IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 5 Ver. I (Sep- Oct. 2014), PP 39-47
www.iosrjournals.org

A New Design and Analysis of BUS Body Structure

Prasannapriya. Chinta [*1], Dr.L.V.Venugopal Rao [*2]
M. Tech (Machine design) Department of Mechanical, from MVGCE, chinthalavalasa, vizayanagaram.#[1]
Associate Professor, Department of Mechanical, from MVGCE, chinthalavalasa, vizayanagaram.[#2]

Abstract: *Optimization of mechanical response of automotive and body designs are increasingly relies on new models. Generally in international market for passenger's buses design processes can rely on supercomputing facilities. Nowadays for the passenger buses have many local producers which construct vehicles based on local needs. In the competitive to stay these producers comply with the same requirements and weight reduction of their international counterparts without access to latest computation facilities. This paper proposes a new method for designing a bus body structure is designed and modelled in 3D modelling software Pro/Engineer. The original body is redesigned by changing the thickness and reducing the number of elements so that the total weight of the bus is reduced. The present used material for structure is steel. It is replaced with composite materials Kevlar and S 2 Glass Epoxy. The density of steel is more than that of composite materials, so by replacing with composites, the weight of the structure is reduced. Structural and Dynamic analysis is done on both the structures using three materials to determine the strength of the structure. Analysis is done in Ansys.*

Index terms: *Bus body structure, Redesign, Structural finite element Analysis.*

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, bit 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 philosophy. 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 MAN, BENZ, VOLVO, ISUZU, DAEWOO, HINO and so on. The chassis consists of two main types; the single piece and the three joint combination parts. The single piece chassis is used for the medium bus size with one floor, whereas the three combination parts are used for the long bus size or two floor bus. The second part is the bus body structure. The comprises of bus body have six main components the left and right frame side, the front and back frame side, the top and bottom frame side. In that the top frame side is sometime called roof frame side. The bottom frame side is also called floor frame side. The left and right side are similar but the left side is normally composed of two passenger doors. On the other hand, the right side has two doors the driver door and emergency door. In addition, the both frame sides are installed by mirrors and welded with sheet metal. They are concerned to be critical parts. They must be strong. The parts need to be analytical tests by at least simulation or physical test.

Torsion and bending tests are widely simulated by FE analysis. However, the strength of this design is affected by the manufacturing. For example the special type of welding such as MIG, TIG, and spot welding arc much better than the normal arc welding process. However, such manufacturing process is not concerned in this study. The third part, the top frame or the roof frame is considered as the critical part that is needed to be a strength part in order to be ensured safety for the passengers. This part must be sufficiently strong. It must be supported by the total weight from different loads such as interior components, air conditioners passenger carrying loads even the aero dynamic load. Then, the back frame and the front frame are mostly supported and joined with the left and right sides as well as the roof frame and the floor frame. These two parts need to be both strong and beauty style. Therefore the shape is quite become curvature, slop and good aero dynamic. The existing part is further combined by a lot of pieces which is here called trusts. The trusts are can be typed such as straight trusts, angle trusts, diagonal trusts and so on.

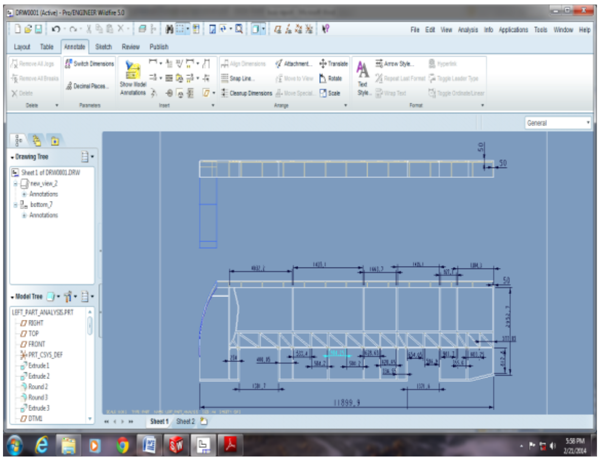
II. Related Work

Bus body manufacturing process consists of several steps. The first step is to design the body which is matched with a selected chassis. The critical dimensions are the length, the width, the height. They must be balanced in order to keep the vehicle stability. The second step is to check the chassis details. They are mirrors;

turn left and right lights, control panel, mileage panel, fuel vessel, battery, spare tire, etc. The third step is to start manufacturing process. The chassis frame is drilled at the supporting point which has 16 points. The angle plates are extended all of 16 points and tightened by bolts and nuts. The next step is to put the beam bar which is used to support the whole mass of bus body. They are 8 beams. However, such beams are adjusted to be flat. The next step is to install the 13 major columns which are used to fix with the mirrors. Then, the door columns are jointed together. Next, the front columns of the left and right sides are welded. Then the top beams are joined. The trust bars for the front frame is then welded. The front and back console fibers are attracted. The next step is to build left and right frame structure. This step takes a long processes and long times to finish. After that, it is an assembly process. The roof and floor frames are assembled with the left and right frame structure. Next step, it is to construct the seats and walking path. Then, the front and back frames are constructed together with mirrors followings with covering the whole bus body by sheet metals. Then, the interior floor and the whole inside passenger space of the bus are installed.

III. Research Objective

The main goal of this project is to integrated software in buses industry called as BUSCAD. The BUSCAD program is to rectify all the problems related to the different types of buses in the bus industry. Bus could be classified into different types these are minibus, city bus, Intercity Bus, High Dick, Supper high Deck, School bus, ect as shown in figure1, which shows the proposed program.



Computer-aided design (CAD), also known as computer-aided design and drafting (CADD), is the use of computer technology for the process of design and design-documentation. Computer Aided Drafting describes the process of drafting with a computer. CADD software, or environments, provide the user with input-tools for the purpose of streamlining design processes; drafting, documentation, and manufacturing processes. CADD output is often in the form of electronic files for print or machining operations. The development of CADD-based software is in direct correlation with the processes it seeks to economize industry-based software (construction, manufacturing, etc.) typically uses vector-based (linear) environments whereas graphic-based software utilizes raster-based (pixilated) environments. The bus body redesign is developed from the existing work based on the knowledge obtained from the expert working division of the bus body manufacturing. The redesign part is analysed by FE process and the result is compared to the accepted standard values and the previous analysis results from the existing design. The existing model design is not available for the analysis of 2D sketched drawing is currently used systems. In this paper we need the CAD model for the bus body design. Then analysed by the beam method on FEA, it gives effective design has to be modelled by wireframe. The selected loads are either point loads or distribution loads, come from the weights of bus components such as seats, air conditioners, lights and sound systems, doors, windows, liquid or gas tank, etc.This research have analysis each part separately. Finally the loads from one part to be another correctly transferred before the analysis can be preceded.

1.1. Proposal of weight reduction method

In this project, a bus body structure is designed and modelled in 3D modelling software Pro/Engineer. The dimension of the body structure is taken from the journal specified in literature survey. The original body is redesigned by changing the thickness and reducing the number of elements so that the total weight of the bus is reduced. The present used material for structure is steel. It is replaced with composite materials Kevlar and S 2 Glass Epoxy. The density of steel is more than that of composite materials, so by replacing with composites, the weight of the structure is reduced.

Structural and Dynamic analysis is done on both the structures using three materials to determine the strength of the structure. Analysis is done in Ansys.

1.2. Pro/Engineer:

Pro/ENGINEER is a feature based, parametric solid modelling program. As such, it's use is significantly different from conventional drafting programs. In conventional drafting (either manual or computer assisted), various views of a part are created in an attempt to describe the geometry. Each view incorporates aspects of various features (surfaces, cuts, radii, holes, protrusions) but the features are not individually defined. In feature based modelling, each feature is individually described then integrated into the part. The other significant aspect of conventional drafting is that the part geometry is defined by the drawing. If it is desired to change the size, shape, or location of a feature, the physical lines on the drawing must be changed (in each affected view) then associated dimensions are updated. When using parametric modelling, the features are driven by the dimensions (parameters). To modify the diameter of a hole, the hole diameter parameter value is changed. This automatically modifies the feature wherever it occurs - drawing views, assemblies, etc. Another unique attribute of Pro/ENGINEER is that it is a solid modelling program.

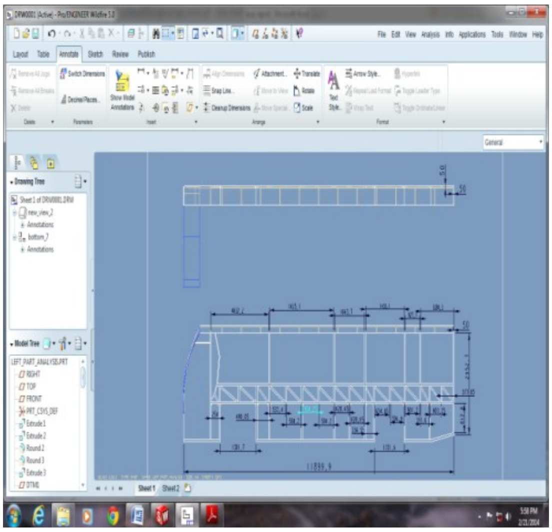
1.3. Engineering Design

Pro/Engineer offers a range of tools to enable the generation of a complete digital representation of the product being designed. In addition to the general geometry tools there is also the ability to generate geometry of other integrated design disciplines such as industrial and standard pipe work and complete wiring definitions. Tools are also available to support collaborative development. A number of concept design tools that provide up-front Industrial Design concepts can then be used in the downstream process of engineering the product. These range from conceptual Industrial design sketches, reverse engineering with point cloud data and comprehensive freeform surface tools.

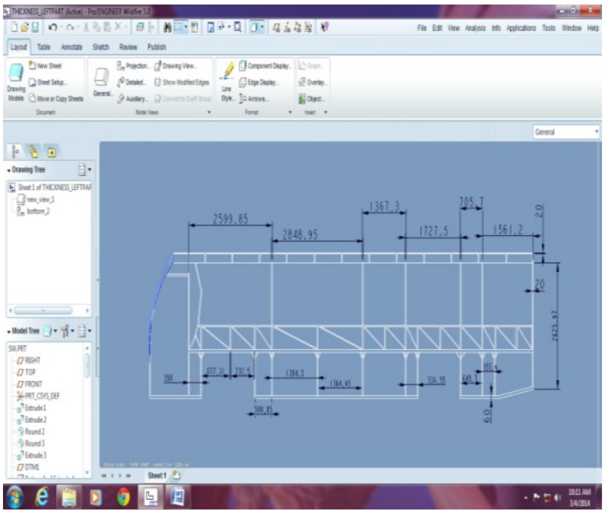
1.4. Manufacturing

By using the fundamental abilities of the software with regards to the single data source principle, it provides a rich set of tools in the manufacturing environment in the form of tooling design and simulated CNC machining and output. Tooling options cover specialty tools for molding, die-casting and progressive tooling design.

Original Model:



Modified Model:



IV. Result Of Fitness Analysis

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested. In practice, a finite element analysis usually consists of three principal steps:

a) **Pre-processing**:

The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical "pre-processor" to assist in this rather tedious chore. Some of these pre-processors can overlay a mesh on a pre-existing CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

b) **Analysis**:

The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations

Kijuj = fi

Where u and f are the displacements and externally applied forces at the nodal points. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.

c) **Postprocessing**:

In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. A typical postprocessor display overlays colored contours representing stress levels on the model, showing a full field picture similar to that of photo elastic or moiré experimental results.

ANSYS is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. ANSYS is also used in Civil and Electrical Engineering, as well as the Physics and Chemistry departments.

Like solving any problem analytically, you need to define (1) your solution domain, (2) the physical model, (3) boundary conditions and (4) the physical properties. You then solve the problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in an otherwise too complex situation. Below describes the processes in terminology slightly more attune to the software.

- ✓ **Build Geometry**

Construct a two or three dimensional representation of the object to be modeled and tested using the work plane coordinate system within ANSYS.

- ✓ **Define Material Properties**

Now that the part exists, define a library of the necessary materials that compose the object (or project) being modeled. This includes thermal and mechanical properties.

- ✓ **Generate Mesh**

At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces.

- ✓ **Apply Loads**

Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions.

- ✓ **Obtain Solution**

This is actually a step, because ANSYS needs to understand within what state (steady state, transient… etc.) the problem must be solved.

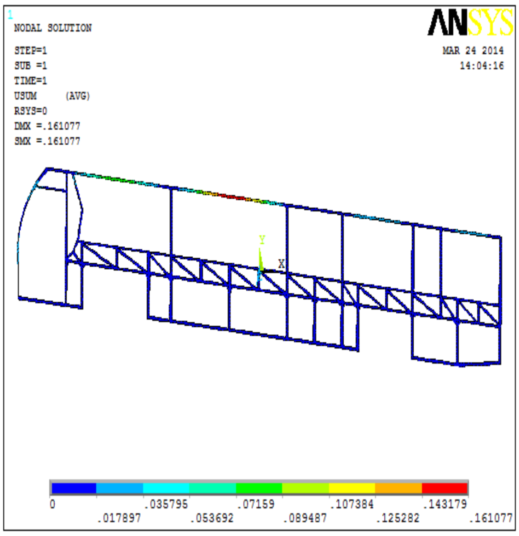
- ✓ **Present the Results**

After the solution has been obtained, there are many ways to present ANSYS' results, choose from many options such as tables, graphs, and contour plots.

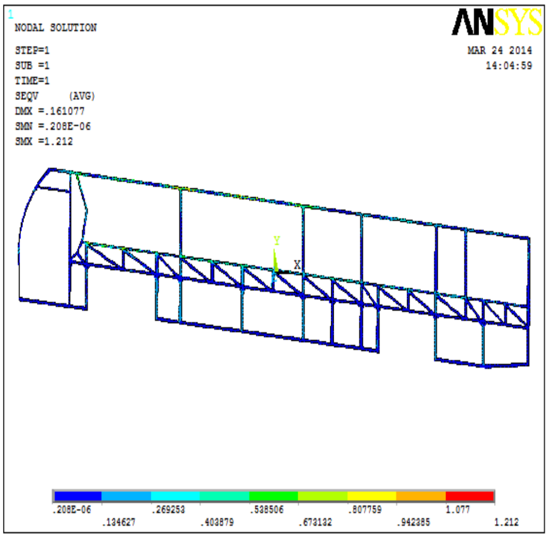
KEVLAR

V. Material Properties

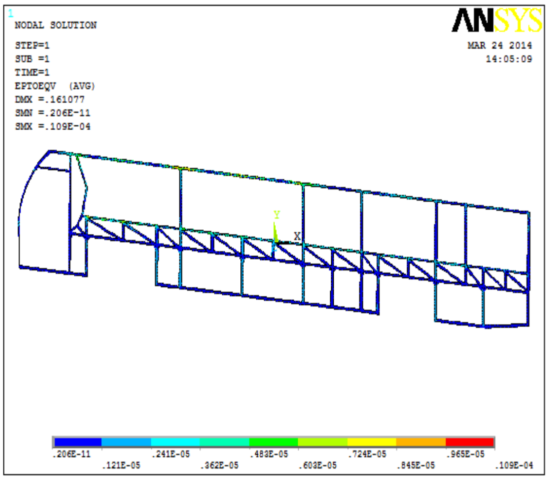
Youngs Modulus (EX) : 112000N/mm^2
Poissons Ratio (PRXY) : 0.36
Density :0.00000144 kg/mm^3
DISPLACEMENT
ORIGINAL MODEL



Stress



Strain:



VI. Results Table

	STELL	KEVLAR	S2_GLASS
Displacement (mm)	0.8795	0.161077	0.207357
VONMISES STRESS (N/mm^2)	1.213	1.212	1.214
STRAIN	$0.594*10^{-5}$	$0.109*10^{-4}$	$0.140*10^{-4}$

Modified:

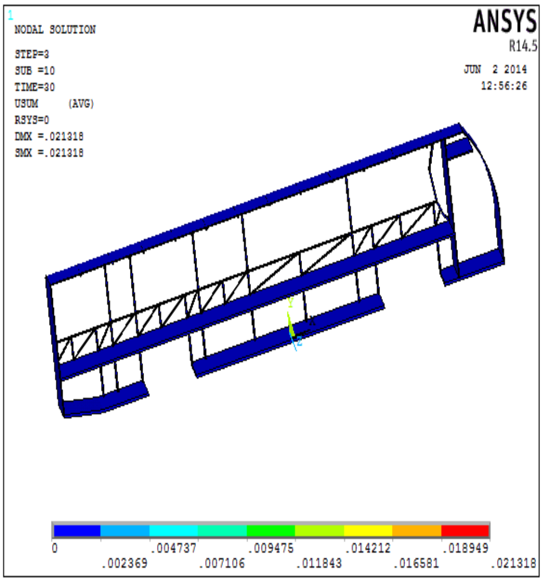
MATERIAL PROPERTIES

Young's Modulus (EX) : $112000 N/mm^2$

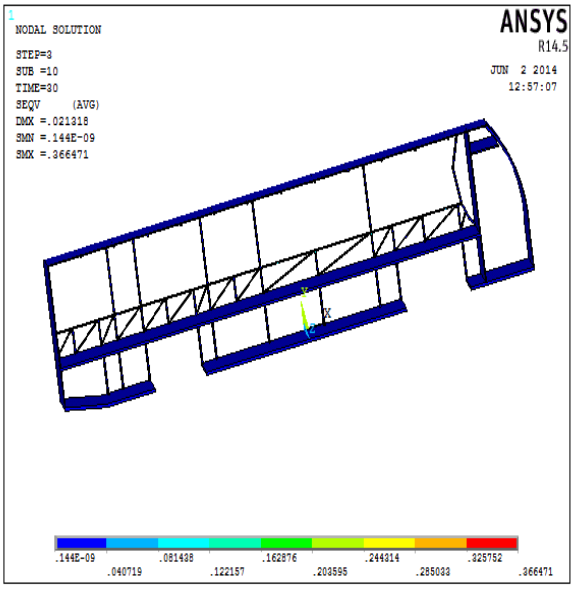
Poissons Ratio (PRXY) : 0.36

Density : $0.00000144 kg/mm^3$

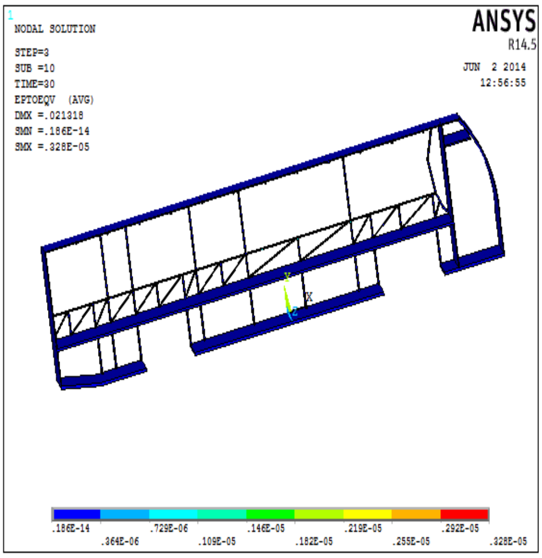
Post Processor:
DISPLACEMENT
General Post Processor – Plot Results – Contour Plot - Nodal Solution – DOF Solution – Displacement Vector Sum



STRESS
General Post Processor – Plot Results – Contour Plot – Nodal Solution – Stress – Von Mises Stres



STRAIN
General Post Processor – Plot Results – Contour Plot – Nodal Solution – Stress – total mechanical Von Mises Strain



MODIFIED MODEL

	STELL	KEVLAR	S2_GLASS
Displacement (mm)	0.363507	0.665631	0.8572
Vonmises stress (n/mm^2)	3.572	3.571	3.57
Strain	0.175*10^{-4}	0.319*10^{-4}	0.412*10^{-4}

VII. Conclusion

In this project, a bus body structure is designed and modeled in 3D modeling software Pro/Engineer. The dimensions of the body structure are taken from the journal specified in literature survey. The original body is redesigned by changing the thickness and reducing the number of elements so that the total weight of the bus is reduced. The present used material for structure is steel. It is replaced with composite materials Kevlar and S 2 Glass Epoxy. The density of steel is more than that of composite materials, so by replacing with composites, the weight of the structure is reduced. Structural and Dynamic analysis is done on both the structures using three materials to determine the strength of the structure. Analysis is done in Ansys. By observing the analysis results, the displacement and stress values obtained are within the limits for the modified model. By using composite materials also, the stress values are within the limits, and strength of the composite materials is more. So it can be concluded that by reducing the thickness and also using composite materials yields better results than original model and conventional steel.

References

[1]. Chandna P., Bhushan G., Singh V. P., Kumar D. and Kumar A., Finite element analysis of a bus body structure using CAE tools. Kurukshetra University, 2008

[2]. S Manokruang, Methodology of Bus-Body Structural Redesign for Lightweight Productivity Improvement, AIJSTPME, 2009, 2(2), 79-87.

[3]. Abaqus Documentation v. 6.11

[4]. P Eriksson, Optimization of a Bus Body Structure, Heavy Vehicle Systems, A Series of the Int. J. of Vehicle Design, 2001, 8(1),1-16.

[5]. Lan F, Chen J and Lin J. Comparative analysis for bus side structures and lightweight optimization, Proc. Instn Mech. Engrs,2004, 1067-1075.

[6]. Yi L. Bus skeleton lightweight design based on reduced model and mixed variables optimization. in Computer Science and Information Technology (ICCSIT), 2010.

[7]. X Zhang. A Study on Shape Optimization of Bus Body Structure Based on Stiffness Sensitivity Analysis. Computer-AidedIndustrial Design & Conceptual Design, 2009.
[8]. A Gauchía, Torsion Stiffness and Weight Optimization of a Real Bus Structure, International Journal of Automotive Technology, 2010, 11(1), 41-47.
[9]. Genta G. The Automotive Chassis: Components Design, Springer, 2009
[10]. Reimpell J. The Automotive Chassis: Engineering Principles, Butterworth Heinemann, 2001
[11]. Singer F. Strength of materials. 3 ed. Engineering Textbook Series, Michigan, USA: Harper & Row, 1980
[12]. Syam A. Design Capacity Tables for Structural Steel Hollow Sections. A.I.S. Construction, and A.S. Institute, Australian Steel Institute, 1992