Size Optimization Approach for achieving optimum design of tractor rear tow hook assembly and its correlation with physical test

Solairaj Perumal  
Lead Engineer  
Mahindra & Mahindra  
Mahindra research Valley  
Mahindra world city,  
Chennai-603 004  
P.SOLAIRAJ@mahindra.com

Kumar Abhay  
Lead Engineer  
Mahindra & Mahindra  
Mahindra research Valley  
Mahindra world city,  
Chennai-603 004  
KUMAR.ABHAY@mahindra.com

Dinesh Redkar  
Principal Engineer  
Mahindra & Mahindra  
Mahindra research Valley  
Mahindra world city,  
Chennai-603 004  
DINESH.REDKAR@mahindra.com

Abstract

This paper focuses on a rear tow hook and mounting bracket which is to be used in Agricultural tractors, is designed to overcome the issues such as Cost, Weight, Assembling time, serviceability and over Material consumption observed in the current tow hooks assembly and few unconventional design approaches for strengthening a Tow hook. In agricultural industry, towing tractors have become an important and constantly growing means of safe and comfortable to transport loads. Agricultural towing tractors can increase productivity and reduce the cost of moving goods and material over long distances. By transporting a linked series of loads, towing tractors offer increased flexibility and efficiency. A tow hook is a device attached to the chassis of a vehicle for towing object and device. The design and development of a tow hook system is governed by strength requirement as per the real world usage pattern and IS-8300-1976.

A finite element model was generated consisting of Tow hook & mounting brackets by using Hypermesh. Static and modal analyses were performed in simulating different loading conditions during haulage applications. Based on the results, Size optimization approach was used to obtain optimum design of the rear tow hook system and finally 34.5% weight saving was achieved with the help of Optistruct tool. The suggested optimized tow hook assembly model met the durability criteria and successfully implemented on one of the farm tractor. No failure was observed during physical validation in lab test and more than 84% of strain correlation was obtained.

Introduction

Tow hooks’ primary function is to withstand towing loads. In agricultural industry transporting heavy disabled trailer, large truck; tractor tow hook and draw bar can be used. One end of the tow hook and its supporting bracket is fixed to rear housing of the tractor, while the other end is attached to the trolley or trailer. A tow hook helps in transporting vehicles, industrial equipment from one place to another. Tow hooks are attached to supporting structure by bolt or weld connections and have to meet certain strength and impact requirements. Bolt strength, clamp load, slippage should be considered in design, if tow hook s are bolted to the attachment structure.

During haulage operation, the tow hook and its mounting bracket is subjected to high stresses to its structure due to the high pull and pushing load applied. Due to the high load applied on the tow hook & its supporting structure can fail during operation.
The current tow hook assembly has issues such as Cost, Assembling time, serviceability and over Material consumption. To overcome these issues the new design proposals of tow hook and mounting brackets were tested iteratively in finite element method. In today’s scenario due to competition in agricultural tractor industry it is essential to have components with lowest cost of design, production, easy to assemble and operation. Design of the component plays an important role; hence the handling and utilization of the equipment should be quicker, easier and user-friendly. Nowadays, the optimization methods developed for engineering problems and integrated in FEM software that are most used are known and divided into size, shape and topology optimization categories [1] & [5]. The size optimization deals with optimization of any definable variable of the problem, for plate, sheet thicknesses and cross section dimensions of beams [2] & [4].

Present work is focused on few unconventional design approaches followed to find out the structural strength of tow hook design. Size optimization method is carried out, to get the optimum size of the tow hook and its supporting brackets.

**Process Methodology**

The procedure followed in order to do the analysis is shown below

- Solid modeling of existing design
- To generate finite element model by using hypermesh software.
- Loads and boundary conditions
- Modal analysis of tow hook system for identifying fundamental natural frequencies
- Linear static analysis for von mises stress distribution at critical zone by using haulage normal and abuse load case.
- Structural Fatigue analysis for predicting minimum life cycle.
- Few unconventional design approaches followed to identify the feasible design
- To obtain optimum design of tow hook system by using size optimization approach
- Finalize the design & re-analysis
- Physical validation and correlation
Finite element modelling

CAD model of existing tow hook system can be generated in any modeling software like Catia, Pro-E and Unigraphics etc., then transferred to finite element method software as shown in Fig.2. The Tow hook system is meshed with commercially available pre-processing software like hypermesh tool.

The Tow system consist of Tow hook plate, side support, rear mounting solid block, L plates and mounting brackets. Tow hook and its supporting bracket is meshed with linear four noded quadrilateral shell elements. Rear mounting solid blocks are meshed with eight noded hexahedral element. All mating part weld is modelled by a row of linear four node quadrilateral shell elements.

In a tow hook System, fasteners are included and bolt head is modelled with rigid element and bolt shank is modelled as a 1D beam element. The rigid element connections simplify the model and analysis extensively. The FE Model of tow hook system is shown in Fig.2. Material used for modal and structural analysis of tow hook system is BSK460. However, in recent years, it is mostly used as material for the tow hook because of and high corrosion resistance. Isotropic material properties are used in the FE analysis.

Results & Discussions

Modal analysis and Results:

Modal analysis is carried out for the existing design considered by meshing the tow hook system and applying the boundary conditions. The boundary condition is set for the model as shown in Fig.3, which indicates that it is fixed in all degrees of freedom at rear housing mounting location.
Once the boundary conditions are applied on tow hook system, constrained modal analysis is carried out to find out the dynamic characteristic like mode shape and natural frequency of the tow hook system as shown in Fig. 4. Normal modal analysis gives modal strain energy at each location of the tow hook system. From modal analysis first fundamental frequency is found greater than ground level frequency. The existing tow hook system, natural frequencies are found by using FEM solver which are listed in the below tables. As per CAE analysis, existing design is meeting the frequency acceptance criteria.

### Table 1: Natural Frequency of an existing Model

<table>
<thead>
<tr>
<th>Mode#</th>
<th>Natural Frequency (w.r.t Acceptance limit) - Existing model</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>23.2 x</td>
</tr>
<tr>
<td>2</td>
<td>39.4 x</td>
</tr>
<tr>
<td>3</td>
<td>40.7 x</td>
</tr>
<tr>
<td>4</td>
<td>61.3 x</td>
</tr>
<tr>
<td>5</td>
<td>68 x</td>
</tr>
</tbody>
</table>

Figure 4: First mode shape and strain energy contour plot of an Existing model tow hook system

**Static analysis and Results:**

During tractor haulage application, four different load cases have been considered for static analysis. The boundary conditions are applied and all degrees of freedom (DOF) at rear housing mounting locations are constrained. The materials considered for the analysis in order to find the stress and displacement at different loading condition is BSK 460.

The Static analysis is performed with constant load applied as body force in pull, push, upward and downward directions. In static analysis, pull load is result of trolley dragged out from ditch or horizontal surface. A horizontal pull load is applied at the center. Push load is a result of extreme braking conditions. Upward and downward loads are the result of the trolley going over a steep or bump under acceleration load. Each of these loads has been applied separately. The design requirement was that the tractor chassis system should survive a tow load equivalent of a fully loaded trolley. From past experience it was observed that this tow load is the worst load case in the field application. Four different worst loading conditions are considered for the static analysis of tow hook system as shown in Fig. 5 & Fig. 6.
The weight of the trolley considered for static analysis is 15 ton (normal) and 30 ton (abuse) [3]. The structural strength of tow hook system is to be carried out by applying pull, push, upward and downward load case. Static analysis is done by taking the existing tow hook model along with the whole assembly.

As per Structural analysis, Max Von mises stresses were observed in the existing tow hook system which are more than the endurance strength of the component as shown in Fig.7 & Fig.8. So that fatigue life calculations were carried out to get the fatigue life cycle of the component. The resulting fatigue life is meeting the test standard.

**Challenges**

As per customer feedback, during tractor field application the following issues were observed.

- Complexity in tow hook system design
- Difficult to access the rear mounting bolts
- Difficult to move tow hook assembly in up and down direction due to overweight with respect to hitch point
- Number of child part has to be reduced so that cost of fabrication for tow hook system reduces
- Need to reduce assembling time with tractor chassis
- To reduce number of fasteners & better serviceability
- To reduce assembly time, parts need to be standardized

Few unconventional design approaches were followed to achieve simple solution in place of complex design as shown in Fig.9. Complex solutions tend to produce more waste and harder for people to manage. It is important to choose the best design shape of tow system and to ensure that each part will be able to withstand the load. The selected tow hook
system should be easy to fabricate. Fabrication plays an important role in design selection. As number of component increases, cost of fabrication increases.

The design of tow hook system is decided on the basis of its rigidity, strength, cost, durability and reliability. The type of design to be used for tow hook system depends on its application requirements and operating conditions. Static and modal analyses were performed in Optistruct simulating different loading conditions during haulage applications.

Initially few unconventional design approaches were carried out and based on the results one design concept was finalized. Once the design concept was finalized, the size optimization dealt with variable thickness of the structure to obtain optimum design of tow hook system. In this study, we used size optimization to reduce weight by reducing the thickness of tow hook system.

**Tow hook assembly Size Optimization**

Nowadays optimization became inevitable part among the virtual validation processes of designs in industries. Eventually all designs which satisfies the acceptance criteria of particular analysis will be undergoing optimization process in order to reduce the material cost. Also, for the design which fails to meet acceptance criteria pertaining to the analysis performed, optimization tool helps in significant levels of design improvement. In optimization process, we are finding the values of design variables which provides an optimal solution for the objective function by satisfying all the constraints.

Tow hook optimization has been performed using Optistruct FE software with ‘Size optimization’ method which is normally used for FE models with shell elements. For instance, in a shell element model of a tow hook system, the shell element thickness on the property assigned to the model would be varied and optimized. As the optimization takes place on the property, changing of individual element thicknesses is not possible, all the elements assigned a property must have uniform thickness. It is possible to set discrete values that represent manufacturable dimensions [1].
The FE model with loads and constraints for size optimization is shown in figure 11. The thickness of shell elements of tow hook FE model is used as design variable. For each plates and welds of tow hook system, separate design variables are assigned. In each optimization step, design variables will be executing the values according to the predefined increments within lower and upper bounds defined for each of the design variables. Volume of tow hook system and Von-mises stress induced are assigned as responses. Minimization of the response, volume of tow hook system, is chosen as objective of the optimization problem. The response, Von-mises stress induced, is considered as constraints for the optimization problem.
Parameters defined during Size Optimization Setup are

**Design Variables** – Thickness of shell elements in design space

**Design Constraints** - Stress < Yield strength

**Objective** - Minimize Volume

Design variables and optimization parameters are defined in Optistruct to determine the optimal thickness for the tow hook system. Optistruct uses only 6 design iterations to reach optimum shell thickness for given conditions.

Based on the optimization results, the thickness of tow hook system is tabulated below. Standard thickness value is chosen from optimization result, to achieve optimum design of tow hook system. Finally **34.5% weight saving** was achieved. Thickness plot of tow hook system is shown in Fig.13.

![Thickness plot of tow hook system](image)

**Table 2: Sizing Optimization Summary**

<table>
<thead>
<tr>
<th>Design Variable ID</th>
<th>Design Variable Label</th>
<th>Lower Bound</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>top_plate</td>
<td>1.000E+01</td>
<td>2.000E+01</td>
</tr>
<tr>
<td>2</td>
<td>plate_10</td>
<td>5.000E+00</td>
<td>1.000E+01</td>
</tr>
<tr>
<td>3</td>
<td>side_sup</td>
<td>5.000E+00</td>
<td>1.000E+01</td>
</tr>
<tr>
<td>4</td>
<td>l_angle_</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>5</td>
<td>side_pla</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>6</td>
<td>bottom_1</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>7</td>
<td>l_angle_</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>8</td>
<td>rib_ext</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>9</td>
<td>c_channel</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>10</td>
<td>Z1</td>
<td>3.000E+00</td>
<td>3.000E+00</td>
</tr>
<tr>
<td>11</td>
<td>Z6</td>
<td>3.000E+00</td>
<td>6.000E+00</td>
</tr>
<tr>
<td>12</td>
<td>Z6_DOU</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>13</td>
<td>Z7_DOU</td>
<td>0.000E+00</td>
<td>1.600E+01</td>
</tr>
</tbody>
</table>

**Table 1: Tow Hook System Description**

<table>
<thead>
<tr>
<th>Tow Hook System</th>
<th>Description</th>
<th>Existing Model</th>
<th>Before Optimization</th>
<th>After Optimization</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tow Hook Bracket</td>
<td>Base Support Plate thickness (mm)</td>
<td>10.0</td>
<td>8.0</td>
<td>4.0</td>
</tr>
<tr>
<td></td>
<td>Side Support Plate thickness (mm)</td>
<td>10.0</td>
<td>8.0</td>
<td>8.0</td>
</tr>
<tr>
<td></td>
<td>Stiffener Plate thickness (mm)</td>
<td>10.0</td>
<td>6.0</td>
<td>3.0</td>
</tr>
<tr>
<td></td>
<td>Support Plate thickness (mm)</td>
<td>8.0</td>
<td>8.0</td>
<td>4.0</td>
</tr>
<tr>
<td></td>
<td>L-Plate thickness (mm)</td>
<td>10.0</td>
<td>6.0</td>
<td>3.0</td>
</tr>
<tr>
<td>Tow Hook</td>
<td>Side Support Plate thickness (mm)</td>
<td>10.0</td>
<td>10.0</td>
<td>6.0</td>
</tr>
<tr>
<td></td>
<td>Tow hook top Plate thickness (mm)</td>
<td>16.0</td>
<td>16.0</td>
<td>16.0</td>
</tr>
<tr>
<td></td>
<td>Tow hook bottom Plate thickness (mm)</td>
<td>16.0</td>
<td>16.0</td>
<td>16.0</td>
</tr>
<tr>
<td></td>
<td>Vertical Support plate thickness (mm)</td>
<td>16.0</td>
<td>10.0</td>
<td>5.0</td>
</tr>
<tr>
<td></td>
<td>Total Mass (kg)</td>
<td>47.71</td>
<td>37.43</td>
<td>31.26</td>
</tr>
</tbody>
</table>

**Figure 13: Result plot of thickness optimization for tow hook system**

**Figure 14: Optimized rear tow hook system**
Optimum Design of Tow Hook System

Optimized tow hook system is remodeled with solid elements for static and modal analysis. Virtual validation is carried out on the tow hook system with optimum thickness. Based on CAE analysis, results of optimized model are within target value as shown in Fig.15.

![Figure 15: Frequency comparison between existing and optimized tow hook system](image)

Physical validation and correlation

Based on the analysis result, the strain gauge location and orientation are identified based on the max principal direction at uniformly distributed region. Uniaxial strain gauge has been used and CAE result is correlated with the test results.

<table>
<thead>
<tr>
<th>Tow Hook Bracket</th>
<th>Vertical Load case</th>
<th>CAE result (microns)</th>
<th>Test result (microns)</th>
<th>% of correlation</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>222</td>
<td>231</td>
<td>96.1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Tow Hook</th>
<th>Pull Load case</th>
<th>CAE result (microns)</th>
<th>Test result (microns)</th>
<th>% of correlation</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>258</td>
<td>217</td>
<td>84.1</td>
</tr>
</tbody>
</table>

Table 3: Correlation between CAE and Test Results

The strain comparison between CAE and lab for pull and vertical load cases are tabulated above. CAE result is correlated with the “National Accreditation Board for Testing and Calibration Laboratories” certified lab test with 84.1 % and 96.1 % of strain correlation for pull and vertical load case respectively.

Benefits Summary

Using size Optimization technique, the weight of the tow hook system was reduced by 34.5% which is significant savings in weight. Savings of tow hook assembly is shown in Table 4.
<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reduction in weight</td>
<td>34.5%</td>
</tr>
<tr>
<td>Reduction in cost</td>
<td>16.8%</td>
</tr>
</tbody>
</table>

**Table 4: Savings of Tow Hook Assembly**

- Huge cost saving was achieved, which is more than Rs.2 crores per annum (based on the tractor projected volume)
- Achieved simple solutions in place of complex ones
- Better serviceability
- Minimize manufactured scrap
- Minimized manufacturing cost
- Moving up and down is easy, due to less weight as per trolley hook position.
- Customer can easily disassemble and then assemble in the field condition.
- Reduced number of bolt joints
- Increased reliability, because of the simplified production process
- Complex solutions tend to produce more waste and are harder for people to manage

**Future Plans**

After size optimization of tractor rear tow hook assembly, we are planning to do the stochastic (Hyper Study) analysis for further improvement in design. After development of this methodology we can deploy this method in other projects to reduce the weight and product design cycle.

**Conclusions**

In this study, initially the tow hook system design are selected based on the customer feedback by using some unconventional design approach. To overcome customer issue, new design has been developed to improve better serviceability, simplified product design for ease of assembly, standardized parts and reduced assembly time.

The rear tow hook system of tractor was analyzed for different loading condition to improve the performance during haulage application. The final design was achieved, once the iteration meets the design requirements and the same design was considered for the optimization process to reduce the weight. In this study, size optimization was used to reduce the weight, by removing an unnecessary thickness while maintaining the sufficient strength.

By this methodology a tow hook system designed, developed and correlated with physical test. No failure was observed during physical validation in test lab. At the end, a cost effective and reliable product has been made with 84.1% and 96.1% of strain correlation between virtual analysis result and physical test result respectively.

In this study, we have achieved weight saving of 34.5% and Overall assembly cost saving of 16.8%. The optimized model design cycle time also reduced when compared with existing model.

**ACKNOWLEDGEMENTS**

The authors would like to acknowledge our Design, Prototype and testing teams for extending their kind support as this work was made possible through rigorous testing and validation. Also, the authors would like to take this opportunity to thank the R&D Head, CAE Head and management of Mahindra & Mahindra tractor division for continuously encouraging and providing this opportunity for publishing the work.
REFERENCES


