International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 10, October - 2013

Optimization of Weight and Stress Reduction of Dump for Automotive Vehicles

N.Nagendra Kumar [1] **B. Jithendra** [2] **Malaga. Anil Kumar** [3]

1. Student, 10A01D1508, M.Tech (Machine Design), NOVA College of Engineering & Technology, Jangareddy gudem,
2. Assistant Professor, Department of Mechanical Engineering, NOVA College of Engineering & Technology, Jangareddy gudem,
3. Assistant Professor, Department of Mechanical Engineering, PACE Institute of Technology & Sciences, Ongole,

Abstract

The truck industry is a significant lifeline of the country's economic activity. About 90 per cent of vehicles are owned and operated by individual operators. A large majority of the truck cabs, truck bodies and trailers are constructed by units in semi-organized / unorganized sectors spread over the country. There is considerable scope to improve the design of their products. Every extra pound of vehicle weight increases manufacturing cost, lower fuel efficiency and reduces vehicle payload capacity. With this concept of reducing weight and stress reduction the optimized model of tipper dump body is modeled and analyzed. By conducting the Finite Element Analysis on the three Models the optimized parameters, optimized Model-IV is developed and analyzed. For the Model-IV (optimized) stress analysis is carried out and the results are presented.

1. Introduction

The truck industry is a significant lifeline of the country's economic activity. An important facet of this industry is its highly diversified character of ownership. About 90 per cent of vehicles are owned and operated by individual operators having 1 to 3 vehicles in their fleet. Last two decades have witnessed phenomenal increase in economic activity in India and to keep pace with the development, there is a necessity to accommodate higher levels of transportation. Equally important is the safety of these transportation modes and means. A large majority of the truck cabs, truck bodies and trailers are constructed by units in semi-organised / unorganized sectors spread over the country. There is considerable scope to improve the design of their products and process.

2. Finite Element Analysis

2.1 Introduction

The finite element is a mathematical method for solving ordinary and partial differential equations. Because it is a numerical method, it has the ability to solve complex problems that can be represented in differential equation form. As these types of equations occur naturally. In virtually all fields of the physical sciences, the applications of the Finite element method are limitless as regards the solution of practical Design problems.

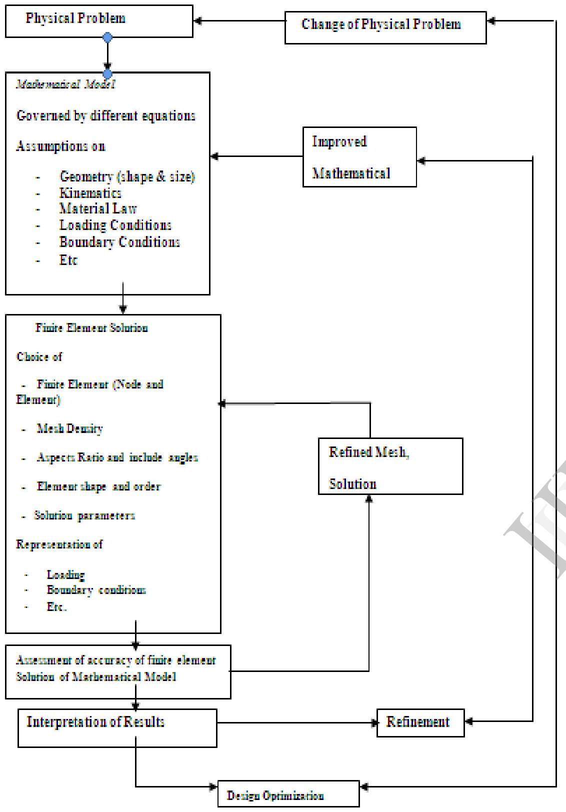
FEA consists of a computer model of a material or design that is loaded and analyzed for specific results. It is used in new product Design, and existing product refinement. A Design Engineer shall be able to verify a proposed design, which is intended to meet the customer specifications prior to manufacturing or construction. Things such as, modifying the design of an existing product or structure in order to qualify the product or structure for a new serviced condition. Can also be accomplished in case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

Terms commonly used in finite element method

- **Descritization**: The process of selecting only a certain number of discrete points in the
  body can be termed as Descritization.
- **Continuum**: The continuum is the physical body, structure or solid being analyzed.
- **Node**: The finite elements, which are interconnected at joints, are called nodes or nodal points.
- **Element**: Small geometrical regular figures are called elements.
- **Displace Models**: The simple functions, which are assumed to approximate the displacement for each element. These functions are called the displacement models or displacement functions.
- **Local coordinate system**: Local coordinate system is one that is defined for a particular element and not necessary for the entire body or structure.
- **Global system**: The coordinate system for entire body is called the global coordinate system.

- **Natural coordinate system**: Natural coordinate system is a local system, which permits the specification of point with in the element by a set of dimensionless numbers, whose magnitudes never exceeds unity.
- **Interpolation function**: It is a function, which has unit value at one nodal point and a zero value at all other nodal points.
- **Aspect ratio**: The aspect ratio describes the shapes of the element in the assemblage for two dimensional elements; this parameter is defined as the ratio of largest dimension of the element to the smallest dimension.
- **Field variables**: The principal unknowns of a problem are called the variables.

The process of F.E.A is shown in the below figure



"Figure 1. Process of FEA"

The following are the five basic steps involved in an FEA analysis:

1. Discretization of the Domain
2. Applications of Field/ Boundary conditions
3. Assembling the system equations
4. Solution for the system equations
5. Review of result.

3. FEA Software

There are many fea softwares available in the market. Some of them mostly used in Industry are ANSYS, ANSYS WORKBENCH, MSC NASTRAN, ABACUS.

3.1 Introduction to Ansys Workbench

The ANSYS Workbench is the framework upon which the industry's broadest and deepest suite of advanced engineering simulation technology is built. An innovative project schematic view ties together the entire simulation process, guiding the user through even complex multiphysics analyses with drag-and-drop simplicity. With bi-directional CAD connectivity, an automated project level update mechanism, pervasive parameter management and integrated optimization tools, the ANSYS Workbench delivers unprecedented productivity, enabling Simulation Driven Product Development.

3.2 Ansys Workbench Modules

- Design Modeler Geometry
- Simulation
- Finite Element Model
- AutoDyn
- Blade Geometry
- Meshing

3.3 Overall Steps For Using Simulation

This section describes the overall workflow involved when performing any analysis in Simulation. The following workflow steps are described:

- Attach Geometry
- Define Part Behavior
- Define Connections
- Apply Mesh Controls/Preview Mesh
- Define Analysis Type
- Establish Analysis Settings
- Define Initial Condition
- Apply Loads and Supports
- Solve
- Review Results
- Create Report (optional)

4. Problem Description

4.1 Description

In the present scenario, the automotive industry has been one of the rapid growing industries. Today there is demand on trucks, not only on the cost and weight aspects but also on the improved complete vehicle features and overall work performance In addition to this number of variants that are possible due to different types of designs and modularization, call for several design iterations to arrive at a suitable combination. The project work deals with tipper load/dump body. A large majority of the truck load bodies are constructed by units in unorganized sectors. There is considerable scope to improve the design of their product.

For optimization of dump body design, three models are chosen whose specifications are taken from the local industry. These are having a 14 cu.m capacity of volume.

4.1.1 Objective

The main objectives of the work is

- To reduce body weight.
- To determine the critical point which has the highest stress
- To modify the design of tipper body to get an optimized in terms of reducing weight and reducing stresses.

4.2 Methodology

The methodology of work is outlined below

- Geometric Modeling of three models of tipper load body assembly in Pro-E3.0.
- Static analysis for three models of dump body for same (geometric, volumes) geometric features and loading conditions. In order to solve the problem of the project, a detailed finite element analysis is proposed to determine the total deformation and Equivalent stress in static condition using the analysis software ansys workbench.
- After analyzing the three models, a Fourth model (optimized) is developed and analyzed.

4.3 Design Parameter Details

The design parameters are listed below

4.3.1 Body Specifications for Three Models

Volume/load capacity	14cu.m
Dimensions :	
Length	4480mm
Width	2350mm
Height	1300mm
Bottom Floor thickness	6mm
Side guard thickness	5mm
Head Board thickness	5mm
Channels used for Cross Bearers :	
Box channel for Model-I	75mm*75mm*4mm
C-Channel for Model-II,III	200mm*75mm*4mm
Channels used for Long members :	
C-Channel	100mm*50mm*4mm
Material :	
For dump body	Mild Steel
Type of material carry	Sand, iron ore, boulders, coal, Road construction Material/Earth

“Table1. Body Specifications for Three Models”

4.3.2 Manufacturing Details

Welding :	
Process	Arc Welding
Electrode	Mild Steel Electrode
Electrode Size	3.15mm*350mm
Current Range	90-130Amp
Process	Cold rolling of sheets

“Table2. Manufacturing Details”

4.3.3 Selection of material for dump body

The following factors considered while selecting material:

1. Availability of the material
2. Suitability of the material for the working condition
3. Cost of the material

Properties of Mild Steel:

- Contains 0.16-0.29% carbon, therefore it is neither brittle nor ductile.
- It is cheap and malleable.
- It is often used when large amount of steel is needed, for example as structural steel.

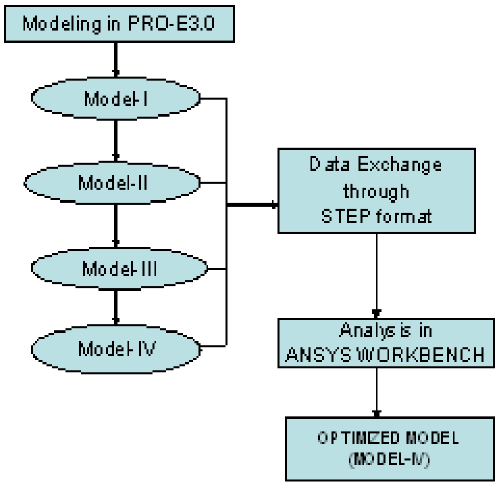
5. Modeling And Analysis

5.1 Modeling

The geometries under consideration are generated in the Pro-E CAD Modeling package. It is a powerful program used to create complex designs with great precision. It has properties like Feature-based nature, Bidirectional associative property and parametric nature. Parametric features are helpful in reusing three models of truck dump body to create new variant design. The three models are considered as viz., Model I, Model II and Model III. The three dump bodies are modeled.

5.2 Data Exchange

The Pro-E file is saved in *.stp format. STEP (Standard for Exchange of Product Data) is an exchange for product data in support of industrial automation. Product data is more general than the "product definition data" which forms the core philosophy of IGES. The general emphasis of STEP is to eliminate the human presence from the "product data". The central unit of data exchange in the STEP model is the application, which contains various types of entities. This approach maintains all the meaningful associatives and relationships between the application entities. Therefore STEP is to represent all product information, in a common data format, throughout a product's entire life cycle.



"Figure2. Flow chart of approach to problem solution"

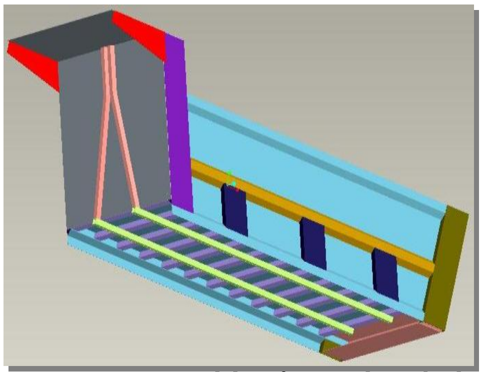
5.3 Model-I

5.3.1 Geometric Model of Dump-Body

Geometric model of dump body is depicted in figure2 and is generated in Pro-E3.0 CAD Modeling package. The model has length of 4880mm, width of 2360mm and height of 1300mm.The material of dump body is Mild steel with 250 MPa of yield strength and 460.MPa of Utimate tensile strength. The other properties of dump body material are tabulated in table3.These properties above mentioned related to all the three models. No. of parts used for this Model-I are 53. The bottom, sides and head board sheets thicknesses are 6mm,5mm and 5mm respectively for Model-I.

Modulus Elasticity E(MPa)	Density ρ(kg/m³)	Poisson Ratio	Yield Strength (MPa)	Tensile Strength (MPa)
2e+005	7850	0.3	250	460

"Table3. Properties of tipper dump body"



'Figure3. Pro-E Model-I of tipper dump body"

Total Mass	2379.9kg
Center of Mass :	
Xc	1838.5mm
Yc	665.45mm
Zc	338.43mm
No. of parts	53

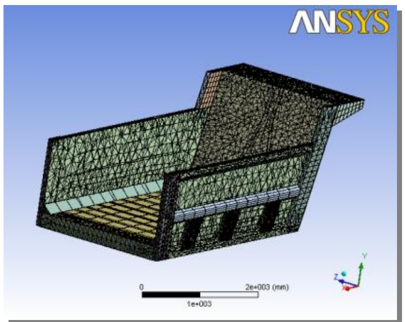
"Table4. Geometry details of Model-I"

5.3.2 The Model after Meshing

The automatic mesh generate option is chosen. The element type is Solid element mid side nodes and is program controlled. The elements are in as in Table5.

Element Types	SOLID186, SOLID187, TARGE170, CONTA174
No. of Nodes	99863
No. of Elements	47792

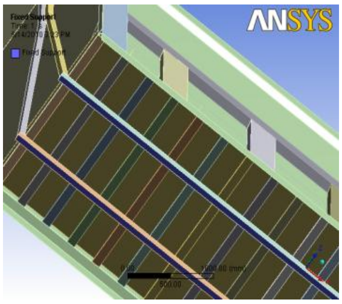
"Table5. Meshing details of Model-I"



"Figure4. Meshing of Model-I"

5.3.3 Boundary Conditions

A fixed support is given at the bottom surface of cross bearers as shown in figure5. Since the cross bearers are placed on subframe so the Ux, Uy,Uz are taken as zero displacement.



“Figure5. Boundary condition representation of Model-I”

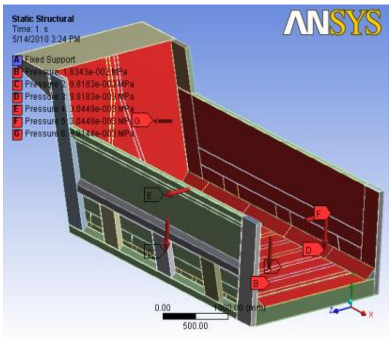
5.3.4 Loading

The tipper dump body model is loaded by static forces from material it carry. For this 14cu.m capacity dump body the load it carries is 18tons. The load is assumed as a uniform pressure obtained from the maximum loaded weight divided by the total contact area between load it carry and upper surface of bottom sheet.

Bottom sheet = 18tons of load (Vertical force)

Side sheets = 10% of load (Horizontal force or side thrust)

Head Board = 15% of load



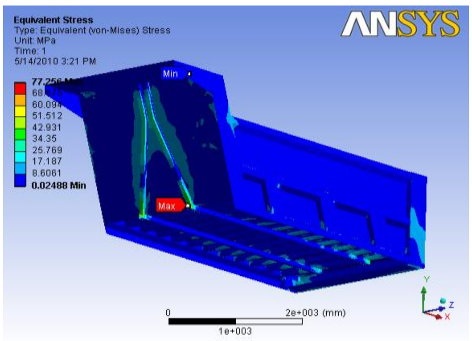
“Figure6. Static load representation of Model-I”

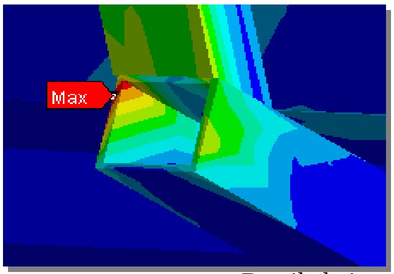
5.3.5 Solution

5.3.5.1 Equivalent stress

The maximum equivalent stress occurred at front side of cross bearer where hydraulic channel is placed on it. The detailed view is as shown below.

Equivalent Stress	Max (MPa)	Min (MPa)
	77.25	0.02





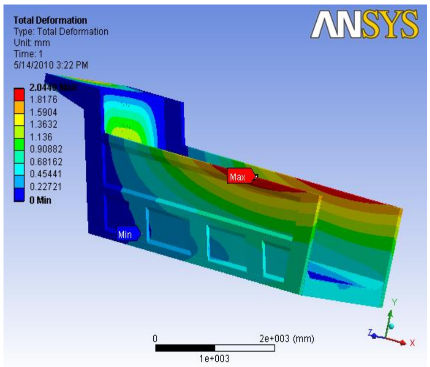
Detailed view

“Figure7. Von-Mises stress distribution and critical point location of Model-I”

5.3.5.2 Total deformation

The maximum deformation occurred at side sheet top surface.

Total deformation	Max(mm)	Min(mm)
	2.04	0

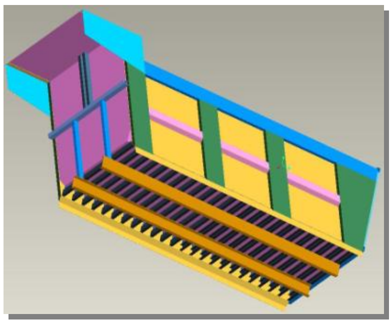


“Figure8.Total deformation and maximum displacement location of Model-I”

5.4 Model-II

5.4.1 Geometric Model of Dump Body

The model-II of dump body is modeled in Pro-E. The no. of parts used for this model-II is 105. The bottom, sides and head board sheets thicknesses are 6mm, 5mm and 5mm respectively for Model-II.



“Figure9. Pro-E Model-II of dump body”

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 10, October - 2013

Total Mass	2477.4kg
Center of Mass :	
Xc	501.27mm
Yc	718.68mm
Zc	-2653.8mm
No. of parts	105

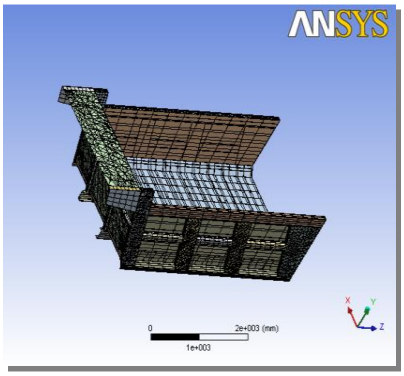
"Table6. Geometry details of Model-II"

5.4.2 Model after Meshing

The option automatic mesh generation is chosen and element types are Solid element mid side nodes and are set under program control. The model after meshing is as shown if fig.10 the meshing details of Model-II are shown in Table7.

Element types	SOLID186, SOLID187, TARGE170, CONTA174
No. of nodes	86117
No. of elements	26528

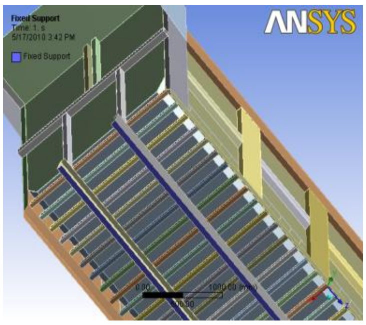
"Table7. Meshing details of Model-II"



"Figure10. Meshing of Model-II"

5.4.3 Boundary Conditions

A fixed support is given at the bottom surface of cross bearers as shown in figure11. Since the cross bearers are placed on subframe the Ux, Uy,Uz are taken as zero displacement.

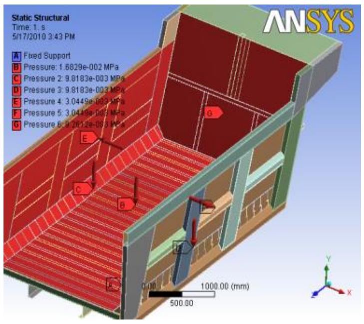


"Figure11.Boundary condition representation of Model-II"

5.4.4 Loading

The tipper dump body model is loaded by static forces from material it carry. For this 14cu.m capacity dump body the load it carries is 18tons. The load is assumed as a uniform pressure obtained from the maximum loaded weight divided by the total contact area between load it carry and upper surface of bottom sheet.

Bottom sheet = 18tons of load (Vertical force)
Side sheets = 10% of load (Horizontal force or side thrust)
Head Board = 15% of load



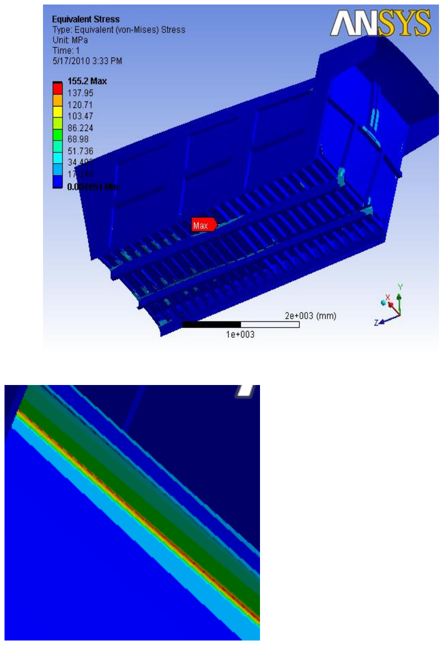
"Figure12. Static load representation of Model-II"

5.4.5 Solution

5.4.5.1 Equivalent Stress

The maximum equivalent stress occurred at bottom inner side of cross bearer. The detailed view is as shown below.

Equivalent Stress	Max (MPa)	Min(MPa)
	155.2	0



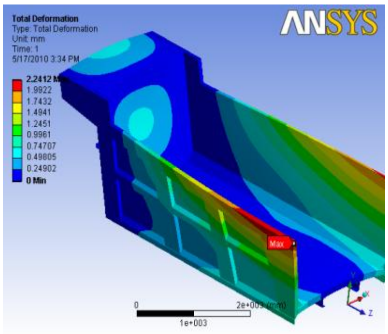
Detailed view

"Figure13. Von-Mises stress distribution and critical point location of Model-II"

5.4.5.2 Total deformation

The maximum deformation occurred at side sheet of top channel surface.

Total deformation	Max(mm)	Min(mm)
	2.24	0

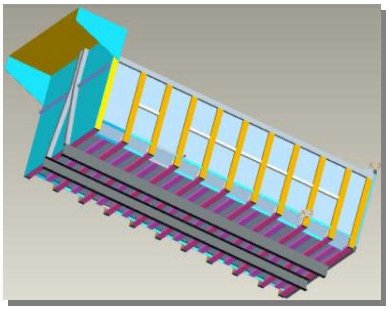


"Figure14. Total deformation and maximum displacement location of Model-II"

5.5 Model-III

5.5.1 Geometric Model of Tipper Dump Body

The model-III of dump body is modeled in Pro-E. The no. of parts used for this model-III is 169. The bottom, sides and head board sheets thicknesses are 6mm, 5mm and 5mm respectively for Model-III.



"Figure15. Pro-E Model-III of Dump Body"

Total Mass	2075.1kg
Center of Mass :	
Xc	-79.614mm
Yc	623.21mm
Zc	-2050.9mm
No. of parts	169

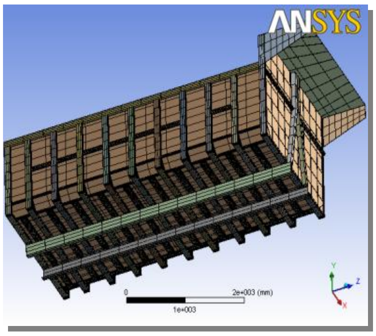
"Table8. Geometry details of Model-III"

5.5.2 The Model after Meshing

The option automatic mesh generation is chosen and element type is Solid element mid side nodes and is set under program control. The model after meshing is as shown if fig.16 the meshing details of Model-III are shown in Table9.

Element types	SOLID186, SOLID187, TARGE170, CONTA174
No. of nodes	152296
No. of elements	50054

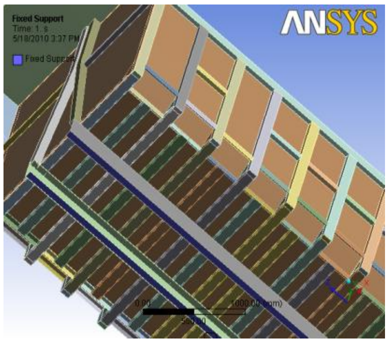
"Table9. Meshing Details of Model-III"



"Figure16. Meshing of Model-III"

5.5.3 Boundary Conditions

A fixed support is given at the bottom surface of cross bearers as shown in figure6.16.Since the cross bearers are placed on subframe the Ux, Uy,Uz are taken as zero displacement.

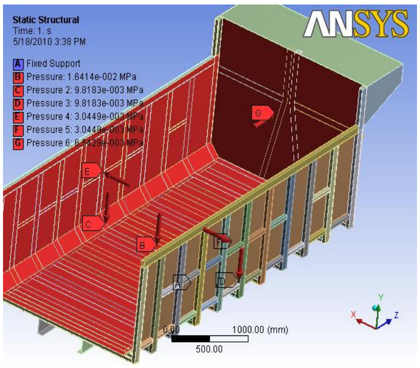


"Figure17. Boundary condition representation of Model-III"

5.5.4 Loading

The tipper dump body model is loaded by static forces from material it carry. For this 14cu.m capacity dump body the load it carries is 18tons. The load is assumed as a uniform pressure obtained from the maximum loaded weight divided by the total contact area between load it carry and upper surface of bottom sheet.

Bottom sheet = 18 tons of load (Vertical force)
Side sheets = 10% of load (Horizontal force or side thrust)
Head Board = 15% of load



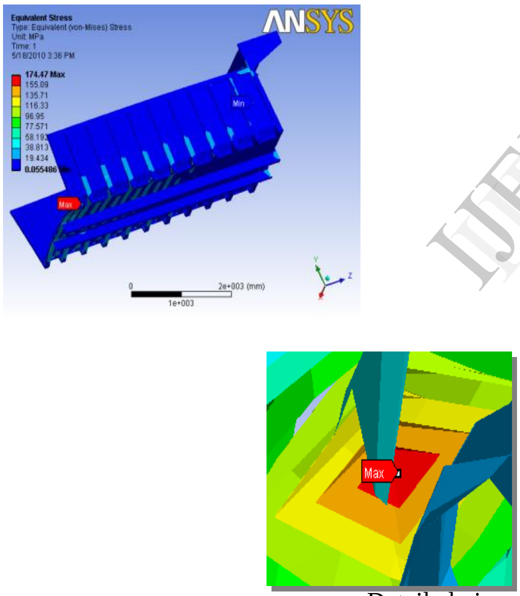
“Figure18. Static load representation of Model-III”

5.5.5 Solution

5.5.5.1 Equivalent Stress

The maximum equivalent stress occurred at bottom inner side of angular section. The detailed view is as shown below.

Equivalent Stress	Max (MPa)	Min(MPa)
	174.47	0.05



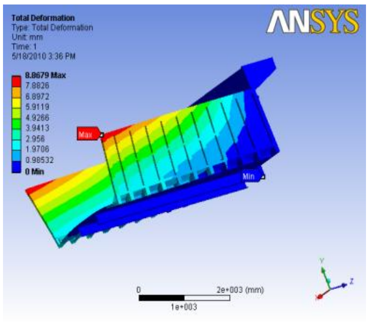
Detailed view

“Figure19. Von-Mises stress distribution and critical point location of Model-III”

5.5.5.2 Total deformation

The maximum deformation occurred at side sheet of top channel surface.

Total deformation	Max(mm)	Min(mm)
	8.86	0

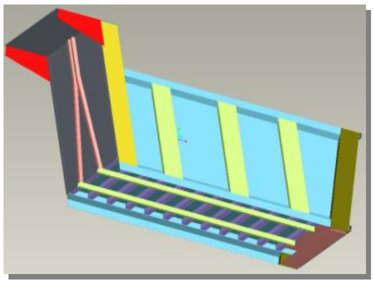


“Figure20. Total deformation and maximum displacement location of Model-III”

5.6 Model-IV (Optimized Model)

5.6.1 Geometric Model of Dump Body

The model-IV of dump body is modeled in Pro-E. The no. of parts used for this model-IV is 51. The bottom, sides and head board sheets thicknesses are 5mm, 4mm and 4mm respectively for Model-IV.



“Figure21. Pro-E Model-IV of Dump Body”

Total Mass	1991.8kg
Center of Mass :	
Xc	1818.9mm
Yc	661.33mm
Zc	327.27mm
No. of parts	51

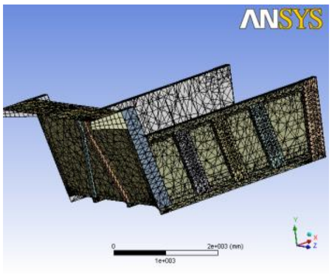
“Table10. Geometry details of Model-IV”

5.6.2 The Model after Meshing

The option automatic mesh generation is chosen and element type is Solid element mid side nodes and is set under program control. The model after meshing is as shown if fig.22.the meshing details of Model-III are shown in Table11

Element types	SOLID186, SOLID187, TARGE170, CONTA174
No. of nodes	100490
No. of elements	49597

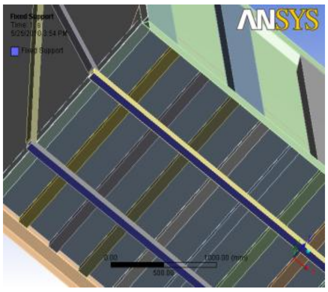
“Table11. Meshing Details of Model-IV”



“Figure22. Meshing of Model-IV”

5.6.3 Boundary Conditions

A fixed support is given at the bottom surface of cross bearers as shown in figure23.Since the cross bearers are placed on subframe the Ux, Uy,Uz are taken as zero displacement.



“Figure23. Boundary condition representation of Model-IV”

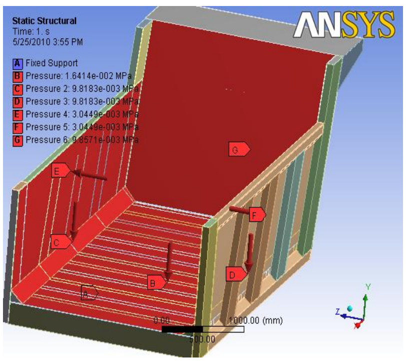
5.6.4 Loading

The tipper dump body model is loaded by static forces from material it carry. For this 14cu.m capacity dump body the load it carries is 18tons. The load is assumed as a uniform pressure obtained from the maximum loaded weight divided by the total contact area between load it carry and upper surface of bottom sheet.

Bottom sheet = 18tons of load (Vertical force)

Side sheets = 10% of load (Horizontal force or side thrust)

Head Board = 15% of load



“Figure24. Static load representation of Model-IV”

6. Results And Discussion

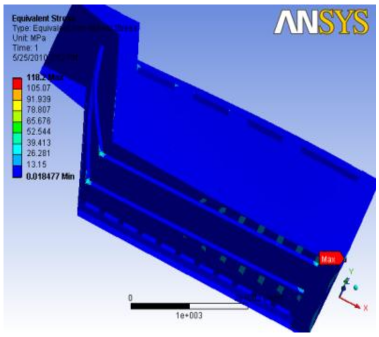
The weights of the models are shown in the table12. and the weight of the optimized model is 1.99 tons. It is giving a saving in weight of 388.1kgs comparing with Model-I, 485.6 kgs comparing with Model-II, 83.3 kgs comparing with Model-III.

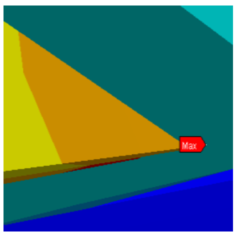
	MASS (kgs)	TOTAL DEFORMATION Maximum (mm)	EQUIVALENT STRESS (N/mm²)	No. of parts for fabrication
MODEL-I	2379.9	2.0	77.2	53
MODEL-II	2477.4	2.2	155.2	105
MODEL-III	2075.1	8.8	174.4	169
MODEL-IV (Optimized)	1991.8	1.8	118.2	51

“Table12. Comparisons of Mass, total deformation and Equivalent stress values of four models “

The three models are analyzed in ANSYS WORKBENCH. The obtained results are compared. An optimized model is developed. All the models are compared for stress and deformation. The results obtained for the optimized model are shown in the figure7.1 and figure7.2. The maximum equivalent stress occurred at bottom side of first rib section. The detailed view is as shown below in figure25.

Equivalent Stress	Max (MPa)	Min(MPa)
	118.2	0



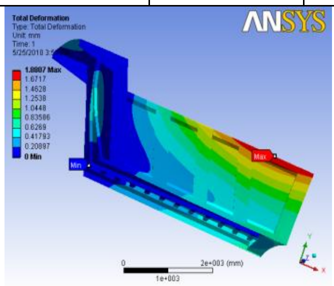


Detailed view

“Figure25. Von-Mises stress distribution and critical point location of optimized Model”

The maximum deformation occurred at the top surface of the side sheet and is shown in the figure7.2. The values are

Total deformation	Max(mm)	Min(mm)
	1.8	0



"Figure26. Total deformation and maximum displacement location of optimized Model"

Therefore the maximum stress obtained for the Optimized Model-IV is below the allowable stress of 125 MPa for Mild steel with factor of safety of 2, and the design is safe in static condition. If more than 18 tons load is applied on this model the maximum stress will not exceed the allowable stress and the model can withstand the load.

7. Conclusions And Future Scope

7.1 Conclusions

By conducting the FEM Analysis on the three Models of existing tipper dump bodies and by using AIS-093 code amended by ARAI weight reduction and stress reduction is done.

The following are the conclusions made from the investigation by comparing the three Model parameters Optimized Model is generated.

1. For the Optimized Model stress analysis is carried out and the equivalent stress is 117MPa and total deformation is 1.8mm is obtained.
2. Weight reduction of optimized model comparing with the other three models is 16.3%, 19.6% and 4% respectively.
3. By weight reduction, the material cost and fabrication cost is reduced for the vehicle.
4. Number of parts in the fabrication for the optimized model is reduced compared to the three models.

7.2 Future Scope

1. Since the total analysis is done in static conditions, based on these results Dynamic analysis can be done.
2. Other grades of Alloy steels can be used as material for dump body.
3. Mountings and sub frames can be included in the model of dump truck for analysis.

References

[1.] Mauritz Coetzee Axis Developments Ltd., Pretoria, South Africa, Heavy-Duty Lightweight, ANSYS Advantage, Volume I, Issue 2, 2007 www.ansys.com.

[2.] Sridhar Srikantan, Shekar Yerrpalli and Hamid Keshtkar, Durability design process for truck body structures, International Journal of vehicle Design, Volume 23, 2000.

[3.] R.J. Yang, Ching-Hung Chuang, Dingdongs Che and Ciro Soto, New application of topology optimization in automotive industry, International Journal of vehicle Design, Volume 23, 2000.

[4.] "Code of Practice for Construction and Approval of Truck Cabs, Truck Bodies and Trailers", The Automotive Research Association of India publication, 2008

[5.] S.Timoshenko and D.H.Young, Engineering Mechanics, McGraw-Hill International publication 4th edition.

[6.] PSG College of Technology, Design Data, DVP Printers Publication revised edition (1978).

[7.] Ibrahim Zeid, Mastering CAD/CAM, Tata McGraw-Hill Publications edition (2007).

[8.] R.B.Gupta, Automobile Engineering, Delhi Publication edition (2003).

[9.] Roslan Abd Rahman, Mohd Nasir Tamin, and Ojo Kurdi, Stress Analysis of Heavy Duty Truck Chassis as a Preliminary Data for its Fatigue Life Prediction using Fem, Jurnal Mekanikal, December 2008.