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.

References


Index Terms

Computer Science

Information Sciences
Keywords
Superstructure  Rollover  Finite Element  Survival Space  Simulation  Epoxy  Shear Transverse.