ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 5, Special Issue 3, January 2018

PARAMETRIC DESIGN AND OPTIMIZATION OF TRAILER'S PULLER ARM USING FEA

SRINIVAS PRABHU M
Assistant Professor, Department of Mechanical Engineering,
NMAMIT, Nitte-574110, Udupi, Karnataka, India
srikles@nitte.edu.in

NITHIN NARAYAN
Scholar, Department of Mechanical Engineering,
NMAMIT, Nitte-574110, Udupi, Karnataka, India

ABSTRACT

This paper deals with the parametric design and optimization of trailer's puller arm. The main objective of this paper is to save material and hence to obtain cost reduction, all the while maintaining the operating stress under the safe stress limit for the material being used. The objectives of the project are fulfilled by parametric CAD design, force calculation, material selection, Finite Element Analysis (FEA), design optimization of trailer's puller arm using design and analysis iterations. The following methodology is adopted in the project. Parametric modelling of trailer's puller arm structure followed by the manipulation of the overall dimensions of the design with little effort. The modelled geometry is discretized using the appropriate finite number of elements. The differential equations generated out of these elements is solved and assembled. The result will be captured in finite element solver and the post processor. Iterative solution of the problem is obtained and will be compared with each other. The design will be finalised based on the results and the optimized design is proposed

Keywords— Parametric CAD Design, Design Optimization, Finite Element Analysis, Trailer's Puller Arm, Iterative solutions

I. INTRODUCTION

Currently industries are working on optimising the design so as to reduce the cost of the product, to be competitive in the market. In the view of this, the mechanical engineering industries are focusing towards certain structural members which are not engineered properly, means they are either under designed or over designed. This happens due to the thought that, more the wall thickness of the member the more will be the strength more the cross section area more robust will be the design [1]

In this project an attempt has been made to design and optimize the Trailer's Puller Arm. It involves modelling with the help of parametric CAD design, Force calculation, Material selection, Finite Element Analysis (FEM), Design optimization using design tools and analysis iteration. A Trailer is an unpowered vehicle design for connection to a tractor, Truck or Prime mover vehicle with strong traction power. Puller arm will be connected to the front end of the trailer, which is used to connect the trailer to the powered vehicle to pull the loads. The parametric features-based modelling revolutionized the CAD industry. It fundamentally changed the way that engineering organizations not only developed 3D models, but also how they made changed to the designs as well. By adopting a parametric approach to creating models, engineers are also setting the clear parameters, features and relationships of the models, which is intended to capture the product's behavior. Parametric features-based CAD tools provide a lot of power and help to automate some of the engineering processes involved in product development. Parametric design is defined as any set of physical properties whose value determine the characteristics or behavior of something. It enables you to generate a variety of information about your design-mass, properties, a drawing or a base model.[2] In optimization of a design, the design objective could be simply to minimize the cost of production or to maximize the efficiency of production. An optimization algorithm is a procedure which is executed iteratively by comparing various solutions till an optimum or a satisfactory solution is found. With the advent of computers, optimization has become a part of computer-aided design activities. Optimization of a design is the process by which an objective function is defined with respect to several fixed parameters along with other design variables, and re-evaluated to improve the target value of the function. There are many different ways to optimize and also different range

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 5, Special Issue 3, January 2018

of optimization processes such as one, two or multiple parameter optimization. Optimization is possible in any aspect of mechanical design if one introduces stringent criteria by which the design must satisfy its functions and can then be developed with respect to what is expected from it. This introduces the concept of "needs analysis", which summarizes the needs to be met by the design being developed.[3]

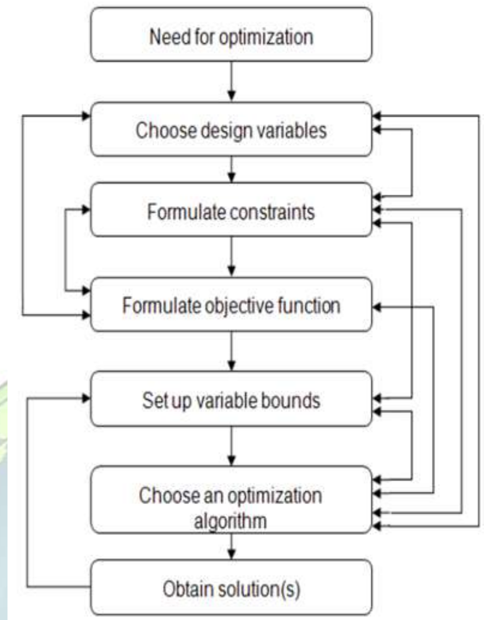

Fig 1. Flow chart of optimization process [4]

II. FINITE ELEMENT METHOD

The Finite Element Method is a numerical method for solving problems of engineering and mathematical physics where their behaviour/governing equations are expressed by integral or differential equations. In actual practice, engineering problems involve complicated geometries (continuums), loadings and varying material properties. Due to this, it may be impossible to specify the boundary conditions, consider material properties and solve the governing differential equation. In such a situation, we go for numerical methods such as "Finite Element Method" (FEM) to get approximate but acceptable solutions. The basic concept is that a body or structure may be divided into smaller elements of finite dimensions called 'finite elements'. The original body or structure is considered as an assemblage of these elements connected at a finite number of joints called 'Nodes' or 'Nodal points'. The concept of discretisation used in finite difference method is adopted here.

III.COMPUTER AIDED MODELLING

The software used for modelling is Creo Parametric 2.0.PTC Creo is a scalable, interoperable suite of product design software that delivers fast time to value. It helps teams create, analyse, view and influence product designs downstream utilizing 2D CAD, 3D CAD, parametric & direct modelling.
PTC Creo, formerly known as Pro/ENGINEER is a parametric, integrated 3D CAD/CAM/CAE solution created by Parametric Technology Corporation (PTC). The application runs on Microsoft Windows platform, and provides solid modelling, assembly modelling and drafting, finite element analysis, direct and parametric modelling, sub-divisional surfacing, and tooling functionality for mechanical engineers [5]

A. *Dimensions*

These are the initial dimensions based on which the trailer's puller arm is modelled [6]

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 5, Special Issue 3, January 2018

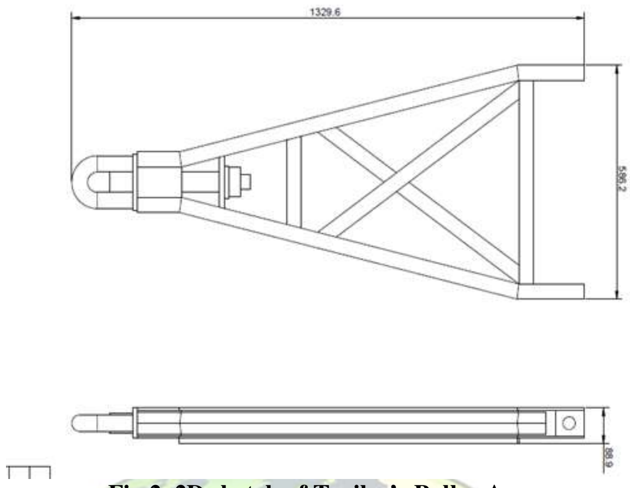

Fig 2. 2D sketch of Trailer's Puller Arm

B. Model

The trailer arm is modelled using the basic commands .The finished model is as follows
For the analysis we have considered axial symmetry

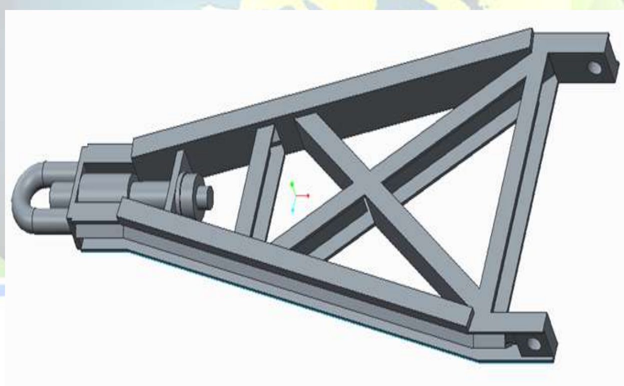

Fig 3. Isometric view of the designed model

IV. FINITE ELEMENT MODELLING

HyperMesh is a high-performance finite element pre- and post-processor for major finite element solvers, which allows engineers to analyse design conditions in a highly interactive and visual environment. Advanced automation tools within HyperMesh allow users to optimize meshes from a set of quality criteria, change existing meshes through morphing, and generate mid-surfaces from models of varying thickness.
HyperMesh enables engineers to receive high quality meshes with maximum accuracy in the shortest time possible. A complete set of geometry editing tools helps to efficiently prepare CAD models for the meshing process.

A.Element type

The element type used here is shell 181.After completing the meshing in Hypermesh, the element type is defined as shell 181 and then it is imported into ANSYS for analysis.Shell 181 is suitable for analysing thin to moderately thick shell structures. It is 4 noded element with six degrees of freedom at each node: translation in the x,y,z directions and rotations about the x,y and z axes.

B.Material properties

STEEL

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 5, Special Issue 3, January 2018

Steels are alloys of iron and carbon, widely used in construction and other applications because of their high tensile strengths and low costs. Carbon, other elements within the iron act as hardening agents that prevent movement of dislocations that otherwise occur in the crystal lattice of the iron atoms.

A36 STEEL
A36 Steel is a standard steel alloy that is common structural steel in the United States. The A36 standard was established by the standards organization ASTM International (American Society for Testing and Materials).It is a low carbon steel with good machinability.

Density	**7800 kg/m^3**
Young's Modulus	200 GPa
Poisson's Ratio	0.26
Shear Modulus	75 GPa
Yield Strength	250 MPa

Table 1. A36 Steel properties

C. MESHED MODEL

Fig 4.Isometric Meshed View

Since the force applied on the solid spline part will be equally distributed on the rectangular face plate and the difficulties occurred while selecting the element type for the solids, the solid parts are not considered.

V. ANALYSIS

ANSYS software is used for analysing the meshed model. ANSYS is a computer aided engineering tool which can be used to perform finite element based analysis [8]. For this analysis ANSYS 11 software is used.

A. CALCULATIONS

ASSUMPTIONS

Total Load Applied	**5 tonne**
Material	A36 Steel Alloy
Wheel Diameter	8 inches
Factor of Safety	1.5
Coefficient of friction	0.04

TABLE 2. Assumptions

Step 1: Based on given load we can determine the load per wheel
W=5000/4=1250 N per wheel

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 5, Special Issue 3, January 2018

Step 2: Radius of the wheel R=4 inches =0.1016m
Step 3: Force to overcome rolling friction
F=μN/R where
N=1250*9.81=12262.5N
i.e. F=(0.04*12262.5) / 0.1016 =4827.75 N per wheel
Step 4: So for 4 wheels, the total force required to overcome rolling friction =4827.75*4
=19311.02N
Step 5: Number of nodes on the rectangular face is found to be 230, therefore force on each node= 19311.02/230=83.96 N

B.ANSYS SOLUTIONS

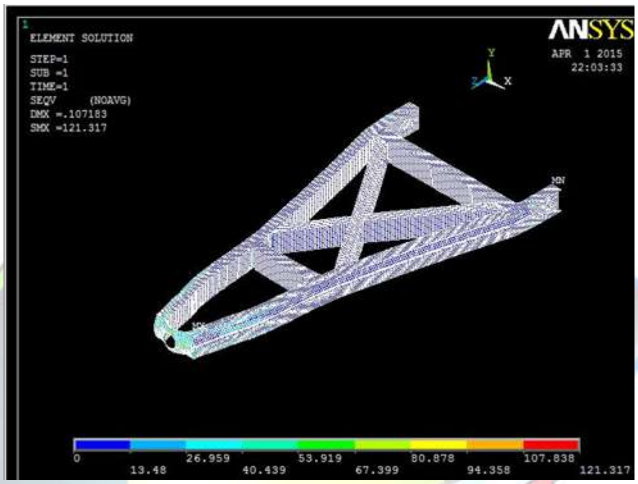

Fig 5.Analysis of Model with thickness 6mm

Displacement	*0.107183*
Max Stress	121.37 N/mm^2

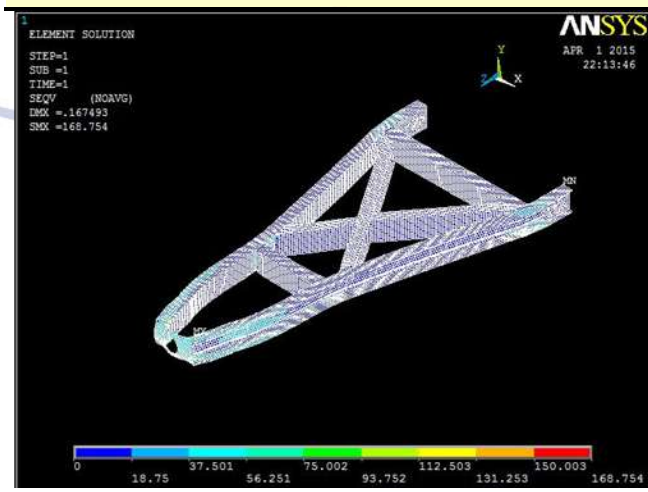

Fig 6.Analysis of Model with the thickness 3.2mm

Displacement	*0.167493*
Max Stress	168.754 N/mm^2

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 5, Special Issue 3, January 2018

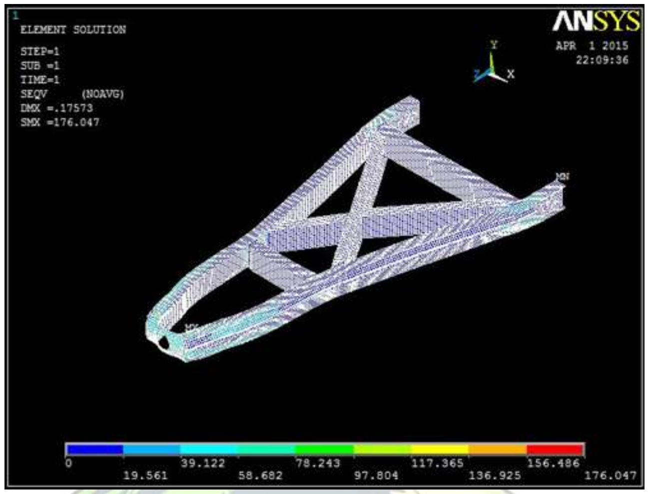

Fig 7.Analysis of Model with the thickness 3 mm

Displacement	***0.17573***
Max Stress	176.067 N/mm^2

VI RESULTS AND CONCLUSION

A. ITERATIONS

For the C cross section

Thickness	Stress	Displacement	Safe Stress
3mm	176.067N/mm^2	0.17573 mm	170 MPa
3.2	168.754	0.167493	170
3.4	162.55	0.160204	170
5	132.787	0.121799	170
6	121.317	0.107183	170

B. CONCLUSION

When the thickness is maximum i.e. 6mm,the maximum stress is obtained is 121 N/mm^2,which is much lesser than the allowable stress value of 170 N/mm^2.From the table we can observe a gradual increase in the maximum stress value when the thickness is being gradually reduced. When the thickness reaches a value of 3 mm, the maximum stress value is higher than the allowable stress limit. A maximum stress value of 168 N/m^2 has been obtained when the thickness is 3.2 mm. Therefore the design is optimized when the thickness is 3.2mm and while keeping in mind that there is further scope for optimization.

VII REFERENCES

1. Project report on Design Analysis of Hydrodynamic Journal Bearing, N.M.A.M. Institute of Technology, Nitte June 2007
2. Concurrent Engineering Blog: An Introduction to parametric design.
3. Systems engineering written by Hendrick. W. Bode
4. International Journal on design optimization
5. www.ptc.com
6 .Cadtech Engineering Pvt. Ltd
7. Ansys Help Menu
8.Project Report on Design and Analysis of Connecting Rod, N.M.A.M Institute of Technology Nitte, June 2002