along with the structural steel in FE model has lead to the weight reduction by 13% as compared to pure steel model. But cost of the composite materials is a major issue for the transportation industry to replace the existing conventional steel material. Thus the bus superstructure should be optimized for weight, strength and cost in order to achieve the best design. The thickness of beam elements used in the superstructure can be optimized so that resulting weight of the superstructure is minimum corresponding to those optimum thickness values. Also the maximum stresses arising due to the loading must be less than yield stress of the material and all the beam elements must bear some minimum stress. Thus the objective function to be used for weight minimization may include maximum stress and minimum stress constraints.

## 6. REFERENCES

[1] J. Karlinski, M. Ptak, P. Dzialak, and E. Rusinski, “Strength analysis of bus superstructure according to Regulation No. 66 of UN/ECE,” *Arch. Civ. Mech. Eng.*, vol. 14, no. 3, pp. 342–353, 2014.

[2] K. T. Kang, H. J. Chun, J. C. Park, W. J. Na, H. T. Hong, and I. H. Hwang, “Design of a composite roll bar for the improvement of bus rollover crashworthiness,” *Compos. Part B Eng.*, vol. 43, no. 4, pp. 1705–1713, 2012.

[3] M. Bin Yusof, M. Amirul, and A. Bin, “Effect ofMass onBus Superstructure Strength Having Rollover Crash,” *Int. Sci. Index*, vol. 6, no. 8, pp. 1443–1449, 2012.

[4] V. Belsare, P. C. Pathak, and M. Kulkarni, “Rollover Analysis of Passenger Bus as per AIS-031,” *Int. J. Eng. Res. Dev.*, vol. 4, no. 5, pp. 49–59, 2012.

[5] D. Valladares, R. Miralbes, and L. Castejon, “Development of a Numerical Technique for Bus Rollover Test Simulation by the F . E . M .,” *Proc. World Congr. Eng. 2010*, vol. II, 2010.

[6] H. Ning, G. M. Janowski, U. K. Vaidya, and G. Husman, “Thermoplastic sandwich structure design and manufacturing for the body panel of mass transit vehicle,” *Compos. Struct.*, vol. 80, pp. 82–91, 2007.

[7] H. Ning, U. Vaidya, G. M. Janowski, and G. Husman, “Design , manufacture and analysis of a thermoplastic composite frame structure for mass transit,” *Compos. Struct.*, vol. 80, pp. 105–116, 2007.

[8] H. Ning, S. Pillay, and U. K. Vaidya, “Design and development of thermoplastic composite roof door for mass transit bus,” *Mater. Des.*, vol. 30, no. 4, pp. 983–991, 2009.

[9] G. M. UDAY K. VAIDYA, FRANCIS SAMALOT, SELVUM PILLAY and G. H. A. K. G. JANOWSKI, “Design and Manufacture of Woven Reinforced Glass/Polypropylene Composites for Mass Transit Floor Structure,” *J. Compos. Mater.*, vol. 38, no. 21, 2004.

[10] L. Kwasniewski, C. Bojanowski, J. Siervogel, J. W. Wekezer, and K. Cichocki, “Crash and safety assessment program for paratransit buses,” *Int. J. Impact Eng.*, vol. 36, no. 2, pp. 235–242, 2009.