International Journal of Advanced Computer Technology (IJACT)

ISSN:2319-7900

DESIGN AND ANALYSIS OF CRANE HOOK WITH DIFFERENT MATERIAL

Patel Ravin B, Patel Bhakti K., Patel Priyesh M, Patel Dixit H. (All B.E students) ,Patel hirak s.(assistant professor)
Department of mechanical engineering

Abstract

In this paper the design of the hook is done by analytical method and design is done for the different materials like forged steel and high tensile steel. After the analytical method design and modeling of hook is done in modeling software (solid edge) .The modeling is done using the design calculation from the modeling the analysis of hook is done in FEA software (ANSYS).This result lead us to the determination of stress in existing model. By predicting the stress concentration area, the hook working life increase and reduce the failure stress

Introduction

Crane hooks are highly liable components and are always subjected to failure due to accumulation of large amount of stresses which can eventually lead to its failure.Crane hooks are the components which are generally used to elevate the heavy load in industries and constructional sites. A crane is a machine, equipped with a hoist, wire ropes or chains and sheaves used to lift and move heavy material. Cranes are mostly employed in transport, construction and manufacturing industry. Overhead crane, mobile crane, tower crane, telescopic crane, gantry crane, deck crane, jib crane, loader crane are some of the commonly used cranes. A crane hook is a device used for grabbing and lifting up the loads by means of a crane. It is basically a hoisting fixture designed to engage a ring or link of a lifting chain or the pin of a shackle or cable socket. Crane hooks with trapezoidal, circular, rectangular and triangular cross section are commonly used. So, it must be designed and manufactured to deliver maximum performance without failure. Thus the aim of this research is to study stress distribution pattern within a crane hook of various cross sections using analytical, numerical and experimental methods. [1]

The crane hooks are vital components and are most of the time subjected to failure due to accumulation of large Amount of stresses, which are ultimately leading to failure. Cranes are subjected to continuous loading and unloading. This causes fatigue of the crane hook. If the crack is developed in the crane hook, it can cause fracture of the hook and lead to serious accident. Bending stress, tensile stress, weakening of the hook due to wear, plastic deformation due to overloading, excessive thermal stresses are some of the other reasons of failure. [2]

Trapezoidal cross section

$$H = 0.93 \times C = 0.93 \times = 80 \text{ mm}$$
$$M = 0.6 \times C = 0.6 \times 8.5 = 51 \text{ mm}$$

$$= \frac{\frac{(b_1 + b_2)}{2} \times h}{\frac{(b_1 \times r_1 - b_2 \times r_1)}{h} \log_e \frac{r_2}{r_1} - (b_1 - b_2)}$$

$$= \frac{\frac{M \times H}{2}}{\left[\left(\frac{M_0 \times r_0}{H}\right) \log \frac{r_0}{r_1} - M\right]}$$

$$= \frac{\frac{51 \times 80}{2}}{\left[\left(\frac{51 \times 122.5}{80}\right) \log \frac{122.5}{42.5} - 51\right]}$$

$$r_n = 64.4 \ mm$$

$$(\sigma_t)\ total = (\sigma_{t)ri} + \sigma_t) = 179.67 + 24.04$$
$$= 207.71\ N/mm^2 \gg 120\ N/mm^2$$

Mathematics

Table:1-First trial of hook design for forged steel.

Hence, the hook section has to be modified .i.e. bigger section has to be selected and above procedure is repeated so that

$$(\sigma_t) total \leq 80\ N/MM^2$$

ISSN:2319-7900

$$H = 0.93 \times C = 0.93 \times = 80\ \text{mm}$$
$$M = 0.6 \times C = 0.6 \times 8.5 = 51\ \text{mm}$$

$$= \frac{\frac{M \times H}{2}}{\left[\left(\frac{M_0 \times r_0}{H}\right)\log\frac{r_0}{r_1} - M\right]}$$

$$= \frac{\frac{51 \times 80}{2}}{\left[\left(\frac{51 \times 122.5}{80}\right)\log\frac{122.5}{42.5} - 51\right]}$$

$$r_n = 70.1899\ mm$$

$$(\sigma_t)\ total = (\ \sigma_{t)ri} + \sigma_{t)}$$
$$= 86.0457 + 12.4571$$
$$= 98.5028\ N/mm^2$$

The sizes suggested by **IS-3815** are with permissible stress of about **200**N/mm^2**.** Proposing the enlarged section of the hook**, M= 75mm** and **H=105mm**.other dimensions remaining the same.

Table:2-Second trial of hook design for forged steel.

The total stress induced **is =112.11**N/mm^2 which may be considered safe for **forged steel.**

Modeling

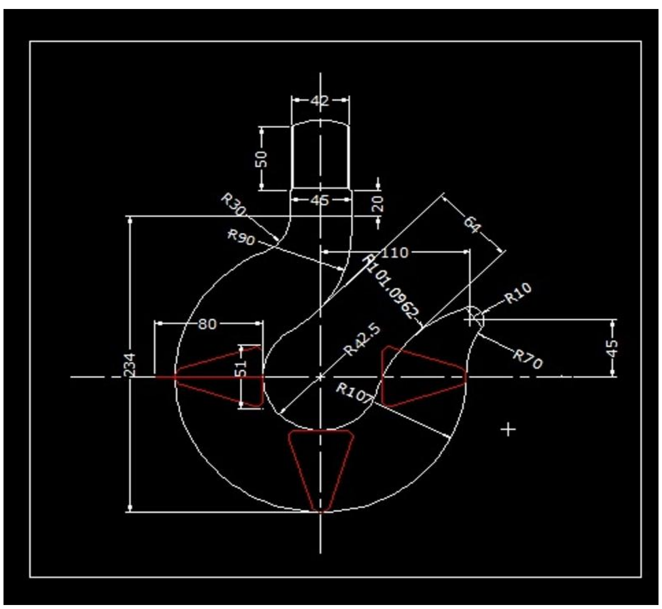

Figure 1. For trial 1

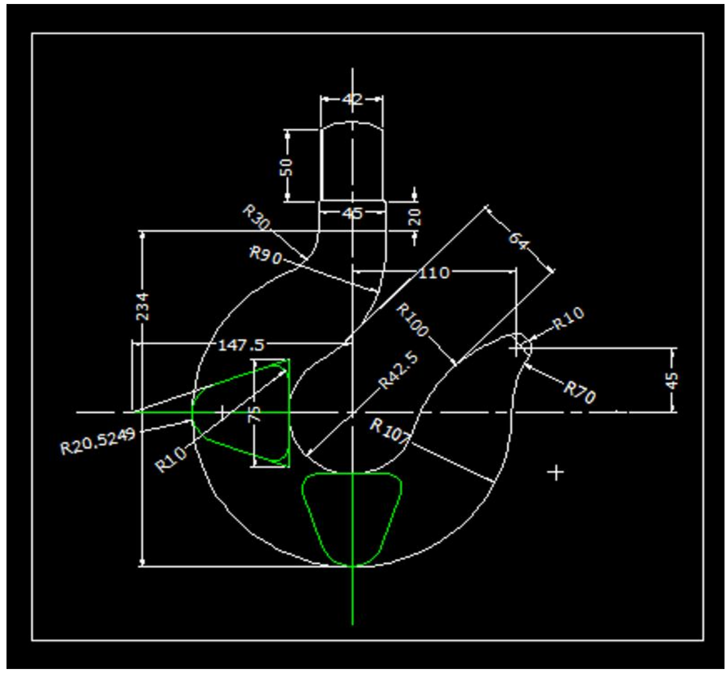

Figure 2. For trial 2

Solid Modeling

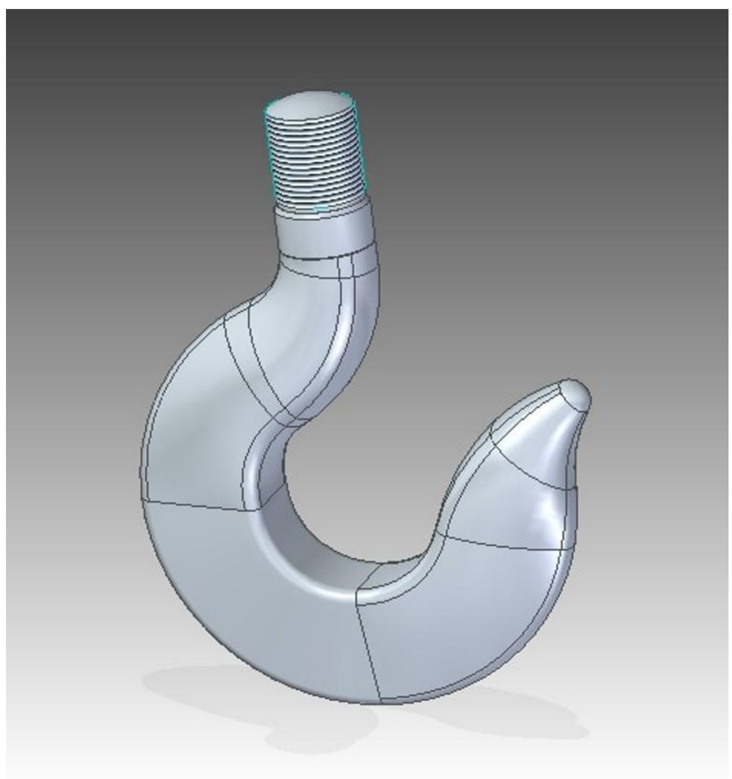

Figure 3. solid model for trial 1.

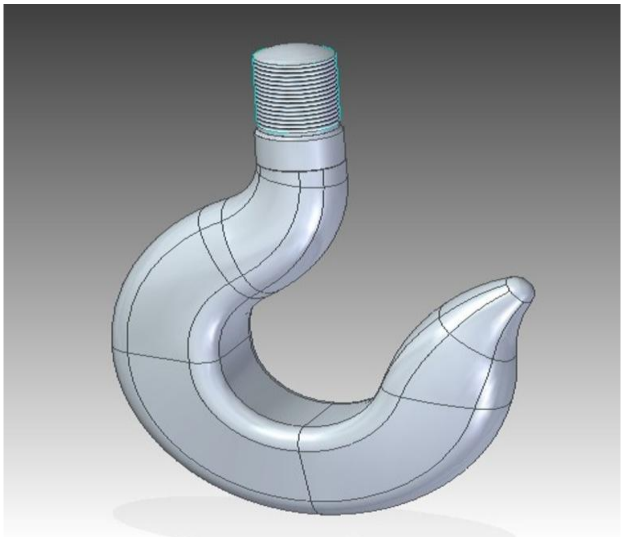

Figure 4. solid model for trial 2.

Meshing

The model is Made in ANSYS workbench used for the Stress analysis. The Finite Element Method (FEM) has developed into a key, indispensable technology in the modeling and simulation of advanced engineering systems in various fields like housing, transportation, manufacturing, and communications and so on The fine meshing is carried out. The meshed model created is shown in figure 5.[4]

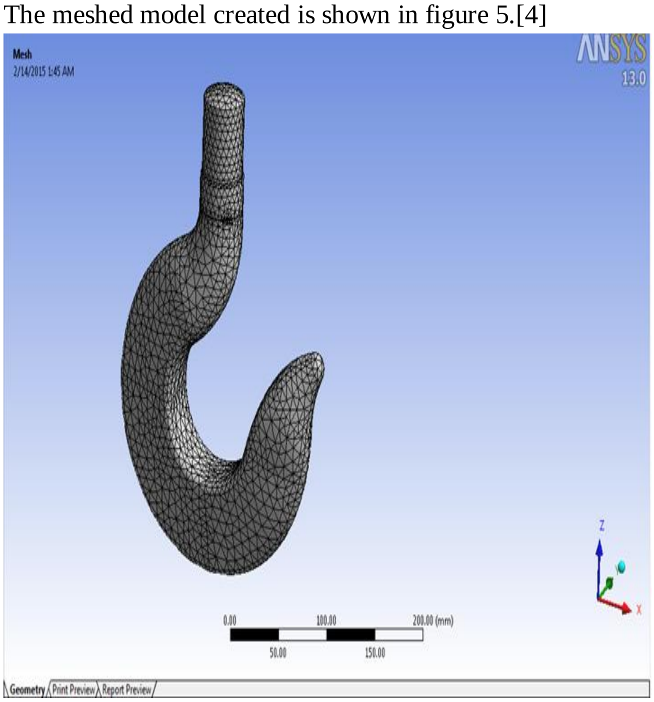

Figure 5. mesh generation for trial 1 and 2

Loading condition

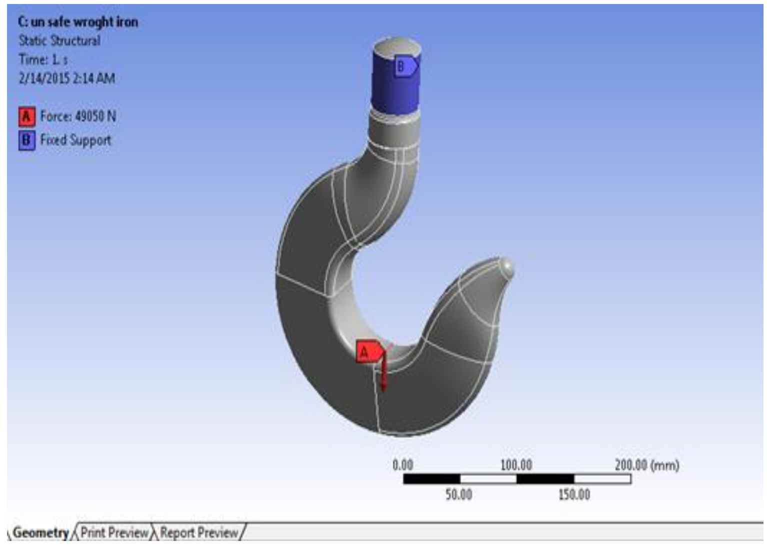

Figure 6. Point load on hook

Stress analysis on materials

For wrought iron

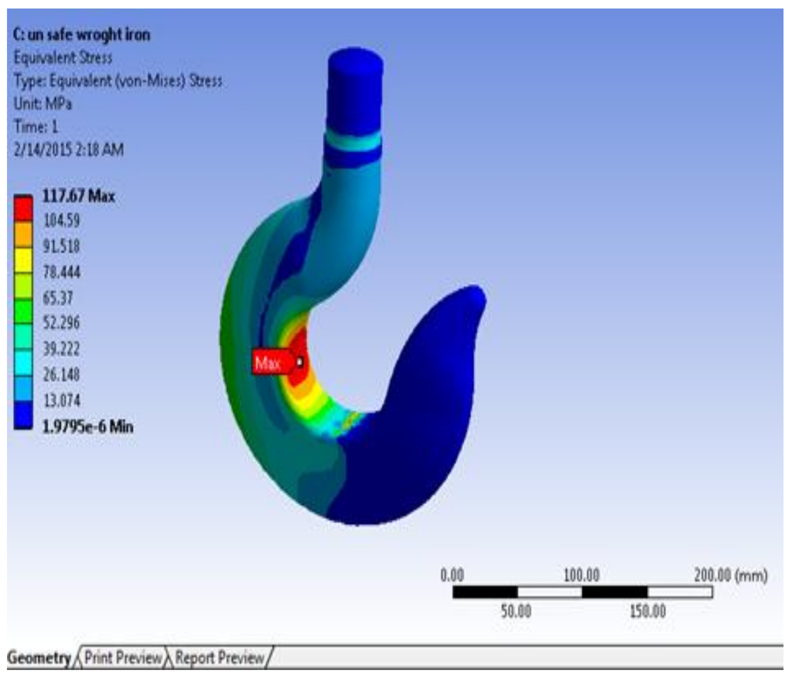

Figure 7. Equivalent stress in trapezoidal section for wrought iron

ISSN:2319-7900

For aluminium alloy

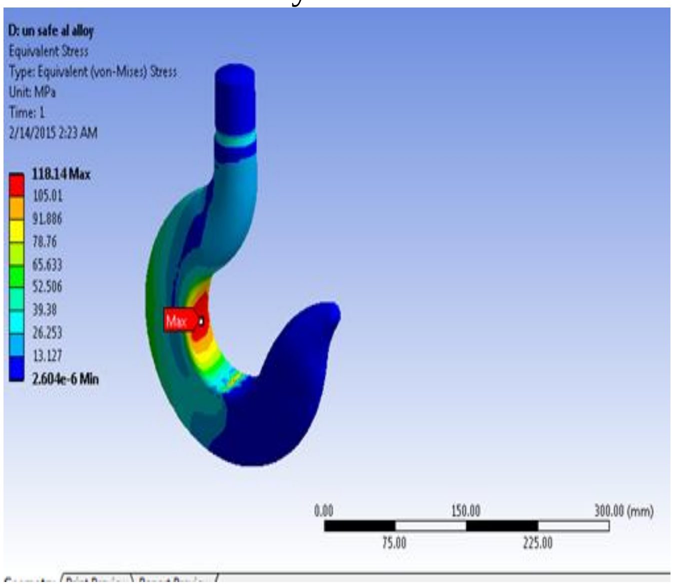

Figure 8. Equivalent stress in trapezoidal section for al.alloy

For forged steel

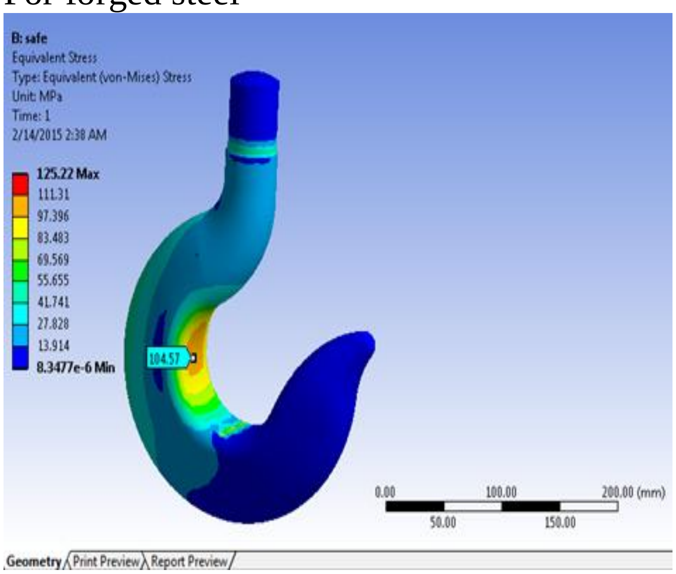

Figure 9. Equivalent stress in trapezoidal section for forged steel

Result and conclusions

The results of stress analysis calculated from FEA analysis for various different material such as Forged Steel ,Wrougt iorn/MS, Allmuniun Alloy. For the different Material, It is observed that keeping the tone are same with different Material topology we will get different results, but from the above table. it is found that the Forged Steel Material gives minimum stress which describe in below table.

Table:3 -First trial of hook design for forged steel.

Material	**Analytical stress** (N/mm2)	**Tone**	**Max.Equivalent Stress (N/mm2)**
Froged Steel	98.50	5 tone (49050 N)	104.00
Wrought iron	125.1	5 tone (49050 N)	117.67
Aluminium alloy	128.6	5 tone (49050 N)	118.41

References

(1) Chetan N. benkar, Dr.N.A. Wankhade (may 2014):-Finite element stress analysis of crane hook with different cross- section.

(2) M.Shaban , M.I.Mohamed, A.E.Abuelezz, T Khalifa (may 2013):- Determination of stress distribution in crane hook by caustic.

(3) Govindnarayansahu,narendrayadav(july-august ,2013) design and stress analysis of various cross section of hook.

(4) Ajeetbergaley , Anshumanpurohit(September 2013):- Structural analysis of crane hook using finite element method.

(5) E .Narayndas ,N. Puodziuniene (april,2012) circumferential stress concentration factor t the symmetric shallow notches of lifting hook of trapezoidal cross section.

(6) Rashmiuddanwadiker (August 26,2011):- Stress analysis of crane hook and validation by photo elasticity.

(7) Takuma nishimura ,Taka muromaki , Kazuyuki Hanahara ,Yukio Tada, Shigeyuki Kuroda and tadahisafukui(Dec-2010):-Damage factor estimation of crane hook.

Biographies

PATEL RAVIN B. received the diploma in Mechanical Engineering from **Gujarat technological university**, ahmadabad, Gujarat in 2012. The student of final year B.E degree in Mechanical Engineering from **Gujarat technological university**, ahmadabad, Gujarat in 2015,
ravin02021994@yahoo.in