ISSN 2395-1621

Combine Harvester Engine Mounting Frame Optimization

#1 Ravikumar Anburaj

[1]ravikumar4026@gmail.com

#1 Alard College of Engineering and Management,
pune, India

M. E. (Design Engineering), Savitribai Phule Pune University

ABSTRACT

Engine mounting frame is an important and critical component in a vehicle system. Here work involves study of Static and dynamics analysis and to determine the key characteristics of a combine harvester Engine mounting frame. It serves the frame work to engine, Cooling Package, pre-cleaner, Exhaust gas processing components and the other parts of the drive line. Hence it should be rigid enough to withstand the shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate bending stiffness, fatigue strength and model stiffness for better handling characteristics. So, strength and stiffness are two important criteria for the design of the chassis. This report is the work performed towards the static structural analysis of the combine harvester chassis. Structural systems like the chassis can be easily analysed using the finite element techniques. So a proper finite element model of the chassis is to be developed. The total capacity of the frame includes cooling package, engine weight and transmission unit which is applied as a point load on the simply supported over hanging beam frame. Design changes and calculations are made to improve the load carrying capacity of the structure. Materials grades and properties has been modified to study as a part of the improvement and to meet the strength requirement. The details will be discussed in the later part of the paper. Here chassis is modelled in CREO. FEA is done on the modelled chassis using the ANSYS Workbench

Keywords- *Engine Mounting Frame, Optimization.*

ARTICLE INFO

Article History

Received :11th Ocober 2015

Received in revised form :

12th October 2015

Accepted : 15thOctober , 2015

Published online :

17th October 2015

I. INTRODUCTION

Engine mounting frame is an important part of the combine harvester. It serves as a structural support and mountings to engine, Cooling Package, pre-cleaner, exhaust gas processing components and the parts of the drive line. Out of these, the frame provides necessary mounting to the Engine accessories which is placed on it. Also the frame should be strong enough to withstand shock, twist, vibrations and other stresses. This frame structure consists of two side members attached with a series of three cross members. Stress analysis carried out using Finite Element Method (FEM) can be used to locate the critical point at which point the frames has the highest stress. This critical point is one of the factors that may cause the fatigue failure. These magnitude of the stress can be used to predict the life span of the harvester chassis frames. The accuracy of prediction life of harvester chassis is depending on the result of its stress analysis.

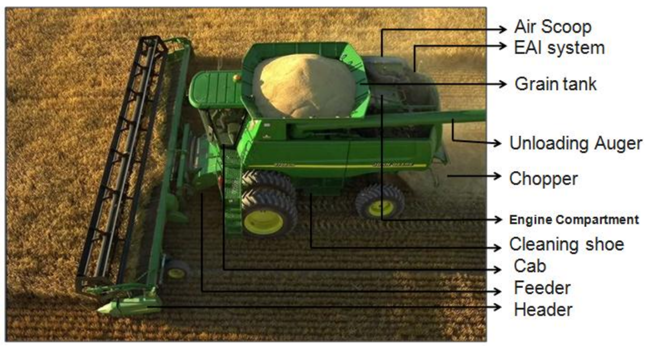

Fig. 1: Functional groups in combine harvester

Here the chassis optimization came to picture due to new configuration of the machine as per the Indian market. It was the de gradation of header from 16 feet to 12 feet header, Since the Indian agricultural lands won't be that much big. So there was a necessary to degrade the configuration of the harvester along with header, feeder, engine, engine mountings and its accessories to meet the energy requirement.

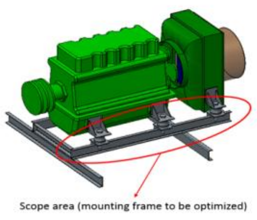

Fig. 2: Frame to be optimized

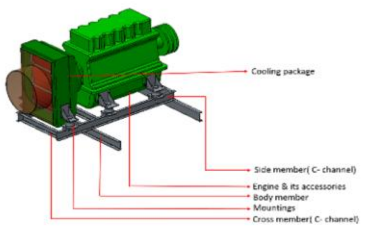

Fig. 3: Functional groups/parts in the assembly

Here our scope of work is to optimize the engine mounting frame, the existing frame chassis having capable to carry 16L engine. But our newly configured engine is just 12L engine which will be having less weight comparatively, so the existing engine frame is over designed for the 12L engine. So the mounting frame design started from scratch to meet the new configuration engine within frames yield limits.

II. LITERATURE SURVEY

Serial No:	1
Author Name:	Abhishek Singh, Vishal Soni, Aditya Singh
Paper Title:	Structural Analysis of Ladder Chassis for Higher Strength
Journal Name	International Journal of Emerging Technology and Advanced Engineering
Year	February 2014
Conclusion	The simulation analysis results have established approving outcome in term of factor of safety at different loading tests. The final products for both designs have been fabricated successfully and verified the practice of design for manufacture (DFM). The fitting and assembly of the rolling chassis have verified the accuracy and acceptable tolerance in design and fabrication.

Serial No:	2
Author Name:	Vishal Francis, Rajnish Kumar Rai, Anup Kumar Singh, Pratyush Kumar Singh, Himanshu Yadav
Paper Title:	Structural Analysis of Ladder Chassis Frame for Jeep Using Ansys
Journal Name	International Of Modern Engineering Research (IJMER)
Year	Apr. 2014
Conclusion	The highest stress occurred is 106.08 MPa by FE analysis. The calculated maximum shear stress is 95. 43 Mpa. The result of FE analysis is bigger 10 % than the result of analytical calculation. The maximum displacement of numerical simulation result is 3.0294 mm. The result of numerical simulation is bigger 5.92 % than the result of analytical calculation which is 2.85 mm. The difference is caused by simplification of model and uncertainties of numerical calculation.

Serial No:	3
Author Name:	Tushar M. Patel1, Dr. M. G. Bhatt2 and Harshad K. Patel3
Paper Title:	Analysis and validation of Eicher 11.10 chassisframe using Ansys
Journal Name	International Journal of Emerging Trends & Technology
Year	April 2013

Conclusion	A quick overview on structural optimization methods has been given including various application examples. Their potential has been shown to be large and it is believed that their spreading in mechanical design could boost innovation in industry considerably. Examples in the automotive have been provided. To be noted that the deferent methods have different characteristics and in a design process it is recommended to rely on more than just one technique. For instance, topology and tonometry optimizations are more suitable for an early development stage, whose outcome could be further reined through size and shape optimizations. On a general basis these techniques do not deliver the shape of the final product, but they give useful hints to the designer in view of the product development and engineering.
Serial No:	4
Author Name:	Swami K.I.1, Prof. Tuljapure S.B.2
Paper Title:	Analysis of Ladder Chassis of Eicher 20.16 Using
Journal Name	IOSR Journal of Applied Geology and Geophysics (IOSR-JAGG)
Year	Jan. 2014
Conclusion	The objective of the analysis had successfully achieved in the area of improvement of torsion stiffness based on the result gained from the finite element analysis, further enhancement of the current chassis had been done through the chassis FE model in order to improve its tensional stiffness as well as reduce the vibration level. Series of modifications and tests were conducted by adding the stiffener in order to strengthen and improved the chassis stiffness as well as the overall chassis performances developed.

III. PROBLEM STATEMENT AND OBJECTIVE

3.1 Problem Statement:

Perform strength analysis and optimization of engine mounting frame for the combine harvester 12L Engine.

3.2 Objective:

The objective is to design a engine mounting frame with good structural strength. The design variable is cross section of the c-channels

IV. METHODOLOGY

4.1 Design of Experiments (DOE)

The Design of experiments (DOE) is a systematic, rigorous approach to engineering problem-solving that applies principles and techniques at the data collection stage so as to ensure the generation of valid, defensible, and supportable engineering conclusions.

Components of Experimental Design:

- Factors or inputs to the process. Factors can be classified as either controllable or uncontrollable variables.
- Levels or settings of each factor in the study.
- Response or output of the experiment.

A well-designed experiment is as simple as possible - obtaining the required information in a cost effective and reproducible manner.

4.2 Multi-Factor Experiments

Multi-factor experiments are designed to evaluate multiple factors set at multiple levels. One approach is called a Full Factorial experiment, in which each factor is tested at each level in every possible combination with the other factors and their levels. The advantage is that all paired interactions can be studied. However, the number of runs goes up exponentially as additional factors are added.

4.3 Designing an Experiment

The design of an experiment involves the following steps

1. Selection of independent variables
2. Selection of number of level settings for each independent variable
3. Selection of orthogonal array
4. Assigning the independent variables to each column
5. Conducting the experiments
6. Analyzing the data
7. Inference

4.4 Process Flow Chart

The Process for the frame optimization is carried out by using below mentioned process flow chart steps.

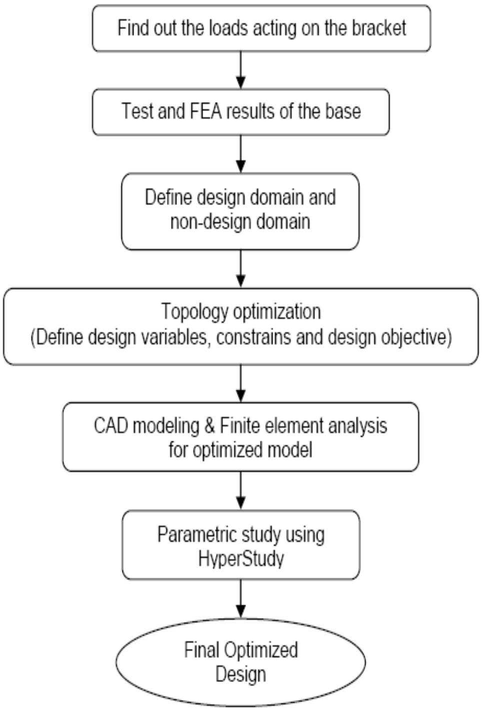

Fig. 4: Process flow chart

4.5 Experimental Setup Overview:

Here the engine, gearbox and cooling package are the loads acting on the chassis, they converted in to point loads. Since the engine is mounted by 4 points on the chassis,the loads are mentioned here as point load 1,2,4&5.
The cooling package is mounted by 2 points at the overhanging portion of the frame,so these loads are mentioned as point load 3&6. The above discussed loads are detailed in below figure.

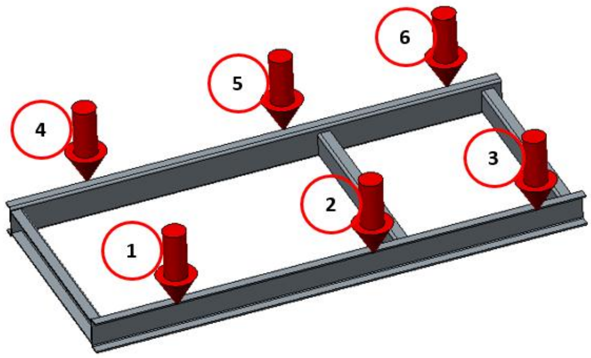

Fig. 5: Converted point load representation

The Load case set up used in FEA simulation used to determine the high stress concentration area; the two rails are supported at bottom two points, by cleaning shoe frames. The top side of the rail is loaded by Engine and cooling package at 6 points in both the rails. This setup will look like a simply supported overhanging beam.

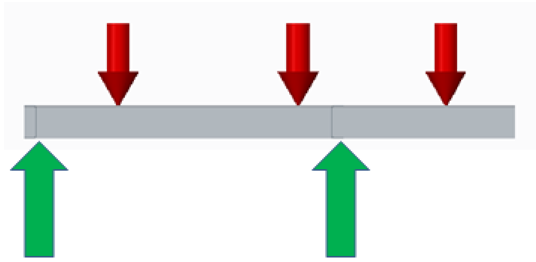

Fig. 6: FEA load case setup overview

4.6 Software Details

Activities	Software Name	Version
Modelling	Creo	2.0
Pre-Processor, Solver and Post-Processor	Ansys	12.0

4.7 PARAMETER SELECTION:

1) 4.7.1 Material Selection

Listed below are the details of the materials assigned to the frame rails and cross member

Properties	Values1	Values2	Values3
Material Name	ST 52	AISI 1080	AISI 1060
Young's Modulus, GPa	210	240	250
Poisson's ratio	0.33	.33	0.28
Density, kg/m3	2700	7860	7860
Yield Strength, MPa	262	280	310

2) 4.7.2 Loads Magnitude

Listed below are the details of prescribed Loads given to the Frame

Load Magnitude		
Load at 1	Load at 2	Load at 6
1.2KN	1.2KN	0.7KN

3) 4.7.3 Location of Load

Three loading positions are considered in simply supported overhanging beam, as per the mounting positions of the engine and cooling package. They are listed below.

Location of Point Loads 1,2&6		
1	2	6
288 mm	838 mm	1288 mm

4) 4.7.4 Cross-Section details

In this evaluation two different cross-sections of the Rails are considered. The first on is the existing cross section of the harvester C110, and the second one is the newly configured cross section for the W 70 combine harvester machine.

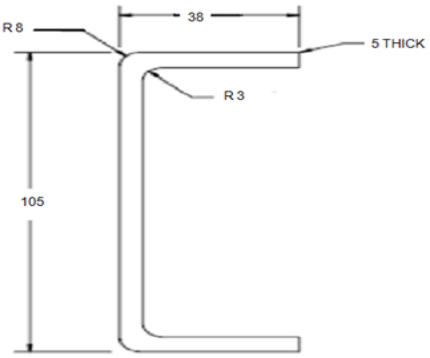

Fig. 7: Cross section details of chassis frame

V. DETAILS OF FEA ANALYSIS

5.1. FEA analysis procedure

Listed below are the steps in FEA

1. CAD Modelling: As per the requirements CAD models are prepared using ANSA Topology options.
2. Pre-processing Steps:
   a. Meshing: The entire CAD surfaced is meshed on the mid-surface with Quadrilateral elements.
   b. Thickness Assignment: Corresponding thickness properties are assigned to all parts
   c. Material Assignment: For every Load case appropriate material properties are assigned.
   d. Boundary Condition:
      i. Two cylinder supports: They are rigidly fixed by giving constraints in all 6 degrees of freedom.
      ii. The punch: It is fully constrained for 5 dof. It is given prescribed displaced in negative Z-direction. The value of displacement varies as per load case.
   e. Contact Definition:
      i. Tied Contact: The chassis cross member beam is connected to the main supporting rail by tied contact, to simulate fastener.
      ii. Surface to Surface Contact is defined between 'Chassis frame and the support frame'
3. Solution: FEA runs will be performed using ANSYS
4. Post Processing: The available results after completion of ANSYS run are post-processed using Meta-Post.

5.2. Expected outcome:

1. Maximum Strength σ_{Max}
2. Minimum Deflection Δ_{Min}
3. Minimum Weight W_{Min}

5.3. Mathematical modelling

A mathematical model is a description of a system using mathematical concepts and language. A mathematical model is an abstract model that used mathematical language to describe the behavior of system. Mathematical models can take many forms, including but not limited to dynamical systems, statistical models, differential equations or game theoretic models. These and other types of models can overlap, with a given model involving a variety of abstract structures.

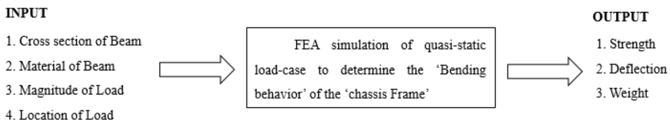

Fig. 8: Experimental model with parameters

VI.PROJECT PLAN/SCHEDULE

S No	Month	Work Done	Out Come	Remarks
1	September & October 14	Literature survey	Topic selection	Completed
2	November & December 14	Discussion on title and Synopsis preparation	Synopsis	Completed
3	January & February 15	Selection of Software Tools and Procedure	CAD and FEA tools selected	Completed
4	March & April 15	Learning of new Tools	Proficiency in tools	Completed
5	May & June 15	CAD Modelling and Analysis	FEA results	Work in progress
6	July 15	Cad Modelling and Analysis	FEA results	To be done
7	August 15	Results and Discussion, Report Rough copy	Report rough copy	To be done
8	September 15	Report writing and paper presentation	Final Report and Paper presentation	To be done
9	December 15	Report writing and paper presentation	Final Report and Paper presentation	To be done

VII.COST ESTIMATION DETAILS

1. Cost of Analysis: Rs x
2. Cost of printing and binding of report: Rs y
3. Cost for paper publication of report: Rs z

REFERENCES

[1]Abhishek Singh1, Vishal Soni2, Aditya Singh3 (2014). Structural Analysis of Ladder Chassis for Higher Strength.
[2]Vishal Francis1, Rajnish Kumar Rai2, Anup Kumar Singh3, (2014). Structural Analysis of Ladder Chassis Frame for Jeep.
[3]Tushar M. Patel1, Dr. M. G. Bhatt2 and Harshad K. Patel3 (2013) Analysis and validation of Eicher 11.10 chassis frame.
[4]Swami K.I.1, Prof. Tuljapure S.B.2 (2014).Analysis of Ladder Chassis of Eicher 20.16 Using FEM
[5]"Safety Companion 2014" from 'www.carhs.de'.
[6]"Pole Side Impact, test explained" from 'www.euroncap.com'
[7]Creo 2.0 user's Guide.
[8]Ansys 12.0 user's manual

