Analysis of Crane Hook under Static Loading Conditions

Sreenivasa R[1]
Dept. of Mechanical Engineering
Jain Institute of Technology, Davanagere
Karnataka, India[1]
rsreenivasadvg@gmail.com

Sirisha P[2], Ruchitha V R[3], Arif L S[4], Sanjay S N[5]
Dept. of Mechanical Engineering
Jain Institute of Technology, Davanagere
Karnataka, India[2,3,4,5]

***Abstract*—A numerical method for approximating solutions to boundary value issues for partial differential equations is the finite element method. Meshing is performed to discretize the geometry or structure, created into small pieces connected at grid or nodal point called elements. A lifting hook is a tool used to grab and lift loads using a hoist or crane, for example. A safety latch is typically included on a lifting hook to keep the load's attachment to the lifting wire rope sling, chain, or rope from being released. One or more built-in pulley sheaves can be used by a hook as a block and tackle to increase the lifting force.The present work carried out on analysis of Crane hook under static loading conditions.**

Keywords: Crane hook, Static Load, Mesh, Boundary Conditions

I. INTRODUCTION

A lifting hook is a tool used to grab and lift loads using a hoist or crane, for example. A safety latch is typically included on a lifting hook to keep the load's attachment to the lifting wire rope sling, chain, or rope from being released. One or more built-in pulley sheaves can be used by a hook as a block and tackle to increase the lifting force. A crane is a particular kind of device that can be used to lift and lower objects as well as move them horizontally. It is often furnished with a hoist rope, wire ropes or chains, and sheaves. The majority of its applications involve lifting and moving big objects. To generate mechanical advantage and move weights faster than a human could, the device makes use of one or more simple machines. Cranes are frequently used in the transport sector to load and unload cargo, in the construction sector to move materials, and in the manufacturing sector to assemble large pieces of machinery. [1]

The domain under consideration's 2D or 3D geometry must be fed into the numerical flow simulation. Mesh is used to segment the domain into smaller units. The discretization of the governing equations over an element is done using numerical techniques. [2]

After completing the drawing, the whole model was then imported in Altair Hypermesh 2019.1 software. Meshing was done in Hyper mesh Desktop 2019.1. Tetrahedral elements are used for mesh generation;Also, quality check for the FE model was conducted. [3]

Material properties are to be input in the software stating the ISOTROPIC property of gray cast iron. MAT1 is the card used for this property. [4]

II. METHODOLOGY

1. Use grid points to represent a continuous structure as a collection of discrete elements.
2. Create element stiffness matrices using the geometry, material parameters, and element properties.
3. Construct a global stiffness matrix out of all the element stiffness matrices.
4. Apply boundary conditions to the model to confine it.
5. Apply forces, moments, pressure, and other loads to the model.
6. Solve the matrix equation {F}=[K]{u} for displacements.
7. Using the findings of the displacement, determine the element stresses and reaction forces.

III. GEOMETRY OF THE MODEL

Use of CATIA V5 for modelling. The crane hook is modeled based on the standard dimensions as shown in Fig 1.

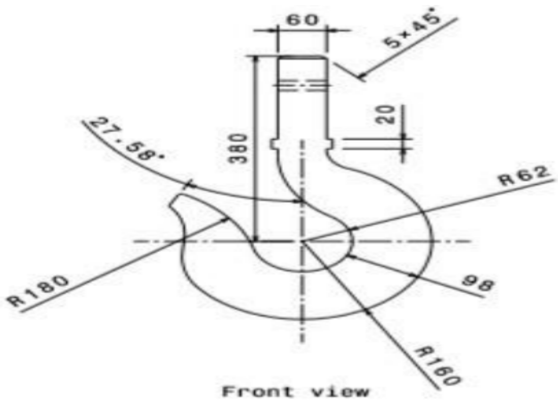

Figure 1: Crane hook with dimensions

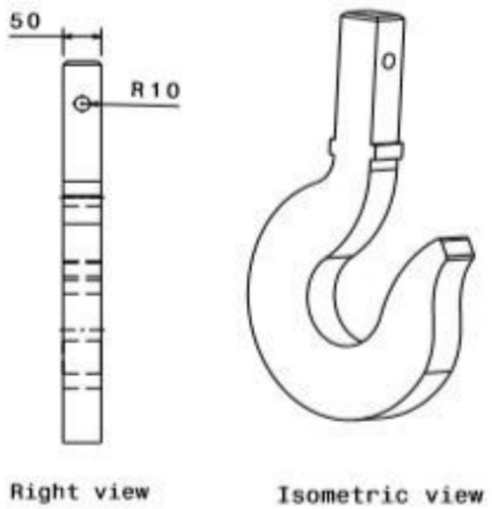

Figure 2: 3D Drawing of Crane hook

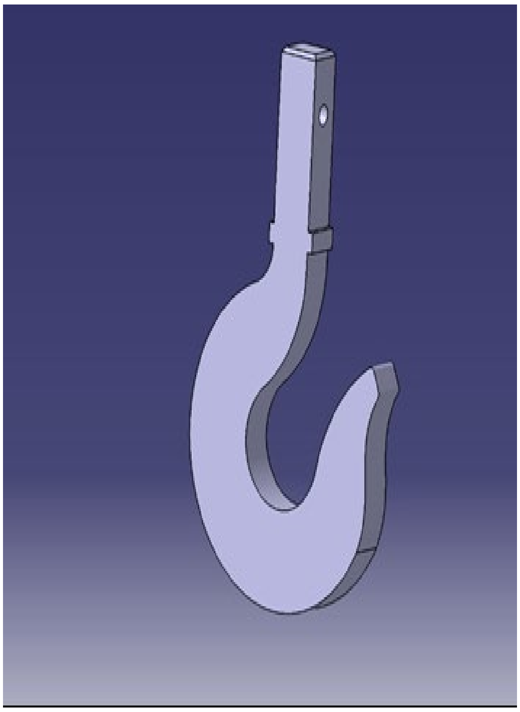

Figure 3: 3D Model of Crane hook

Use of Altair Hypermesh 2019.1 for Linear Static Analysis. The Fig 2 and Fig 3 shows 3D Drawing of Crane hook and 3D model of Crane hook respectively.

i) Pre-processing

The geometry of the domain under consideration in 2D or 3D must be fed into the numerical flow simulation. Mesh is used to delineate the domain into discrete parts. Governing equations over an element are discretized using numerical techniques.

A. Mesh Generation

After completing the drawing, the whole model was then imported in Altair Hypermesh 2019.1 software. Meshing was done in Hypermesh Desktop 2019.1. Tetrahedral elements are used for mesh generation;Also, quality check for the FE model was conducted. Fig 4 shows CTETRA4 element used in the analysis. Fig 5 shows meshed component and Fig 6 shows partial masked view of meshed component in the analysis. [5-6]

Material used: Gray cast iron

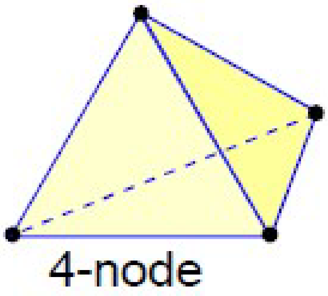

Figure 4: CTETRA4

In this case, CTETRA4 are an excellent alternative to CHEXA4 because of their superior shape. For all structural simulations, such as resolving stress and displacement, it's crucial to employ CTETRA4 with 10 grid points. For these situations, the four grid point CTETRA4 is too rigid. Applications involving heat transmission may use CTETRA4 with four grid points.

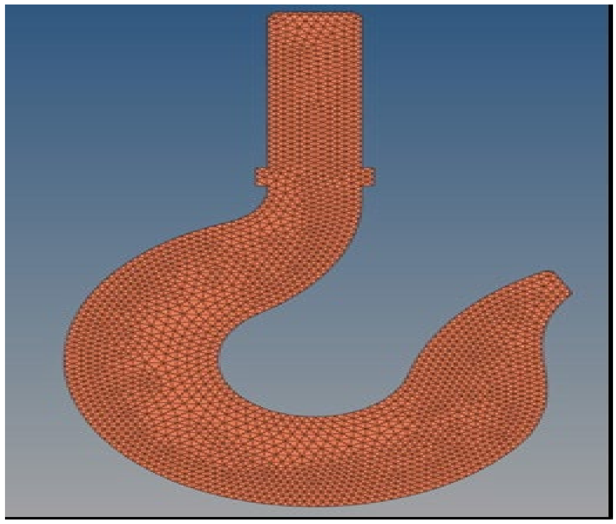

Figure 5: Components meshed with Tetrahedral Elements in hypermesh

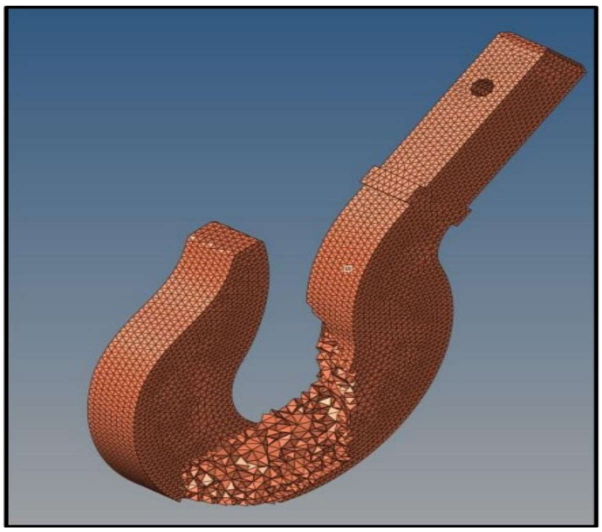

Figure 6: Partial masked view showing internal mesh pattern formed by Tetrahedral Elements

B. Assigning material and property to the FEM model

Material properties are to be input in the software stating the ISOTROPIC property of Gray cast iron. MAT1 is the card used for this property. As we have used 3D TETRA element for the mesh, property card PSOLID is used. The material defined in the previous step is assigned to this card. Fig 7 shows assignment of material to the component for analysis.

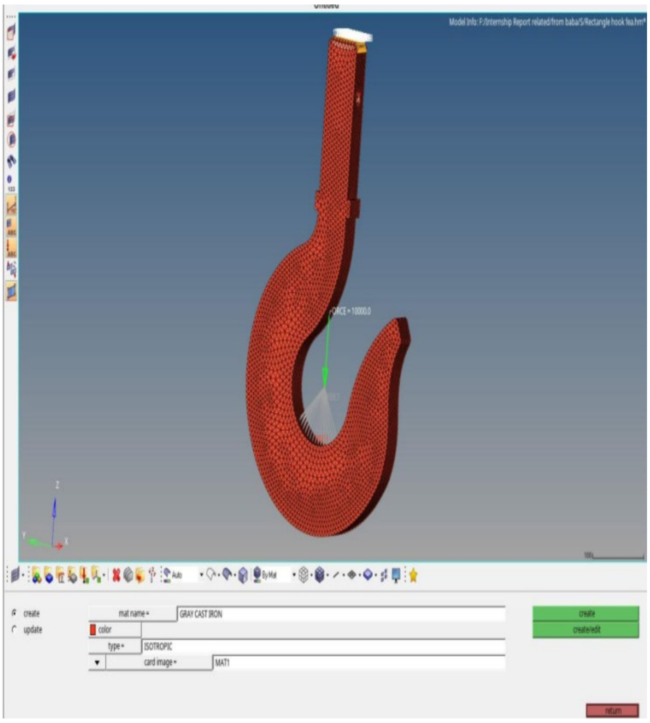

Figure 7: Snippet showing the assignment of MAT1-ISOTROPIC card

Mechanical properties of the material used:

Table 1 Material used: Gray Cast Iron

Mechanical Property	Value	Unit
Density	7200	Tonne/mm^3
Young's Modulus (E)	180	GPa
Poisson's Ratio	0.33	
Tensile Ultimate Strength	430	MPa
Tensile Yield Strength	276	MPa

C. Boundary Condition and Load

Load= 10kN

A Transverse load of 10000N is applied on the curved upper surface of the crane hook, by using a RBE3 element. Figs 8 – 11 shows load assignment to the component for the analysis.

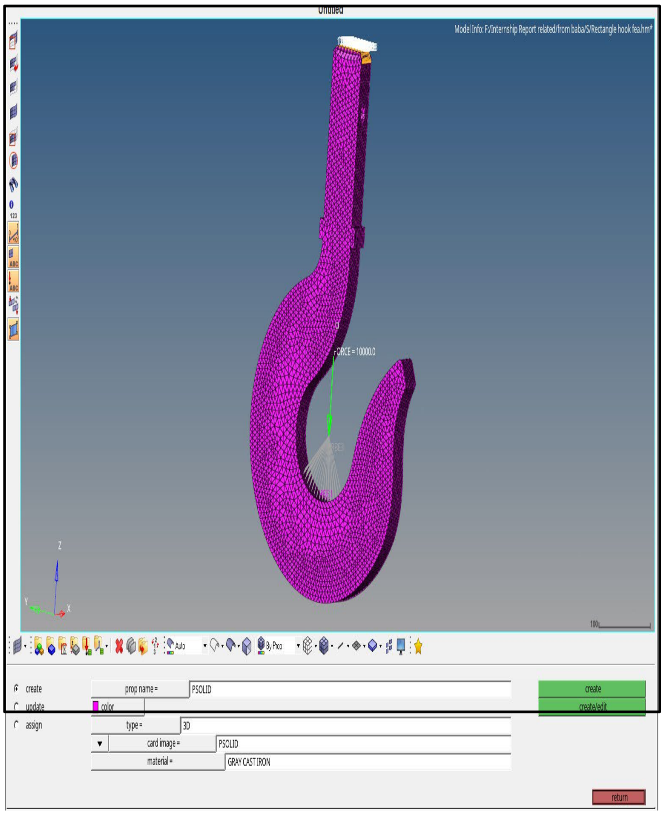

Figure 8: Snippet showing the PSLOID assigned to the FEM model

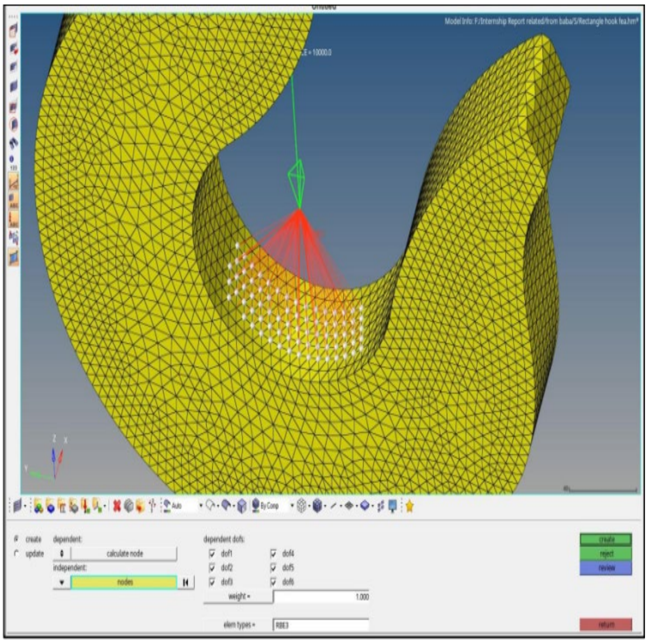

Figure 9: Snippet showing the creation of RBE3 Element

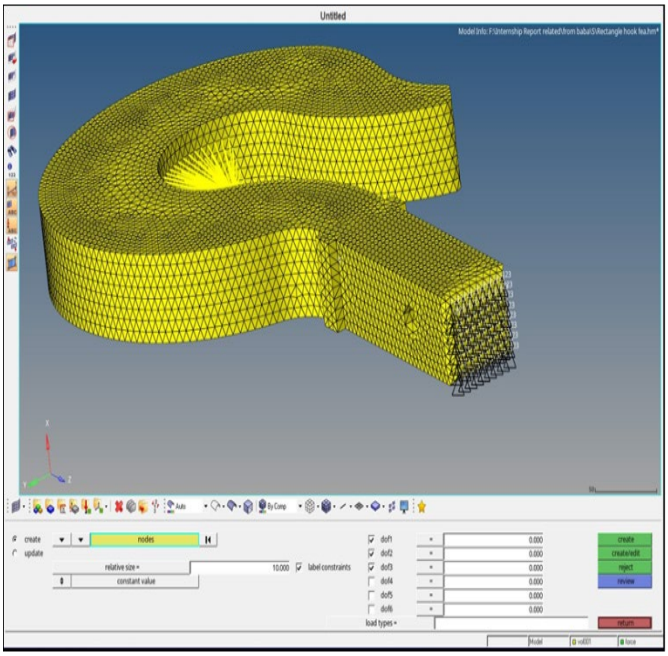

Figure 10: Snippet showing the application of Boundary Condition constraining the FE Model

RBE3 transmits the loads (MPC forces) applied on the dependent node to the selected independent nodes.

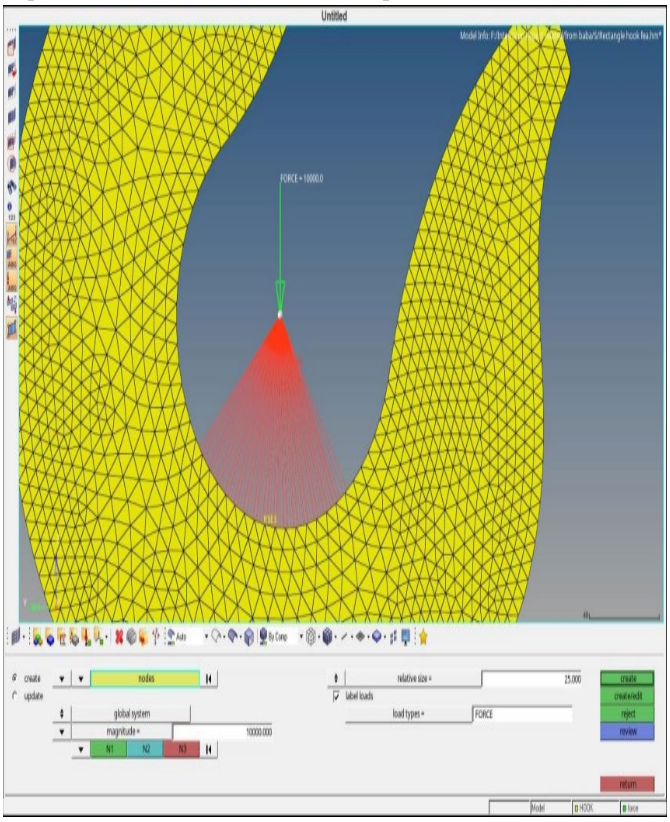

Figure 11: Snippet showing the applied transverse force F=10kN

D. Defining Load Step

The load step is a set of the applied boundary conditions and Force acting on the FE Model. Fig 12 shows boundary conditions applied to the component in load steps.

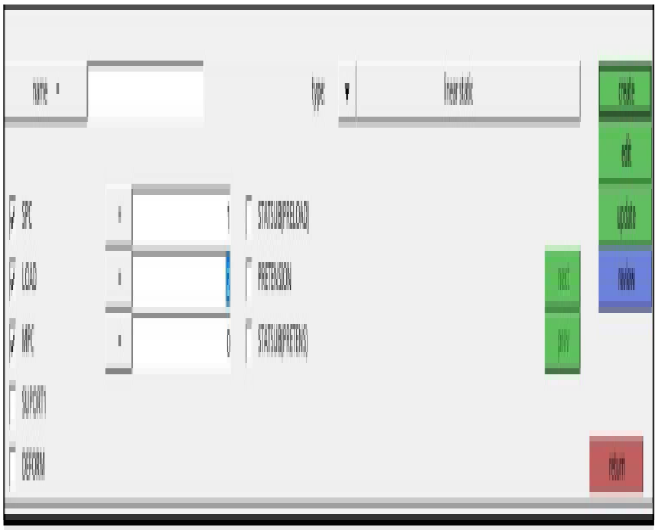

Figure 12: Snippet showing the applied BC and Force in the Load step "Transverse"

Here, a load step Axial's defined by taking the boundary condition 'SPC' and 'Load Force'

Processing (Solving) the FE problem

Solution of the FE problem obtained using Hyper Works Optistruct 2019.1 as Solver. Fig 13 shows hyper works optistruct solver view. [7-8]

Stresses and Displacements are the outputs plotted.

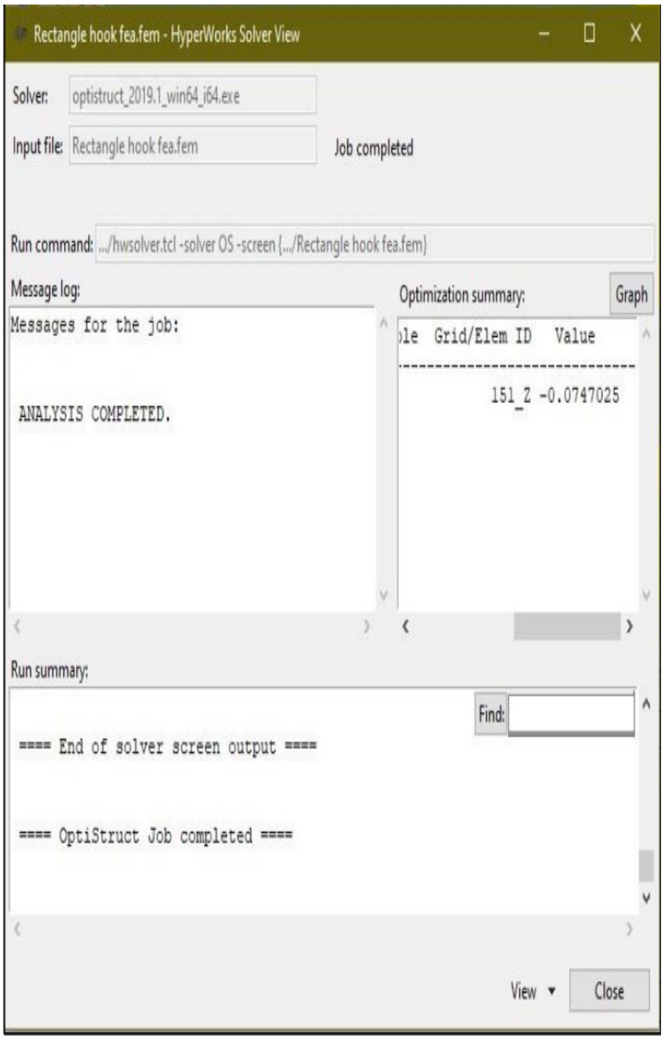

Figure 13: Hyper works Optistruct 2019.1 Solver view

IV. RESULTS

Calculation of allowable stress the material For Gray cast iron

We know that Ultimate strength = 430MPa

Factor of safety considered = 1.5

Hence,

Allowable stress = (Ultimate strength of the material) / (Factor of safety)

= 430/1.5

Allowable stress = 286.66MPa.

Fig 14 and Fig 15 shows displacement and stress plot of the component under load and boundary conditions.

Displacement Plot:

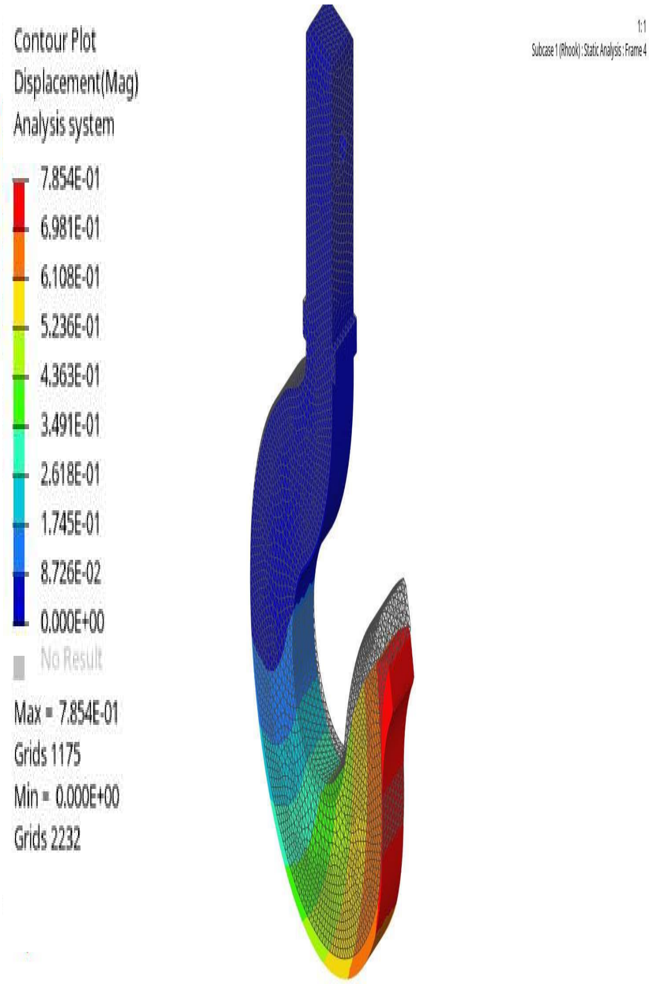

Figure 14: Displacement of the crane hook

Stress Plot:

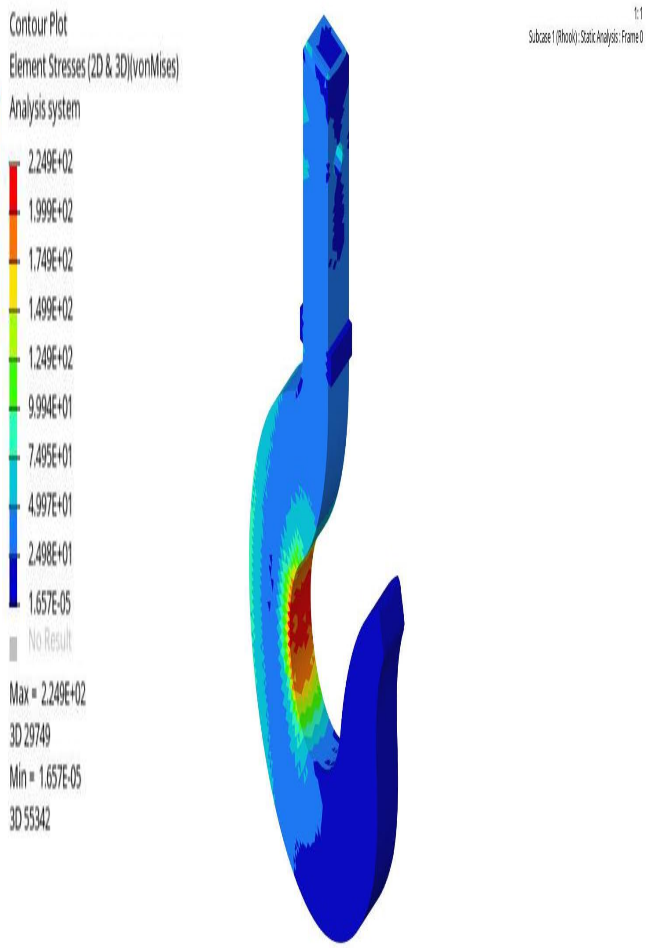

Figure 15: Stress plot of the crane hook

The Crane Hook can safely withstand the load of 10kN.

V. CONCLUSIONS

Table 2 Stress Values

Stress	Value	Unit
Allowable Stress	286	MPa
Peak Stress	224	MPa

According to the findings of the static analysis, the crane hook model can endure static loading circumstances since the resultant stress is lower than the maximum stress that the material can withstand. Therefore, the material selection and crane hook design are secure under static loading circumstances.

REFERENCES

[1] Gupta, R.S. Khurmi, J.K. "A textbook of machine design (S.I. units): [a textbook for the students of B.E. / B.Tech., U.P.S.C. (Engg. Services); Section 'B' of A.M.I.E. (1)]" Eurasia Publishing House. ISBN 81-219-2537-1, 14th Edition, 2008.

[2] G.R. LIU, S.S Quek, "The finite element method: A practical course", Burlington Butterworth-Heinemann ISBN 0750658865, 2nd Edition, 2013.

[3] YijunLiiu.," Introduction to the finite element method.", http://www.yijunliu.com/Teaching/Intro_FEM/Intro_FEM.htm, 2003.

[4] “Mechanical Properties of Ductile Irons” Cast Iron Science and Technology, pp. 456– 471, 2017.
[5] R Sreenivasa, C S Venkatesha, “Study the effect of crack length on stress intensity factor under dynamic fracture loading conditions”, International Journal of Innovative Research in Science, Engineering and Technology, Vol.,5, Issue 4, April 2016, ISSN 2319-8753.
[6] R Sreenivasa, C S Venkatesha, “Study the effect of crack on aircraft fuselage skin panel under fatigue loading conditions”, International Journal of Innovative Research in Technology, Vol., 2, Issue 11, April 2016, ISSN 2349-6002.
[7] Sreenivasa R, Kallesh S S, Naveen Kumar K H, Sandeep S H, “Study the effect of buckling on aircraft fuselage skin panel with or without airframe”, International Journal of Scientific Development and Research, Vol., 1, Issue 6, June 2016, ISSN 2455-2631.
[8] Sowmya R, Sreenivasa R, Kallesh S S, “Design optimization of airframe in aircraft fuselage structure under static loading conditions”, International Journal of Innovative Research in Engineering & Multidisciplinary Present – day Sciences, Vol., 6, Issue 4, July- August 2018, ISSN 2349-7300.