Static Analysis of Crane Hooks with Different Cross Sections–A Comparative Study using ANSYS

[1]MD IRSAD ANSARI, [2]DR. MANISH GANGIL

M.Tech.Scholar[1], Professor[2]

[1]mdirsadansari786@gmail.com, [2]rkdfbhojpal@gmail.com,
Department of Mechanical Engineering, RKDF, University Bhopal, (M.P.) India.

* Corresponding Author: MD Irsad Ansari Manuscript Received: Manuscript Accepted:

Abstract: A crane hook plays an important role for material handling from small to large industries. During transfer of heavy load, the hook is subjected to failure due to severe stress accumulation at its critical section. The design of crane hook is concerned with certain parameters like area of cross section, material, radius of curvature etc. The present study is an endeavor of the static analysis of crane hook with various cross sections subjected to identical loading considering the area of the cross section remains same. Design and load specifications are taken from standard practices. The hook of trapezoidal, T and I- section are modelled in CATIA V5R21 and are analyzed under similar conditions using ANSYS (Workbench 16.2) considering grey cast iron as the material of the model. The theoretical and ANSYS based results are found well in agreement. Apart from that an attempt has been made for the comparative study of maximum equivalent (Von Mises) stress among the various cross sections. A further study can be extended by applying composite or light material on the critical section to reduce the stress accumulation without compromising the load capacity.
Key words: ANSYS (Workbench 16.2), CATIA V5R21, Crane hook, Static analysis

I. Introduction

Crane hooks are highly liable machine parts generally applied for lifting heavy loads in industries and construction sites. It is basically a hoisting fixture designed to engage a ring or link of a lifting chain. Sometimes a pin of shackle or cable socket is used [1]. The hook block associated with the crane gives the required flexibility and adequate safety for lifting of material. The design and study of stress distribution of crane hook is vital as it plays a significant role by providing an efficient approach in material handling. In some case it has no alternative as it reduces manual involvement in material transfer. On the other hand improper design of hook leads to severe accidents and damage to man, machine and material. The recent study focuses on the evaluation of stress of different cross sectioned hook under identical loading condition. And a comparative analysis has been made for finding the best cross section. An extensive research study has been performed on the basis of design parameters using Taguchi method [2]. Osman Ashraf Ansari [3]et al., have presented real time pattern of stress concentration in 3D model of crane hook and the stress distribution pattern is verified for correctness on an acrylic model using diffused light Polari scope set up. To increase the working life by predicting the critical section is the main concern of the hook design and production. Most of the cases the crane hook is subjected to continuous loading and unloading processes. These lead to fatigue failure [3]. If a crack is found at the critical section, it propagates and finally invites accidents. Brittle fracture is more severe than ductile fracture as brittle fracture is sudden and total [4].An experimental based study has been done using diffused light Polari scope for stress analysis of various complex geometry of existing crane hook. The stress pattern of acrylic model of the presented geometries such as C, S, J, double J is carried out both analytically and experimentally. It has been observed that for identical loading condition stress concentration is more in S hook than C hook. Similarly J hook is subjected to more stress accumulation than double J hook. Extensive research work has been done on the basis of weight optimization by modifying the standard trapezoidal cross section and the further comparison has been made with the various basic cross sections. Finally a weight reduction of 18% with an increase of 6.79% stress has been optimized [5,6,7]. The FEA and analytical results and have presented that with the increase in the mass of the material of the trapezoidal section the stress has been affected significantly. Study has been performed considering the material of the crane hook and among forged steel, wrought iron and aluminum alloy, the minimum stress has been observed in case of forged steel [8]. A number of research works have been performed on the various basic cross sections and both analytical and FEA based result has been compared. The present study focuses on some standard cross section and gives an exclusive idea regarding the reduction of stress concentration by applying composites at the critical section. Due the continuous loading and unloading the hook is subjected to severe thermal stress about which a further study can be performed. The crane hook is basically a curved beam. The analysis is carried out by considering the well-known Winkler-Bach Formula for curved beam. Here a simplified formula has been presented for calculating various parameters for different cross sections [9]. The nomenclature of various parameters is in the Table1.

Table 1 General specification of crane hook [9]

Parameters	**Notation**
Radius of outer fiber	R_0
Radius of inner fiber	R_i
Radius of centroidal axis fiber	R
Radius of neutral axis fiber	R_N
Distance of inner fiber from neutral axis	h_i
Distance of outer fiber from neutral axis	h_o
Eccentricity between centroid and neutral axis($R-R_N$)	e

Apart from the above there are certain parameters associated with each of following cross sections shown in Table 2.

Table 2 General specification of cross section of crane hook

Parameters	**Notation**
Width of the cross section at the inner fiber	b_i
Width of the cross section at the outer fiber	b_a
Height between the two width	h
Thickness of the flange/web for I and T section	t
Area of cross section	A

The calculation of the above mentioned parameters are done for individual cross sections as below [9].

The size of different cross sections is depicted below.

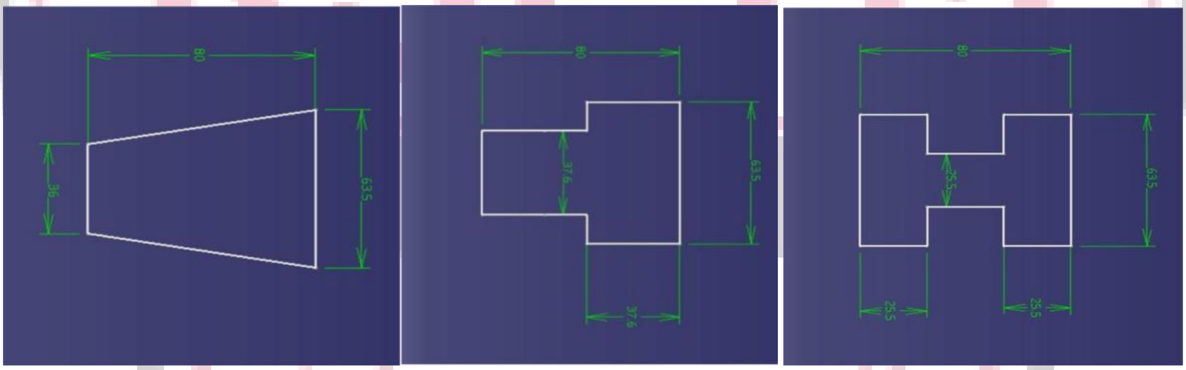


Figure 1 Different cross sections of the crane hook.

Different parameters associated with the cross sections are tabulated below.

Table 3 Specification of various cross-sections

Parameters(mm)	b_o	b_i	h	A (mm^2)	R_o	R_i	R	R_N	e	h	h
Trapezoidal	36	63.5	80	3980	140	60	96.31	90.91	5.4	30.91	49.09
T- Section	63.5	63.5	80	3981.84	140	60	94.82	89.56	5.26	29.56	50.44
I- Section	63.5	63.5	80	3978	140	60	100	93.15	6.85	33.15	46.85

The dimensions are taken from practical understanding and the hook is subjected to 6 tonnes load which is a common standard in industry. Grey cast iron is considered as the hook

2. STATICANALYSIS

The aim of this analysis is to investigate the stresses in the crane hook within the desirable limits to obtain a practical validation for the theoretical results. Performing the geometric modelling in *CATIA V5R21*the hook is subjected to static analysis, done by using *ANSYS (Workbench 16.2)*. The computer compatible mathematical description of the geometry of the object is called geometric modelling. *CATIA* is basically a CAD (computer-aided design) software that allows the mathematical description of the object to be displayed and manipulated as the image on the monitor of the computer [10, 11], on the other hand, *ANSYS* is an engineering simulation software that predicts with confidence about the performance of the product under the real-world environments incorporating all the existing physical phenomena [12]. The layout of static analysis involves meshing, boundary conditions and loading. The geometry of different cross sections is shown below in the Fig 2.

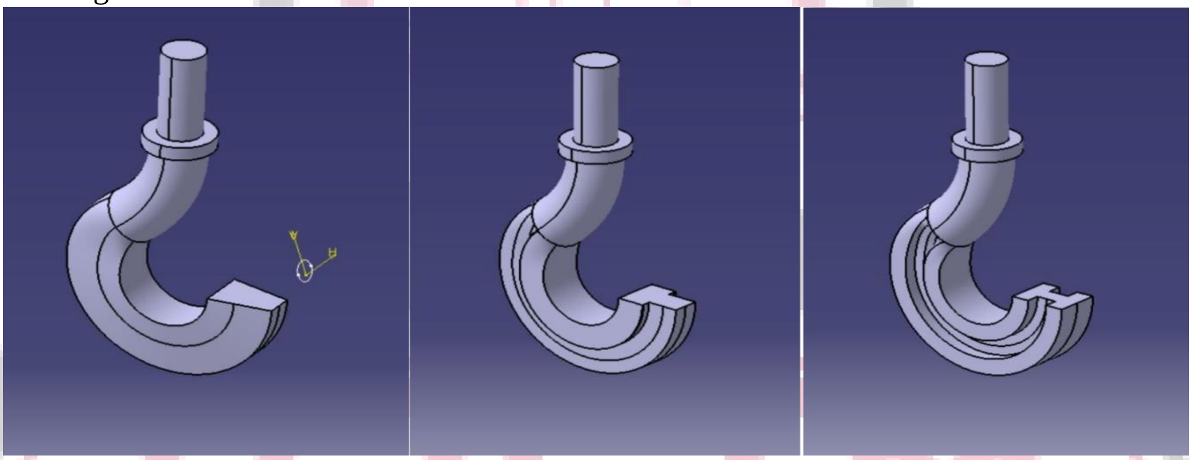
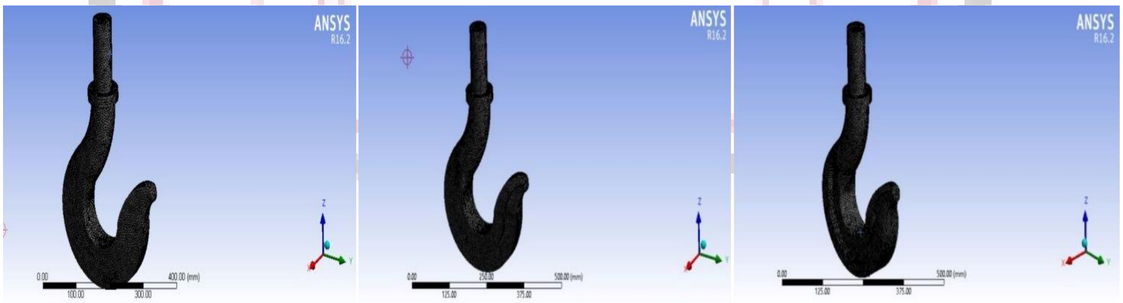


Figure 2 Various cross sections of the geometry.

4.1 Meshing

With the help of mesh generation the entire model is divided into smaller elements so that at each and every element, the required equations are solved. By adopting this method the accuracy of solution is improved [13, 14]. Here, for mesh generation advanced size function with curvature type is taken. The minimum element with in the solution domain under the Adaptive Mesh Refinement segment, the Max. Refinement Loops are taken as 3 and Refinement Depth as 2. Within Patch Confirming Method domain, the method is taken as Tetrahedrons.

The whole geometry is selected for mesh generation and for different cross section these generated meshed models are shown in Fig.3

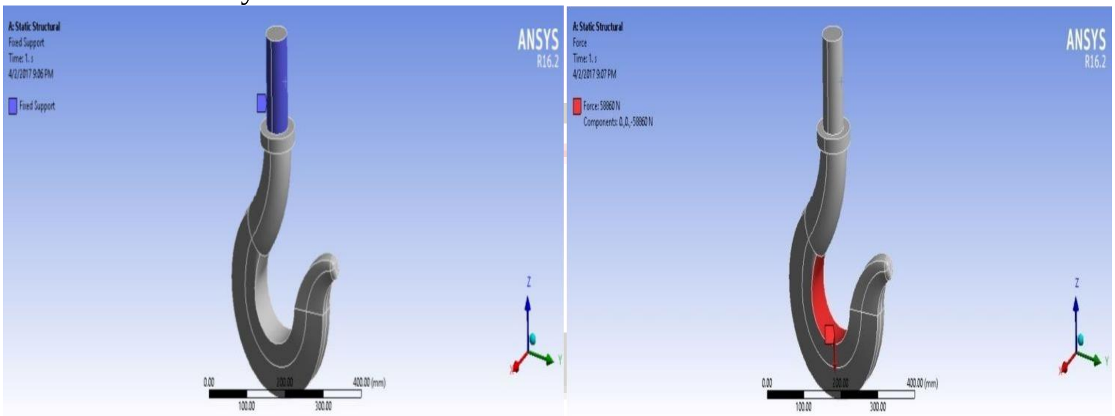


(a) (b) (c)

Figure 3 Meshed models of (a) Trapezoidal, (b) T and (c) I cross sections of the geometry.

4.2. Boundary Conditions

Based on the assumptions of curved beam theory, the boundary conditions are set in ANSYS Workbench. Fixed support (shown in Fig. 4(a)) and force is applied on the face having components in Z direction only. In the following Fig. 4(b) force having magnitude 58860 N has been introduced vertically downward.



(a) (b)

Figure 4 Boundary condition of the geometry.

Here theoretical and ANSYS based results are compared for three distinct cross sections considering the area of cross section remaining constant. All these are presented in the following Table 6.

Sections	Mass (Kg)	Theoretical stress(MPa)	Maximum equivalent (Von Mises) stress(MPa)	% change in stress
Trapezoidal	21.014	150.66	138.11	Reduction of 8.33
T Section	20.837	146.29	138.46	Reduction of 5.35
I Section	21.295	134.12	145.31	Increase of 8.34

A good agreement has been found between the theoretical bending stress and the ANSYS based Von Misses stress. Hence it validates the concept of curved beam during the stress analysis of the crane hook. We further conclude that the maximum stress developed on the hook is well within the safe limits considering the suitable factor of safety in each case. There is a slight variation between the analytical and ANSYS based result and this is due the assumptions made for evaluating the stress by analytical method which is different from actual practices. *ANSYS* is engineering simulation software that predicts with confidence about the performance of a product considering the real-world physics associated with it. The mass obtained from ANSYS for each case has been tabulated on the above table.

Conclusion

In the present work, the crane hook is modelled in CATIA V5R21 and same is analysed in the Static structural domain of ANSYS software. From the result discussed in the preceding section, it is concluded that, for the particular design specifications, the maximum bending for the trapezoidal and T – Section are very close and for trapezoidal section it is the minimum. But the mass optimum section in particular. An extended study can be carried out by replacing the material of the hook at its critical section by suitable composite and rest part with conventional metal for the reduction of stress accumulation. Focus can be made for the joining of metal and composite with different fasteners or suitable adhesives.

References

[1] M.Amareswari Reddy, M.N.V Krishnaveni, B.Nagaraju, M RajaRoy, Static analysis of crane hook with I- section and T- section using Ansys, International Journal of Engineering Trends and Technology (IJETT),Vol 26, Number 2,August 2015.

[2] Mr. A. Gopichand, Ms. R.V.S.Lakshmi, Mr. B. Maheshkrishna, Optimization of design parameters for crane hook using Taguchi method, International Journal of Innovative Research in Science, Engineering and Technology (IJIRSET),Vol. 2, Issue 12, December 2013, ISSN: 2319-8753.

[3] Osman Ashraf Ansari, Dr. P. Sampath Rao, International Journal of Mechanical Engineering and Technology (IJMET) Vol 7, Issue 5, September–October 2016, pp.75– 79, ISSN Print: 0976-6340, ISSN Online: 0976-6359

[4] A Sloboda, P. Honarmandi, Generalized elasticity method for curved beam stress analysis: Analytical and numerical comparisons for a lifting hook, May 2007, Mechanics Based Design of Structures and Machines 35: 319-332. Retrieved fromhttps://en.wikipedia.org/wiki/Fracture.

[5] S.A.Patil, A.H.Shitole, S.C.Pawar, G.G.Shinde, Stress analysis of different geometries of crane hook using diffused light polariscope, International Journal of Recent Research in Civil and Mechanical Engineering (IJRRCME) Vol. 2, Issue 1, pp: (66-77), Month: April 2015 – September 2015, Available at: www.paperpublications.org, ISSN 2393-8471.

[6] Tejas P. Jani, Pritesh G. Biholarav, Nayan R. Solanki, Dakshit J. Jivani, Mr. Anil N. Rathour, Dr. P. H. Darji, Weight optimization of crane hook having 8 tons load capacity by modifying cross section and comparison with various basic cross section, International Journal of Innovative Research in Advanced Engineering (IJIRAE), Vol 2,Issue 4, April 2015, ISSN: 2349-2163.

[7] Vikky Kumhar, Pradip Kumar Talapatra,Novel Kumar Sahu, Design and analysis of crane hook trapezoidal cross section using finite element analysis, VSRD International Journal of Mechanical, Civil, Automobile and Production Engineering, Vol. 5, Issue V May 2015 / 89 e-ISSN:2249-8303,p-ISSN:2319-2208©VSRD.

[8] Patel Ravin B, Patel Bhakti K., Patel Priyesh M, Patel Dixit H.,Patel hirak s., Designandanalysis of crane hook with different material, International Journal of Advanced Computer Technology (IJACT),ISSN:2319-7900.

[9] V.B. Bhandari, Design of machine elements, 2nd ed. New Delhi, India, McGraw Hill Education, ISBN: 0-07-061141-6, 2009, Ch. 4, sec.22, pp. 130-133.

[10] Sushovan Ghosh, Rohit Ghosh, Bhuwaneshwar Patel, Tanuj Srivastava, Dr. Rabindra Nath Barman, Structural analysis of spur gear using Ansys workbench 14.5, International Journal of Mechanical Engineering and Technology (IJMET) Vol 7, Issue 6, November– December 2016, pp.132–141, ISSN Print: 0976-6340, ISSN Online: 0976-6359 Retrieved from www.ansys.com [Accessed: 31 March 2017].

[11] Rohit Ghosh, Sushovan Ghosh, Shirish Ghimire and Dr. Rabindra Nath Barman, Static analysis of multi-leaf spring using ansys workbench 16.0, International Journal of Mechanical Engineering and Technology (IJMET) , vol.7, issue 5, 2016,