Structural Analysis and Optimisation of Industrial Crane Arm

Amarnath G.
Associate Professor,
Dept. of ME,
R.R. Institute of Technology,
Bangalore, India.

Shridharmurthy H. N.
Associate Professor,
Dept. of ME,
R. R. Institute of Technology,
Bangalore, India

ABSTRACT

A crane is a type of machine, generally equipped with a hoist, wire ropes or chainsthat can be used both to lift and lower materials and to move them horizontally. Aim is to reduce the weight of the crane by reducing the thickness at few areas where ever applicable and also maintaining the weight carrying capacity.Structural analysis is the determination of the effects of loads on physical structures and their components. Structures subject to this type of analysis include all that must withstand loads, such as buildings, bridges, vehicles, machinery, furniture, attire, soil strata, prostheses and biological tissue. Structural analysis incorporates the fields of applied mechanics, materials science and applied mathematics to compute a structure's deformations, internal forces, stresses, support reactions, accelerations, and stability. The results of the analysis are used to verify a structure's fitness for use, often saving physical tests. "Structural analysis " will be carried out to find the following.

1. *Mechanical response of the structure, subjected to applied loads and boundary conditions.*
2. *Analysis results will include deformed shape of the structure, displacements and the stresses in various locations.*
3. *Following Pre checks are done before Boundary Conditions and loads are applied to the model*
   i. *Elimination of Free Edges and T-connections in the model*
   ii. *Minimizing Triangular elements.*
   iii. *Assignment of material and physical properties to the respective components created*
4. *Optimize to reduce weight and stress concentration on the ARM and displacements due to weight.*
5. *The Final design is achieved by removing the unwanted material and by reducing the thicknesses were ever applicable from the Basic design*

Keywords: Stresses, Displacement, Optimization, Weight reduction

INTRODUCTION:

A mobile crane is "a cable-controlled crane mounted on crawlers or rubber-tired carriers" or "a hydraulic-powered crane with a telescoping boom mounted on truck-type carriers or as self-propelled models." They are designed to easily transport to a site and use with different types of load and cargo with little or no setup or assembly[1] .

The crane for lifting heavy loads was invented by the Ancient Greeks in the late 6th century BC.[2] The archaeological record shows that no later than c.515 BC distinctive cuttings for both lifting tongs and lewis irons begin to appear on stone blocks of Greek temples.

Structural analysis is the determination of the effects of loads on physical structures and their components. Structures subject to this type of analysis include all that must withstand loads, such as buildings, bridges, vehicles, machinery, furniture, attire, soil strata, prostheses and biological tissue. Structural analysis incorporates the fields of applied mechanics, materials science and applied mathematics to compute a structure's deformations, internal forces, stresses, support reactions, accelerations, and stability.[3] The results of the analysis are used to verify a structure's fitness for use, often saving physical tests. Network analysis methods are essentially management control techniques and were developed during the years 1957 and 1958, the foundations of game theory were laid by von Neumann in 1928 and since then it has been applied to solve several mathematical economics and military problems. Only during

the last few years, game theory has been applied to solve some of the engineering design problems.
Morris mentioned that the general design process encompasses both selecting a suitable design concept and in the final stage, completing a detailed design study which fixes the size, materials, layout, etc, of the actual product. He said the design concept mentioned leads to the examination of a range of models at the project stage with the judgments made concerning the feasible options. So the Optimization methods are concern primarily with the horning down of the selected model to its most efficient and cost efficient and cost effective form.

EXPERIMENTAL WORK:

Basic Steps in Finite Element Method(FEM):
The finite element method is a numerical analysis technique used by engineers, scientist and mathematicians to obtain solution to the differential equations that describe a wide variety of physical and non-physical problems.[6]

1. Discretisation of the continuum: The first step in FEM is to divide the given continuum into smaller regions of finite dimensions called as"finiteelements" The, size, numbers and arrangement of the elements depend on the accuracy of the solution required.
2. Selection of approximating functions: Approximation function are also known as the displacement function. Displacement function is the starting point of the mathematical analysis. This represents the variation of displacement within the element.
3. Formation of the element stiffness matrix: After continuum is discretised with desired element shapes,the individual element stiffness matrix is formulated. Basically it is a minimization procedure.
4. Formation of overall stiffness matrix: After the element stiffness matrix in global co-ordinates are formed, they are assembled to form the overall stiffness matrix. The assembly is done through the noded which are common to adjacent elements. The overall stiffness matrix is symmetric.
5. Formation of the element loading matrix: The loading forms an essential parameter in any structural engineering problem. The loading inside an element is transferred at the nodal points and consistent element matrix is formed.
6. Formation of the overall loading matrix: Like the overall stiffness matrix, the element loading matrices are assembled to form the overall loading matrix. This matrix has one column per loading case and it is either a column vector or a rectangular matrix depending on the number of loading cases.[8]
7. Formation of the overall equilibrium equation: Overall equilibrium equation is the systematic arrangement of the overall stiffness matrix,overall vector and overall displacement vector to get set of simultaneous equation.Overall expression can be expressed as shown below;

$[k]\{Q\}=\{F\}$
Where;
$[K]$is a overall or global stiffness matrix
$\{Q\}$ is a overall or global force displacement vector
$\{F\}$ is a overall or global force vector

8. Incorporation of boundary conditions: The boundary restraint condition are to be imposed in stiffness matrix to avoid the condition of singularity. The solution cannot be obtained unless support condition are included in the stiffness matrix. A solution cannot be achieved until the boundary condition i.e., the known displacement are introduced.
9. Calculation of unknown nodal displacements: After incorporation of boundary conditions. Elimination or penalty method of handling boundary condition are used to calculate unknown nodal displacements.
10. Calculation of strain and stresses: Nodal displacements are utilized for the calculation of strain and stresses using the suitable equations. This may be done for all elements of the continuum or it may be limited to some predetermined elements

Computer Aided Designing using CATIA:
CATIA-Introduction:
CATIA(Computer Aided Three Dimensional Interactive Application) is a PLM/CAD/CAM/CAE commercial software suite developed by Dassault system and marketed world-wide by IBM.
Commonly referred to as a 3D PLM(Product Lifecycle Management) software suite,CATIA supports multiple stages of product development.
The stages range from conceptualization, through design(CAD) and manufacturing(CAM) and manufacturing (CAM) and manufacturing(CAM),until analysis(CAE).CATIA provides open development architecture through

the use of interfaces, which can be used to customize or develop applications.
The application programming interface supported are as follows:

1) The Fortran and C programming languages for version 4 (V4).
2) The Visual Basic and C++ programming languages for version 5(V5)

These APIs are referred to as CAA2(or CAA V5).The CAA2 are component object model(COM) Based interfaces. They Provide integration for products developed on the CATIA suite of software.
Although later versions of CATIA V4 implemented NURBS,vrsion 4 principally used piecewise polynomial surface. CATIA V4 uses a non-manifold solid engine.
Catia V5 features a parametric solid/surface-based package which uses NURBS as the core surface representation. CATIAiswidely used throughout the engineering industry, especially in the automotive and aerospace sectors. In this industry CATIA V4,CATIA V5,Pro/Engineer , and unigraphics's NX range of product are the dominant systems.
Dassault Systems has expanded its reach into the shipbuilding Domain with CATIA V5 release 8, which includes additional functionality serving ship builder needs.
The Boeing Company used CATIA V5 for the Boeing 787 series aircraft. They have employed the full range of DassaultSystemes 3D PLM products, comprising of CATIA,DELMIA , and ENOVIA.Electric Boat used CATIA to design the latest United States fast attack submarine, the USS Virginia(SSN-774).
Today CATIA is widely used throughout the world in the aerospace, automotive and maritime industries. Owing to this widespread popularity and reliability of the Dassault Systems CATIA software(Release 5 version 14 for Microsoft windows operation system),it has been used by us for the Computer Aided Designing portion of our project.
The latest release is V5 release 16(V5R16)

Crane 2D Diagrams:

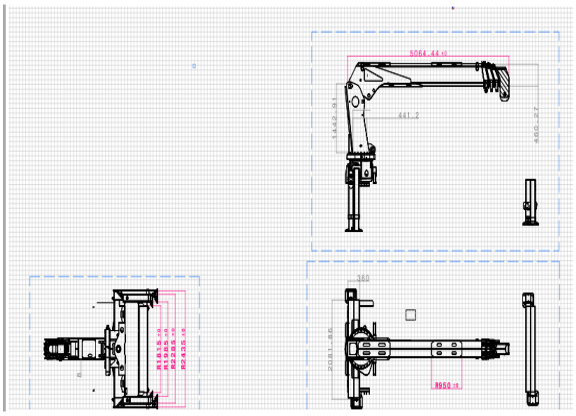

Fig.3.1 Crane Different View

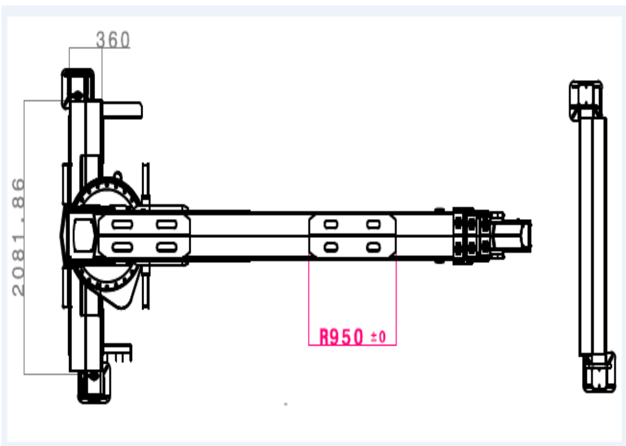

Fig 3.2 Crane top view

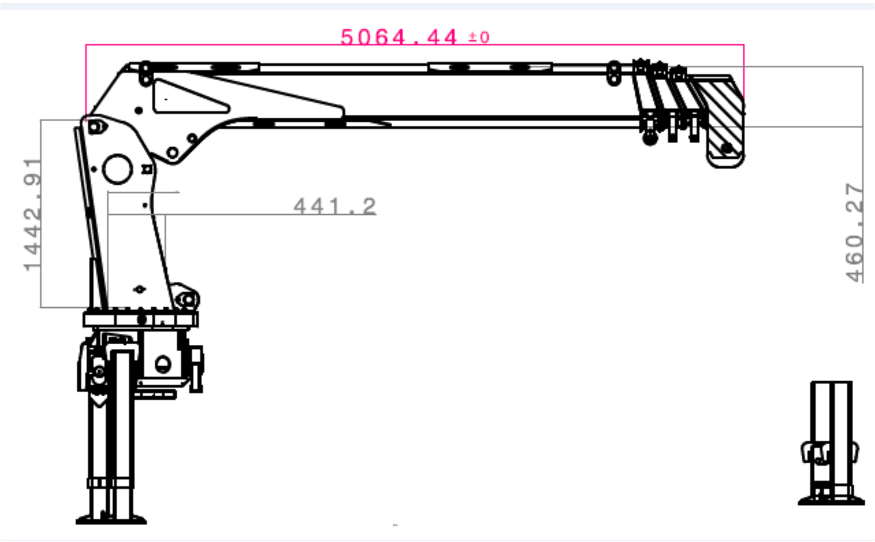

Fig 3.3 Crane side view

Problem Definition:

Definition:

The crane arm is mounted on a vehicle, thus its weight becomes an important factor in working.An lesser weight crane but same weight carrying capacity would be of great advantage and also considering the fact that lesser weight also means reduction in raw material which will in turn reduce the total cost of the product

Modeling, theoretical checking and finite element analysis of the crane arm for heavy operations is the main definition of the problem. The objectives include Analyzing the Original design Re-modeling to reduce weight. Finite element modeling of crane arm.

Design checks:

Analyzing foroptimized model.

Modal analysis to check the resonant condition

Line Diagram

(Basic Design)

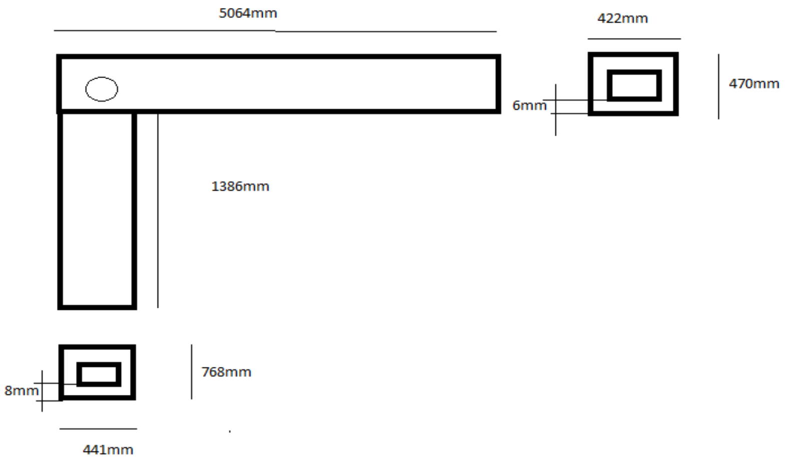

Fig 4.1– Line DiagramBasic Design
Dimensions are in mm

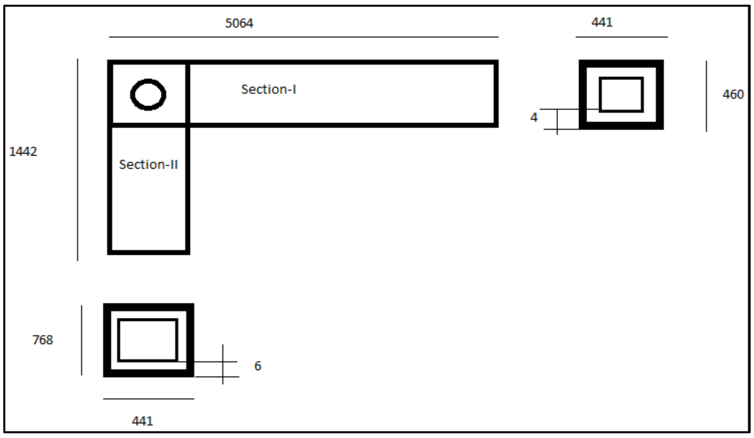

Fig 4.2– Line Diagram New Design
All Dimensions are in mm

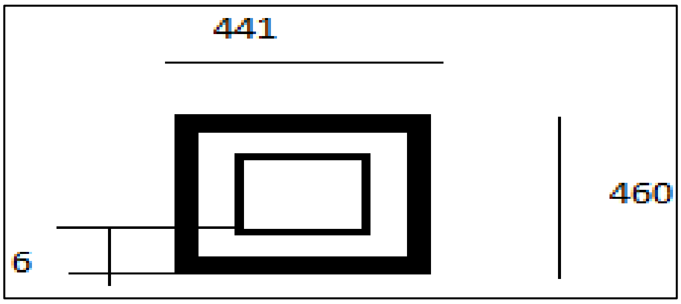

Figure 4.4 Box Section
All Dimensions are in mm

$$I = \frac{B4^3 - bh^3}{12} = \frac{441 \times 460^3}{12} - \frac{429 \times 448^3}{12}$$

$$= 362.6\text{x}10^6 \text{mm}$$

Design Stress

$$= \frac{M\text{ x}Y}{I} = \frac{99254.4 \text{ X } 103 \text{ X } 230}{362.6 \text{ x } 106} = 62.9 \text{ Mpa}$$

$$\boxed{= 62.95 \text{ Mpa}}$$

Deflection (d)

$$d = (Pl^3)/3EI = 11.4$$

Analysis Solution:
Mass Calculation:

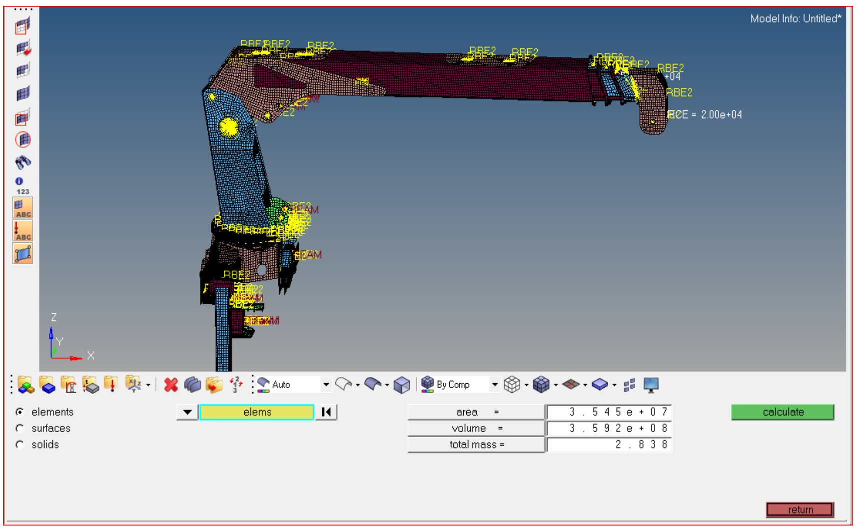

Fig 5.1 Weight of Basic Design

The total mass of basic design is calculated using mass calculator by selecting all the displayed elements in the model which accounts to 2.838 tons or 2838 kgs.
This is the major target of my project is the weight,where I should concentrate to reduce the above said weight.

Self weight Analysis of the Model:

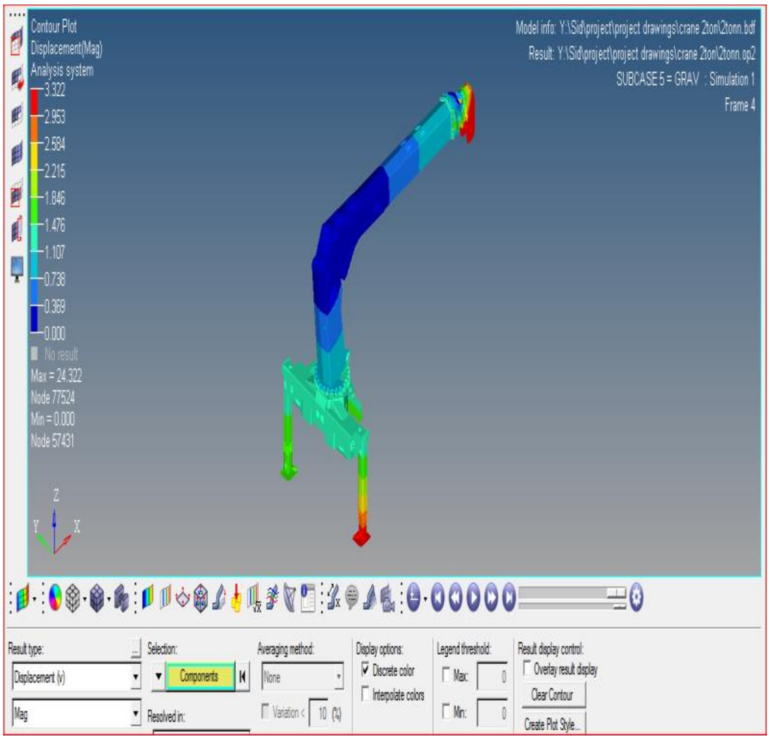

Fig 5.2 Self weight analysis

Most of the times self weight plays an important role in the structural problems. Many mechanical structures fails by its own weight. So a structural check is done to find the stresses and deformation for the self-weightof the model.
The first analysis to be done after meshing is to check if the model is stable or not for its self-weight. Here the maximum deflection is 3.322mm which is under allowable limit for the model.

Modal Analysis:
Mode 1:

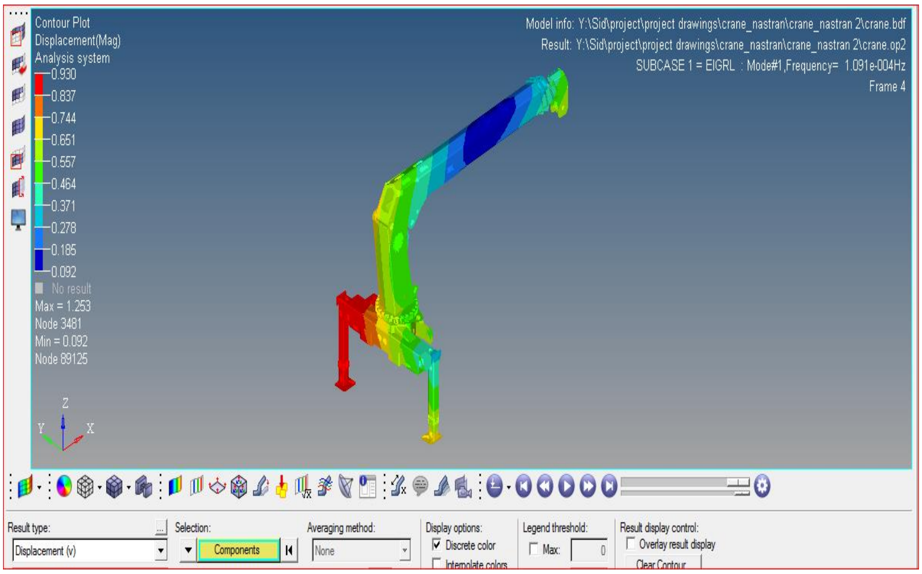

Fig 5.3 Mode1

This is the mode shape for 1stmode.The above figure shows displacement for mode 1 i.e., frequency of 109.1*10-6.Here the displacement is 0.930mm

Mode 2:

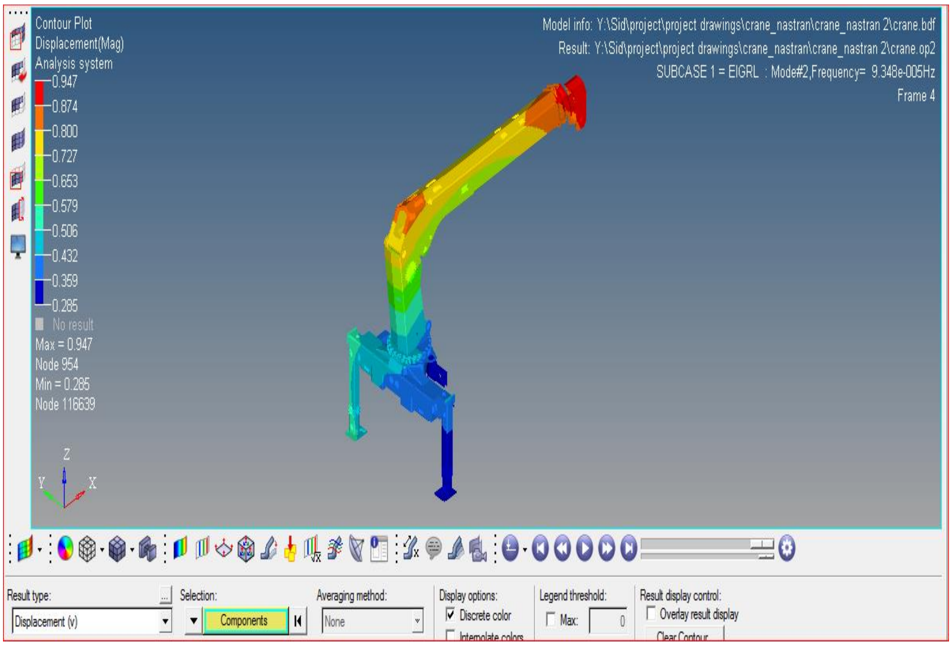

Fig 5.4 Mode 2

Mode 3:
This is the mode shape for 2ndmode.The above figure shows displacement for mode 2 i.e.,frequency of 93.4*10-6.Here the displacement is 0.947mm.

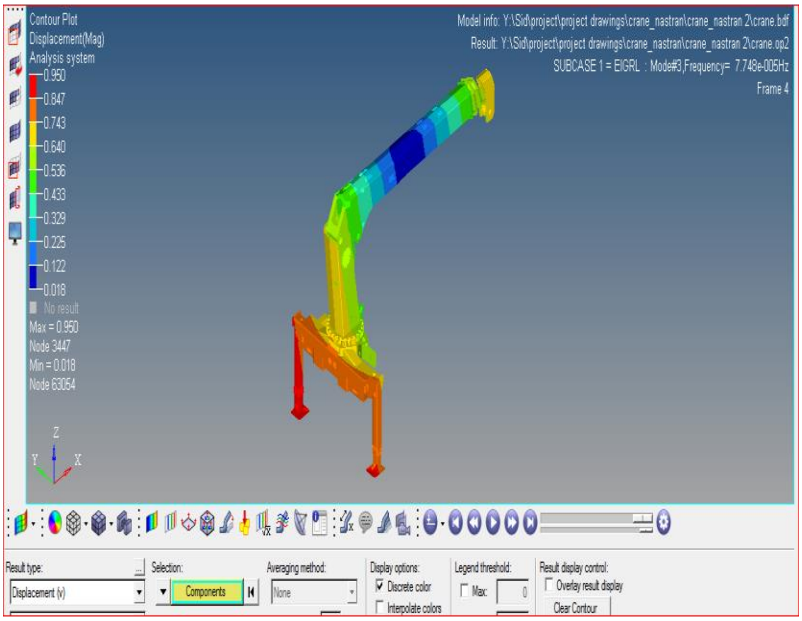

Fig 5.5 Mode 3

This is the mode shape for 3rdmode.The above figure shows displacement for mode 3 i.e.,frequency of 77.4*10-6.Here the displacement is 0.950mm.

Mode 4:

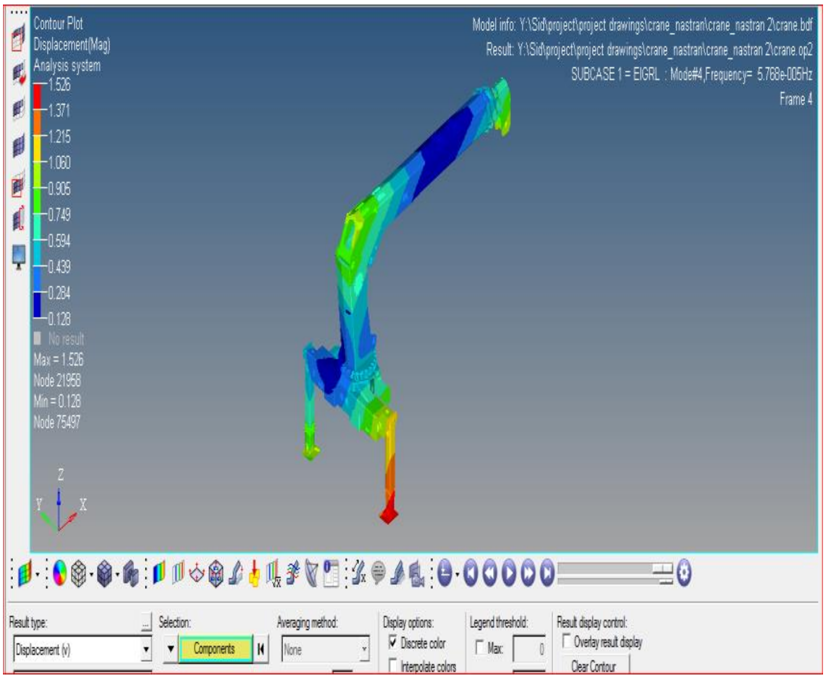

Fig 5.6 Mode 4

This is the mode shape for 4thmode.The above figure shows displacement for mode 4 i.e.,frequency of 57.68*10-6.Here the displacement is 1.526mm

Mode 5:

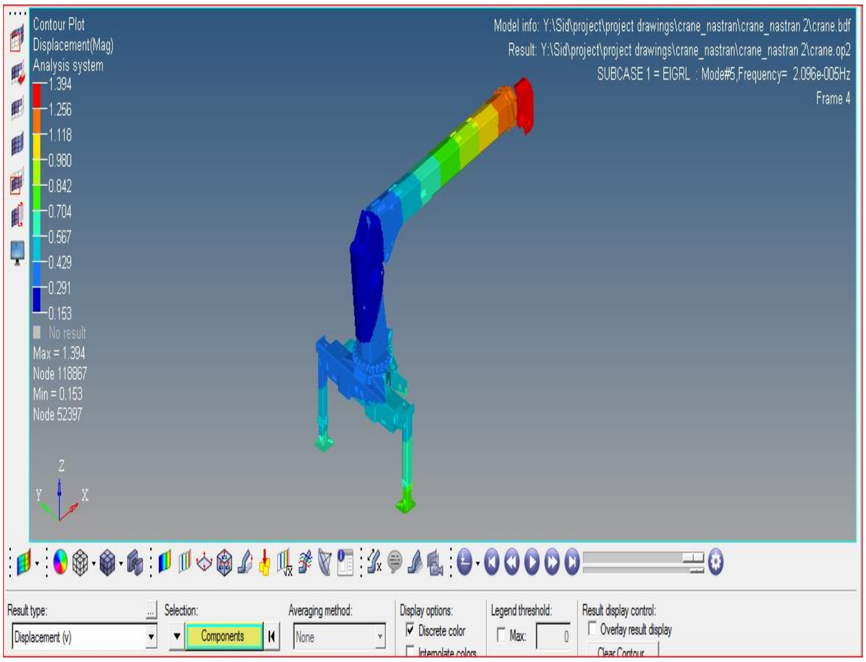

Fig 5.7 Mode 5

This in mode shape for 5thmode.The above figure shows displacement for mode 5 frequency of 20.96*10-6.Here the displacement is 1.394mm.

Modal Analysis Tabulation:

S. No.	Mode	Frequency(Hz)	Deflection(mm)
1	Mode 1	109.1*10-6	0.930
2	Mode 2	93.4*10-6	0.947
3	Mode 3	77.4*10-6	0.950
4	Mode 4	57.68*10-6	1.526
5	Mode 5	20.96*10-6	1.394

RESULTS AND DISCUSSION:

Theoretical vs Analysis result table

	Basic Design				New Design			
	Section-I(6mm)		Section-II(8mm)		Section-I(4mm)		Section-II(4mm)	
	Disp.(mm)	Stress(Mpa)	Disp.(mm)	Stress(Mpa)	Disp.(mm)	Stress(Mpa)	Disp.(mm)	Stress(Mpa)
Theoretical	11.14	62.95	0.108	7.02	16.49	93.19	0.107	9.02
Analysis	8.9	58.65	0.213	6.40	13.24	88.30	0.268	8.25

CONCLUSION:

1. The result analysis shows that the optimized model is suitable to take the specified load.
2. The existing crane model has been extensively tested for various mechanical properties.
3. Stress and Deflections are calculated for various thickness theoretically and suitable thickness value is decided for which stresses and deflections are within limits.
4. Theoretical validation is done for all analysis results to authenticate it.

REFERENCES:

Arrora, J. “Introduction to Optimum design”, 2nd ed., Academic Perss, 2004.
Coulton, J. J. (1974), "Lifting in Early Greek Architecture", The Journal of Hellenic Studies 94: 1–19
ErnstHustedt, AMES Ltd. 1999. (Air New Zealand Engineering).
G.S.G.Beveridge and R.S..Schechter, Optimization : theory and practice, McGraw Hill, New York, 1970.
Hursha Narayan (Robert Bosch Corporation).2002.
Jim Patterson (Hendricson Trailer Supension Systems). 2000.
JoeMetrisin (Florida Turbine Technologies, Inc.). 2002
M.M.Denn, Optimization by variation methods, McGraw-Hill, New York 1969.
McGraw-Hill Dictionary of Scientific & Technical Terms. McGraw-Hill. 1999
Morris A. J. Foundations of structural optimization: a unified approach, John Wiley & Sons, 1st ed., UK, 1982.7.G.B.Dantzig Linear Programming and Extensions, Princeton University Press.
Ravindran A, eklaitisG.V.Ragsdell K.M., “Engineering Optimisation: Methods and Applications”, John Wiley & Sons, UK,2006. 13M. 14.D.Koo, “ Elements of Otimisation” Springer-Verlag, New Yoork, 1977.
S.S.Rao – Optimization, Theory and Applications. Wiley inter-science, 1996.
