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 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

<table>
<thead>
<tr>
<th>Volume/load capacity</th>
<th>14cu.m</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Dimensions</strong></td>
<td></td>
</tr>
<tr>
<td>Length</td>
<td>4480mm</td>
</tr>
<tr>
<td>Width</td>
<td>2350mm</td>
</tr>
<tr>
<td>Height</td>
<td>1300mm</td>
</tr>
<tr>
<td>Bottom Floor thickness</td>
<td>6mm</td>
</tr>
<tr>
<td>Side guard thickness</td>
<td>5mm</td>
</tr>
<tr>
<td>Head Board thickness</td>
<td>5mm</td>
</tr>
<tr>
<td><strong>Channels used for Cross Bearers</strong></td>
<td></td>
</tr>
<tr>
<td>Box channel for Model-I</td>
<td>75mm<em>75mm</em>4mm</td>
</tr>
<tr>
<td>C-Channel for Model-II,III</td>
<td>200mm<em>75mm</em>4mm</td>
</tr>
</tbody>
</table>

#### 4.3.2 Manufacturing Details

<table>
<thead>
<tr>
<th><strong>Welding</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Process</td>
<td>Arc Welding</td>
</tr>
<tr>
<td>Electrode</td>
<td>Mild Steel Electrode</td>
</tr>
<tr>
<td>Electrode Size</td>
<td>3.15mm*350mm</td>
</tr>
<tr>
<td>Current Range</td>
<td>90-130Amp</td>
</tr>
<tr>
<td><strong>Process</strong></td>
<td>Cold rolling of sheets</td>
</tr>
</tbody>
</table>

### 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.

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 Ultimate 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.

<table>
<thead>
<tr>
<th>Modulus Elasticity E(MPa)</th>
<th>Density p(kg/m³)</th>
<th>Poisson Ratio</th>
<th>Yield Strength (MPa)</th>
<th>Tensile Strength (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0e+005</td>
<td>7850</td>
<td>0.3</td>
<td>250</td>
<td>460</td>
</tr>
</tbody>
</table>

“Table3. Properties of tipper dump body”

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.

<table>
<thead>
<tr>
<th>Element Types</th>
<th>SOLID186, SOLID187, TARGE170, CONTA174</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of Nodes</td>
<td>99863</td>
</tr>
<tr>
<td>No. of Elements</td>
<td>47792</td>
</tr>
</tbody>
</table>

“Table5. Meshing details 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.
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

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.

<table>
<thead>
<tr>
<th>Equivalent Stress</th>
<th>Max (MPa)</th>
<th>Min (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>77.25</td>
<td>0.02</td>
</tr>
</tbody>
</table>

5.3.5.2 Total deformation

The maximum deformation occurred at side sheet top surface.

<table>
<thead>
<tr>
<th>Total deformation</th>
<th>Max(mm)</th>
<th>Min(mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>2.04</td>
<td>0</td>
</tr>
</tbody>
</table>

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.
Table 6. 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 Table 7.

<table>
<thead>
<tr>
<th>Element types</th>
<th>SOLID186, SOLID187, TARGE170, CONTA174</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of nodes</td>
<td>86117</td>
</tr>
<tr>
<td>No. of elements</td>
<td>26528</td>
</tr>
</tbody>
</table>

Table 7. Meshing details of Model-II

5.4.3 Boundary Conditions

A fixed support is given at the bottom surface of cross bearers as shown in figure 11. Since the cross bearers are placed on subframe the Ux, Uy, Uz are taken as zero displacement.

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

Figure 12. 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.

<table>
<thead>
<tr>
<th>Equivalent Stress</th>
<th>Max (MPa)</th>
<th>Min (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>155.2</td>
<td>0</td>
</tr>
</tbody>
</table>

Figure 13. Von-Mises stress distribution and critical point location of Model-II

Detailed view
5.4.5.2 Total deformation
The maximum deformation occurred at side sheet of top channel surface.

<table>
<thead>
<tr>
<th>Total deformation</th>
<th>Max(mm)</th>
<th>Min(mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>2.24</td>
<td>0</td>
</tr>
</tbody>
</table>

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.

Element types
<table>
<thead>
<tr>
<th>SOLID186, SOLID187, TARGE170, CONTA174</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of nodes</td>
</tr>
<tr>
<td>No. of elements</td>
</tr>
</tbody>
</table>

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.

5.5.3 Boundary Conditions
A fixed support is given at the bottom surface of cross bearers as shown in figure 6.16.Since the cross bearers are placed on subframe the Ux, Uy, Uz are taken as zero displacement.

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
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.

<table>
<thead>
<tr>
<th>Equivalent Stress</th>
<th>Max (MPa)</th>
<th>Min (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>174.47</td>
<td>0.05</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Total deformation</th>
<th>Max(mm)</th>
<th>Min(mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>8.86</td>
<td>0</td>
</tr>
</tbody>
</table>

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.

<table>
<thead>
<tr>
<th>Total Mass</th>
<th>1991.8kg</th>
</tr>
</thead>
<tbody>
<tr>
<td>Center of Mass :</td>
<td></td>
</tr>
<tr>
<td>Xc</td>
<td>1818.9mm</td>
</tr>
<tr>
<td>Yc</td>
<td>661.33mm</td>
</tr>
<tr>
<td>Zc</td>
<td>327.27mm</td>
</tr>
<tr>
<td>No. of parts</td>
<td>51</td>
</tr>
</tbody>
</table>

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.

<table>
<thead>
<tr>
<th>Element types</th>
<th>SOLID186, SOLID187, TARGE170, CONTA174</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of nodes</td>
<td>100490</td>
</tr>
<tr>
<td>No. of elements</td>
<td>49597</td>
</tr>
</tbody>
</table>

Table10. Geometry details of Model-IV
6. Results And Discussion

The weights of the models are shown in the table 12. 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.

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

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 figure 7.1 and figure 7.2. The maximum equivalent stress occurred at bottom side of first rib section. The detailed view is as shown below in figure 25.

<table>
<thead>
<tr>
<th>Equivalent Stress</th>
<th>Max (MPa)</th>
<th>Min (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>118.2</td>
<td>0</td>
</tr>
</tbody>
</table>

Detailed view

“Table 12. Comparisons of Mass, total deformation and Equivalent stress values of four models”
The maximum deformation occurred at the top surface of the side sheet and is shown in the figure 7.2. The values are

<table>
<thead>
<tr>
<th>Total deformation</th>
<th>Max(mm)</th>
<th>Min(mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1.8</td>
<td>0</td>
</tr>
</tbody>
</table>

“Figure 26. 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