International Journal of Academic Engineering Research (IJAER)
ISSN: 2643-9085
Vol. 5 Issue 1, January - 2021, Pages: 29-35

Design, Analysis and Optimization of Crane Hook

Asad Yousaf[1]

[1]Department of Mechanical Engineering, University of Engineering and Technology Peshawar, Pakistan
asadyousafmarwat@gmail.com

Abstract*: The safety of crane hooks is of prime importance because they are used for lifting the load with the help of chains, links and cranes in most industries. Therefore, it should be designed such that it can withstand greater induced stresses to avoid failure. Keeping in view this important consideration, the research is selected. The main objective of this research is to identify the stress concentration areas of all the sections (types) of crane hook that are trapezoidal, rectangular and triangular crane hooks. Out of all these sections (types), the section (type) with minimum induced stress is selected. After that the shape of this section is modified (parametric optimization) to increase its working life and reduce the failure rates.*

Keywords— Crane Hook, Design, Analysis, Optimization, Induced Stresses

1. INTRODUCTION

Crane hooks are the components which are generally used to lift heavy loads in industries and construction work sites. The correct lifting can move large and heavy objects effectively and decrease manual handling operations. Incorrect lifting however, can lead to tragic misshapes. Each year incorrect lifting procedures cause injuries, loss of work time and property. Individuals, machinery, loads, methods and the work place are all critical variables for correct lifting. Thus such an important component in an industry must be manufactured and designed in such a way so as to deliver maximum performance without failure.

2. OBJECTIVE

The aim of the this research work is to study the stress distribution pattern of a crane hook using finite element analysis (FEA) method and optimizing the existing designs parametrically.

3. METHODOLGY

The three different models of crane hook which are under consideration are generated with the help of design and modeling software, Pro/ENGINEER wildfire 5.0. These models are produced using different steps and are as follows;

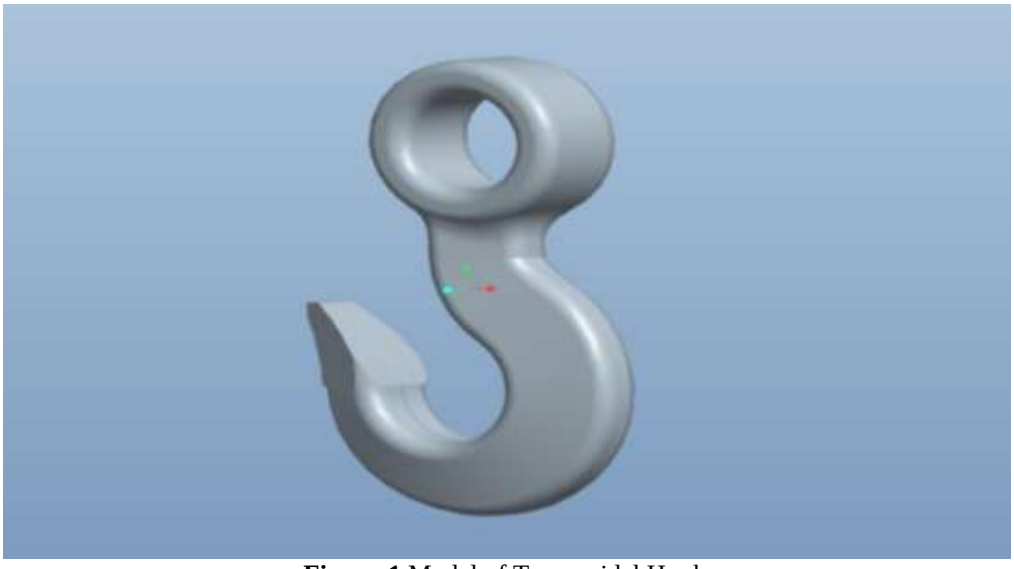

Figure-1 Model of Trapezoidal Hook

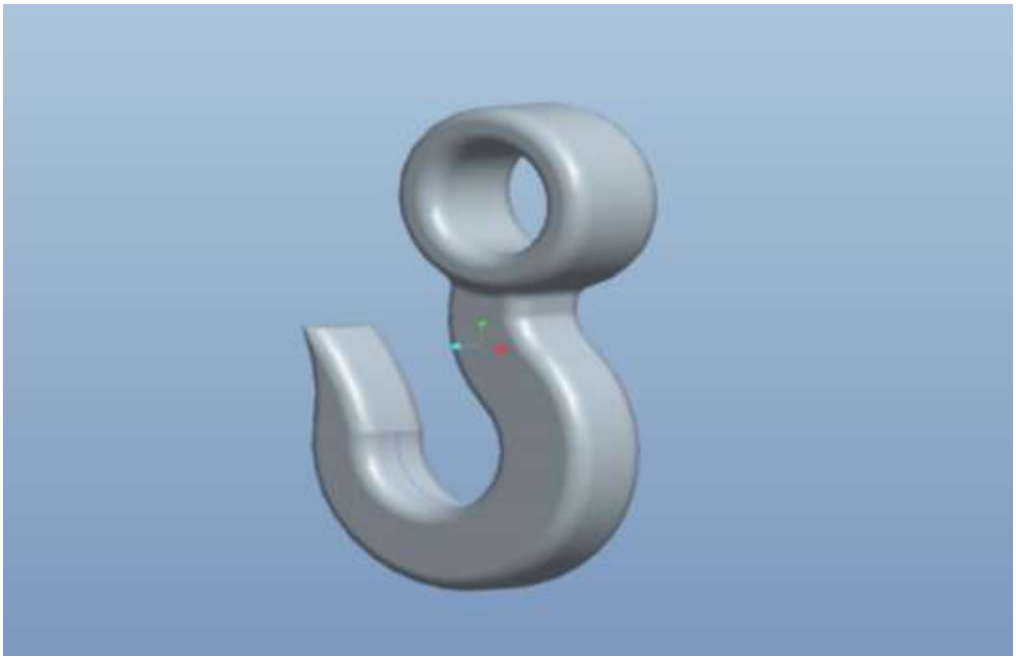

Figure-2 Model of Rectangular Hook

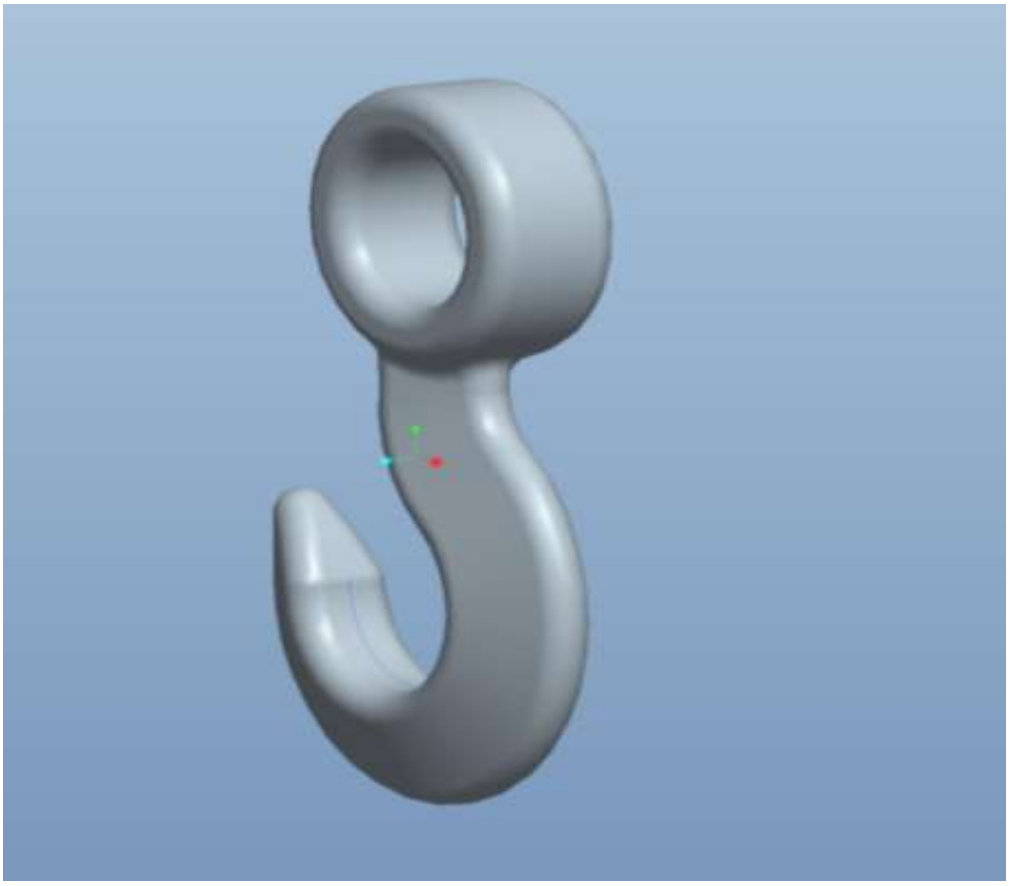

Figure-3 Model of Triangular Hook

The crane hook models designed with the help of Pro/ENGINEER are then subjected to analysis and simulation in real world environment to analyze different stresses developed and also the critical zones, where there are more chances of failure. The results are then used for improving the design of the crane hook. These models of crane hook are imported to the integrated module of Pro/ENGINEER called as Pro/MECHANICA. The same set of materials, constraints and loads are added to all these sections (types). From the maximum value of Von Mises stresses induced, for rectangular its value is 315.9MPa, for triangular it is 383.1MPa and for trapezoidal it is 289.3MPa.

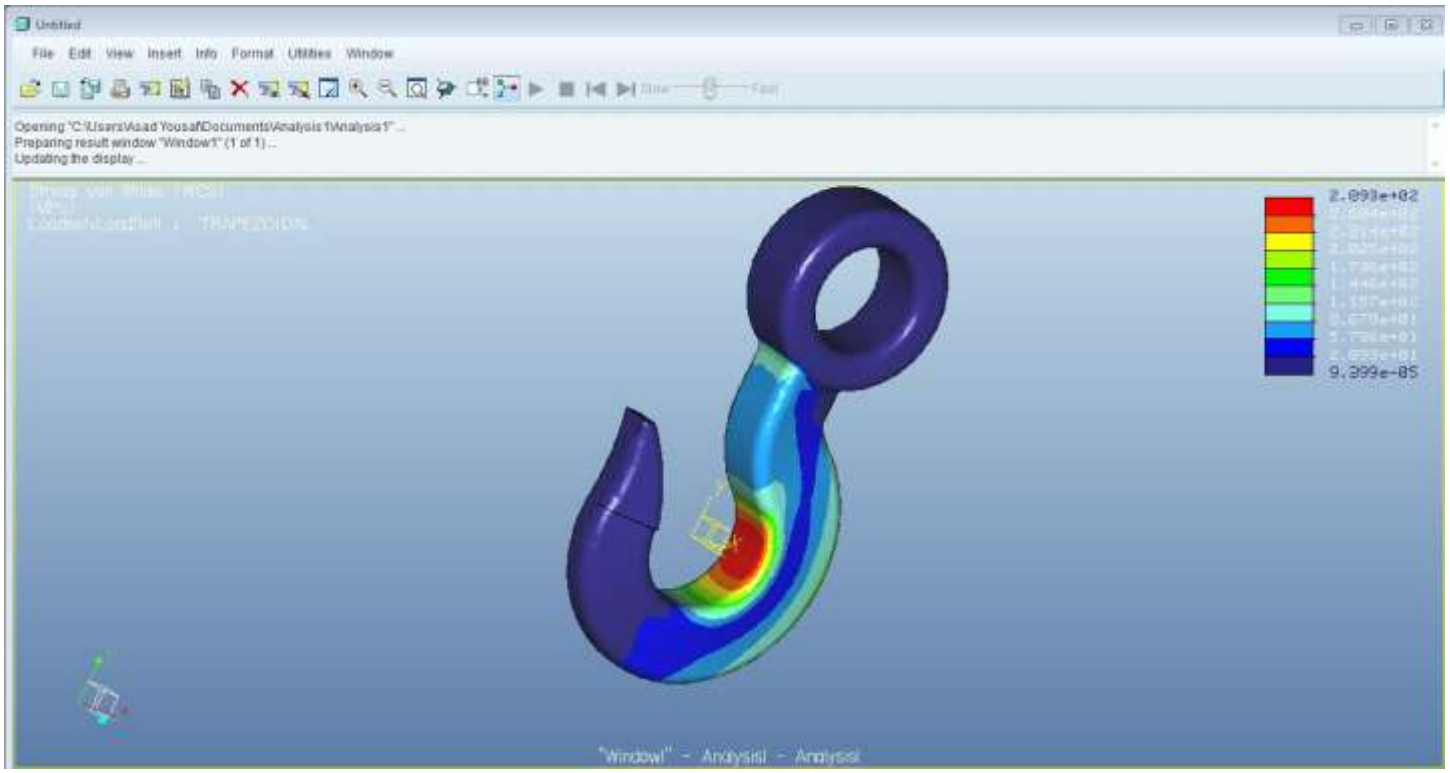

Figure-4 Fringe Plot of Static Analysis of Trapezoidal Section

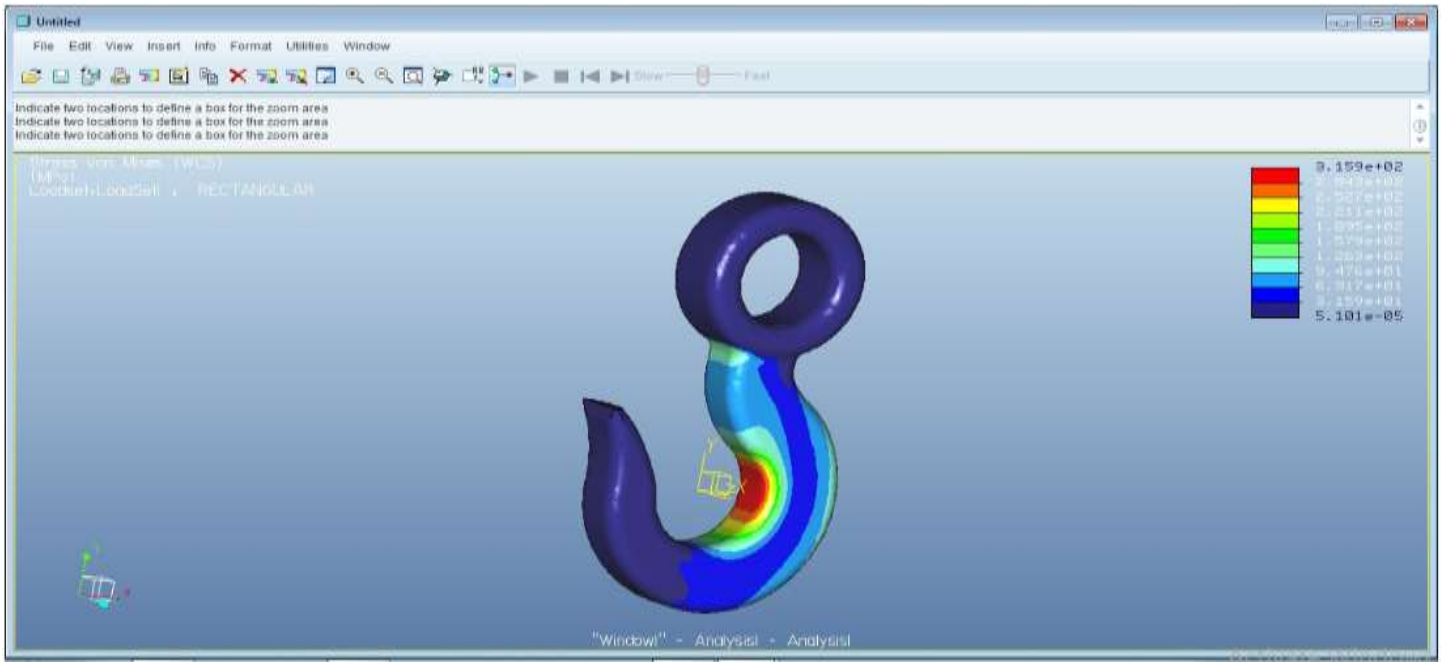

Figure-5 Fringe Plot of Static Analysis of Rectangular Section

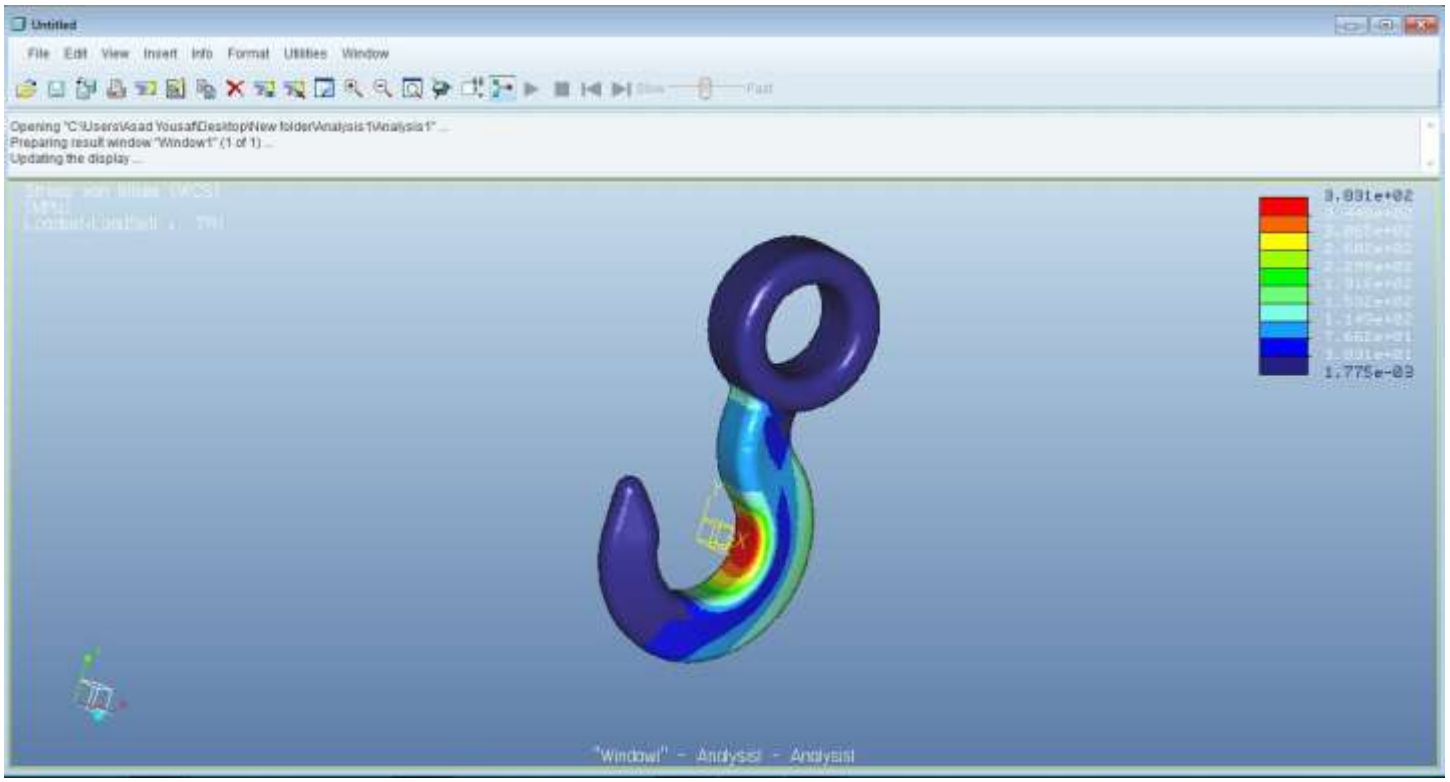

Figure-6 Fringe Plot of Static Analysis of Triangular Section

Therefore from all these three sections of crane hook, we selected the one with less stress induced that is trapezoidal section.

After that the sensitivity study of the trapezoidal section (type) of crane hook is carried out. In sensitivity analysis, one or more design parameters such as dimensions of the crane hook are varied to find the overall effect on the analysis results. Sensitivity study is carried out to find out the parameters that would be taken into account during design optimization. Focus is now to obtain an optimized design of the crane hook i.e. the best design within the given parameters.

For this we run the design optimization analysis. It takes a considerable amount of time after which the results are ready to be shown.

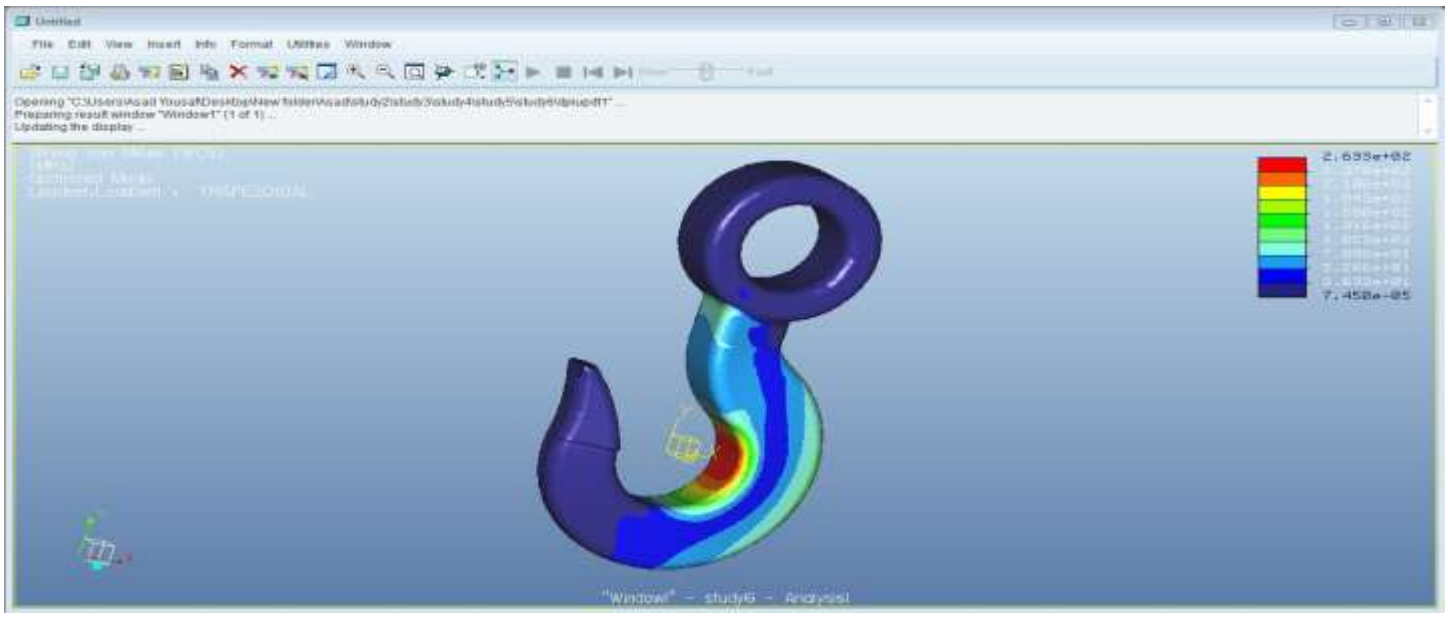

Figure-7 Design Optimized of Trapezoidal Crane Hook

The selected trapezoidal section of crane hook is optimized that is the induced Von Mises stresses are reduced to 263.3 MPa from 289.3 MPa.

4. RESULTS

A. Static Analysis Results

The results obtained from the static analysis of the trapezoidal crane hook are;

1. The maximum stress induced in the trapezoidal crane hook is 289.3MPa.
2. The minimum stress induced is $9.399e^{-05}$ MPa.
3. The area of the maximum stress (red zone) is near the curved portion of the hook.
4. Factor of safety=1.19 (345/289.3).

B. Design Optimization Results

The results obtained from the design optimization analysis of the trapezoidal crane hook are,

1. The maximum stress induced in the trapezoidal crane hook is 263.3MPa.
2. The minimum stress induced is $7.45e^{-05}$ MPa.
3. The area of the maximum stress is still near the curved portion of the trapezoidal crane hook.
4. Factor of safety=1.31 (345/263.3).

The comparison of results is presented in the form of the following table;

Table.1 Comparison of Results

Results	**Static Analysis (Initial)**	**Optimization**

Maximum VM stresses	289.3 MPa	263.3 MPa
Minimum VM stresses	$9.399e^{-05}$ MPa	$7.45e^{-05}$MPa
Factor of Safety	1.19	1.31
Conclusion	Design Improved	

5. CONCLUSION

The main objective of this research was to design 3D models of the three types of crane hooks and then to create a real world environment for these crane hooks to analyze them for the stresses induced. This analysis identified the most suitable type of crane hook and the critical zones in it. Next step was to obtain an optimized design based on the results obtained from analysis, so that the induced stresses and the chances of failure of the crane hook are reduced, and hence the factor of safety of the crane hook is improved.

6. RECOMMENDATIONS

The research could be further extended and the factor of safety of the crane hook be increased by incorporating other design variables like throat radius etc. Hence, the crane hook could be made more safe, failure resistant and user friendly.
The static analysis we have done with the help of Pro/MECHANICA can also be done in other analysis software such as ANSYS.
The analysis can also be done for other materials such as aluminum etc.

7. REFERENCES

[1] Structural Analysis of Crane Hook Joseph Leo .A.1 , ArutPranesh .K.2 , Balasubramani .V.3 (International Journal of Emerging Technology in Computer Science & Electronics (IJETCSE) ISSN: 0976-1353 Volume 12 Issue 2 – JANUARY 2015).

[2] Stress Analysis of Crane Hook Using FEA by B Nagaraju, M RajaRoy, P Venkatesh Reddy, K Satyanarayana Department of Mechanical Engg, Anil Neerukonda Institute of Technology and Sciences, Sangivalasa, Visakhapatnam- 531162,Andhra Pradesh.

[3] Investigation of Stress of A Lifting Hook with Different Methods: A Review by Yogesh Tripathi, U.K. Joshi Postgraduate Student, Department Mechanical Engineering, Jabalpur Engineering College, Jabalpur, India. Associate Professor, Department Mechanical Engineering, Jabalpur Engineering College, Jabalpur, India.

[4] Design and Stress Analysis of various cross section of Hook by Govind Narayan Sahu1, Narendra yadav2 Department Mechanical Engineering B.E. Scholar, Maha Maya College of Technology, Raipur Chhattisgarh India1 Asst. Prof., Rungta College of Engineering and Technology, Raipur Chhattisgarh India2 (International Journal of Modern Engineering Research (IJMER) Vol. 3, Issue. 4, Jul - Aug. 2013 pp-2187-2189 ISSN: 2249-6645).

[5] DESIGN AND ANALYSIS OF CRANE HOOK WITH DIFFERENT MATERIAL by atel Ravin B, Patel Bhakti K., Patel Priyesh M, Patel Dixit H. (All B.E students) ,Patel hirak s.(assistant professor) Department of mechanical engineering (International Journal of Advanced Computer Technology (IJACT) ISSN:2319-7900).

[6] Computer aided analysis and design of hoisting mehanism of an EOT crane, project thesis by Pradyumnakesharimaharana, DEPARTMENT OF MECHANICAL ENGINEERING NATIONAL INSTITUTE OF TECHNOLOGY ROURKELA.

[7] "Pro/MECHANICA Structure Wildfire: Elements and Applications (Integrated Mode)" by YVES Gagnon, SDC Publications (Chapter 1).

Authors

Asad Yousaf

Department of Mechanical Engineering,
University of Engineering and Technology Peshawar, Pakistan.