# COMPARISON OF STRESS BETWEEN WINKLER-BACH THEORY AND ANSYS FINITE ELEMENT METHOD FOR CRANE HOOK WITH A TRAPEZOIDAL CROSS-SECTION

**Yogesh Tripathi[1], U.K Joshi[2]**

*[1]Postgraduate Student, [2]Associate Professor, Department of Mechanical Engineering,*
*Jabalpur Engineering College, Jabalpur, India*
***yogeshtripathi.ucl@gmail.com***

## Abstract

*Crane Hooks are highly liable components and are always subjected to failure due to the amount of stresses concentration which can eventually lead to its failure. To study the stress pattern of crane hook in its loaded condition, a solid model of crane hook is prepared with the help of CATIA (Computer Aided Three Dimensional Interactive Application) software. Pattern of stress distribution in 3D model of crane hook is obtained using ANSYS software. The stress distribution pattern is verified for its correctness on model of crane hook using Winkler-Bach theory for curved beams. The complete study is an initiative to establish an ANSYS based Finite Element procedure, by validating the results, for the measurement of stress with Winkler-Bach theory for curved beams.*

***Keywords:*** *Crane Hook, CATIA, ANSYS, Curved Beam, Stress, Winkler-Bach Theory*

---------------------------------------------------------------------***---------------------------------------------------------------------

## 1. INTRODUCTION

The structure-strength is the key index to response the load-bearing ability of the elevating equipment. Researching and analyzing the static characteristic of the hook that functions at the limited load has an important meaning to design larger tonnage hook correctly [1]. Crane Hook is a curved beam [2] and is widely used for industrial and construction work site for lifting loads by cranes.

In this study, stress analysis is implemented on the hook of DIN 15 401 [3]. Firstly, the 3-D model of the hook is built used CATIA. Secondly, the static analysis on the hook is proceeded by FEM software ANSYS. From the view point of safety, the stress induced in crane hook must be analyzed in order to reduce failure of hook.

## 2. FAILURE OF CRANE HOOKS

To minimize the failure of crane hook, the stress induced in it must be studied. A crane is subjected to continuous loading and unloading. This may causes fatigue failure of the crane hook but the load cycle frequency is very low. If a crack is developed in the crane hook, mainly at stress concentration areas, it can cause fracture of the hook and lead to serious accidents. In ductile fracture, the crack propagates continuously and is more easily detectable and hence preferred over brittle fracture. In brittle fracture, there is sudden propagation of the crack and the hook fails suddenly. This type of fracture is very dangerous as it is difficult to detect. Strain aging embrittlement due to continuous loading and unloading changes the microstructure. Bending stresses combined with tensile stresses, weakening of hook due to wear, plastic deformation due to overloading, and excessive thermal stresses are some of the other reasons for failure. Hence continuous use of crane hooks may increase the magnitude of these stresses and ultimately result in failure of the hook. All the above mentioned failures may be prevented if the stress concentration areas are well predicted and some design modification to reduce the stresses in these areas [4].

## 3. METHODOLOGY ADOPTED

A virtual model of DIN 15 401 lifting hook no. (6) similar to actual sample [3] is created using CATIA software and then model was imported to ANSYS workbench for Finite element stress analysis and the result of stress analysis are cross checked with that of Winkler-Bach formula for curved beams.

## 4. ANSYS FINITE ELEMENT METHOD

ANSYS is a finite element- based tool that provides a powerful design and analysis software package. Today ANSYS is viewed as a “household item” in many design and research institutions around the globe. It is regarded by many researchers and engineers as a modern, accurate, robust and visually sensible tool to provide solutions for numerous engineering and scientific problems.

This article is based on application of ANSYS in stress analysis and design of a lifting hook. As a case study author has considered DIN 15401 (German Standard for lifting hooks) as an example. Initially lifting hook was modeled in CATIA (Computer Aided Three Dimensional Interactive Application) conforming to original dimensions given in DIN standard and imported to ANSYS workbench for analysis purpose. This article is solely based on the authors' professional experience with using this software. Any opinions and findings cited here are those of author and do not necessarily reflect those of the software's manufacturer(s) and their distributor(s).

## 5. WINKLER-BACH FORMULA FOR CURVED BEAMS

For the straight beams, the neutral axis of the cross section coincides with its centroidal axis and the stress distribution in the beam is liner. But in case of curved beams, the neutral axis of the cross-section is shifted towards the centre of curvature of the beam causing a non-linear distribution of stress. The application of curved beam principle is used in crane hooks [5]. This article uses Winkler-Bach theory to determine stresses in a curved beam.

### 5.1 Stresses in a Crane Hook

For the crane hook shown in fig. 1, O is the centre of curvature and the load line passes through O1. The radius of curvature of the centroid is R. Bending moment about the centroid G is

$$M=Wx$$

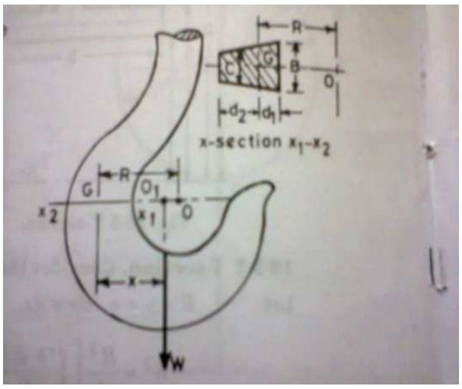


**Fig. 1** Crane hook

This bending moment is such that it is tending to decrease the curvature, i.e. this is a negative bending moment. Therefore, bending stresses at a point x1 and x2 are respectively,

$$\sigma 1=Wx/AR\ \{R2/h2\ (d1/R-d1)-1\} \quad \text{(tensile)} \quad (1)$$

$$\sigma 1=Wx/AR\ [1+R2/h2\ (d2/R+d2)] \quad \text{(compressive)} \quad (2)$$

Direct stress, $\sigma d=W/A$ (tensile)

Therefore Resultant stress at

$$x1=\sigma 1+\sigma d \quad (3)$$

$$x2=\sigma 2-\sigma d \quad (4)$$

### Nomenclature

A= area of the cross section, mm2
M= Uniform bending moment applied to the beam, assumed positive when tending to increase curvature.
R=radius (OG) of curvature of centroidal axis, mm
W=load, N
h= a constant for the cross section of beam.
B= width (inner side of trapezoidal cross-section), mm
C=width (outer side of trapezoidal cross-section), mm
G=centroid of the principal cross-section.
x=distance between load line (O1) and centroid (G) of the principal cross section, mm
O=centre of curvature.
σd=direct stress, N/mm2

## 6. DIMENSIONS, DESIGNATION AND MATERIAL OF HOOK

Drop forged (S) unmachined part (R) with nose (N) for single hook No. 6 has been considered for modeling in CATIA software. Dimensions chosen for hook are tabulated below:

**Table.1** Dimensions of type single hook No. 6 of strength class P

| Hook No. | a1 | b1 | b2 | d1 | e1 | e2 | h1 | h2 | L1 |
|---|---|---|---|---|---|---|---|---|---|
| 6 | 90 | 80 | 67 | 60 | 210 | 227 | 100 | 85 | 385 |
| | r1 | r2 | r3 | r4 | r5 | r6 | r7 | r9 | Weight in Kg. |
| | 10 | 16 | 45 | 135 | 190 | 131 | 112 | 200 | 17.1 |

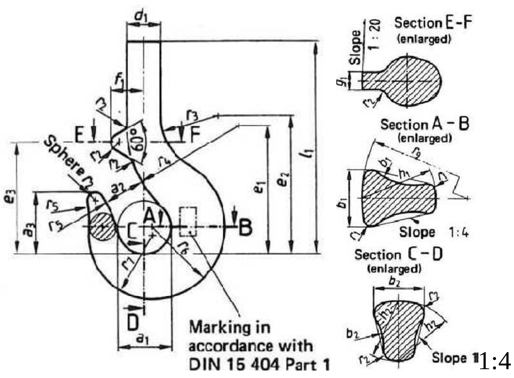


**Fig2.** Single hook DIN 15 401-RSN 6 –P

Type of material used for crane hook is StE 355 steel. The material properties are:

**Table2.** Material Properties of StE 355 [6]

| Quantity | Value | Unit |
|---|---|---|
| Young's modulus | 200000 – 200000 | MPa |
| Tensile strength | 650 – 880 | MPa |
| Elongation | 8 – 25 | % |
| Density | 7700-7700 | kg/m3 |
| Yield strength | 350 – 550 | MPa |

Here material properties are directly assigned in ANSYS.

## 7. ANSYS PROCEDURE FOR FINITE ELEMENT ANALYSIS

### Model

- Geometry – Imported from CATIA in ".stp" format.
- Solid-generated ANSYS geometry.
- Mesh-tetrahedral element selection.

**Table3.** Number of nodes and elements

| | |
|---|---|
| Nodes | 8333 |
| Elements | 4680 |

### Static_Structural

- Analysis settings –analysis settings are used for static structural, single step loading.
- Loads –a load of 12500 kg is applied at principal cross-section (trapezoidal) of the hook.
- Eye section at top of the shank, kept fixed.

### Solution

- Equivalent (Von-Mises stress)
- Total deformation
- Structural error

ANSYS based FE-analysis is shown below:

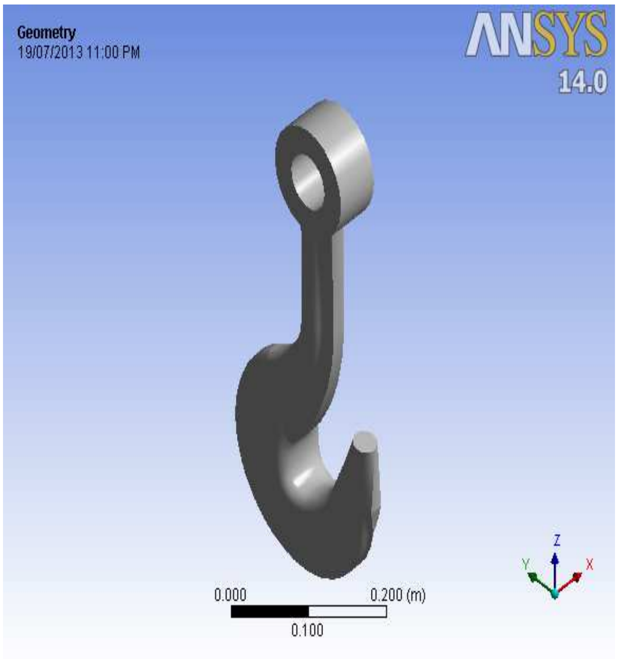


**Fig3.** ANSYS .stp file from CATIA

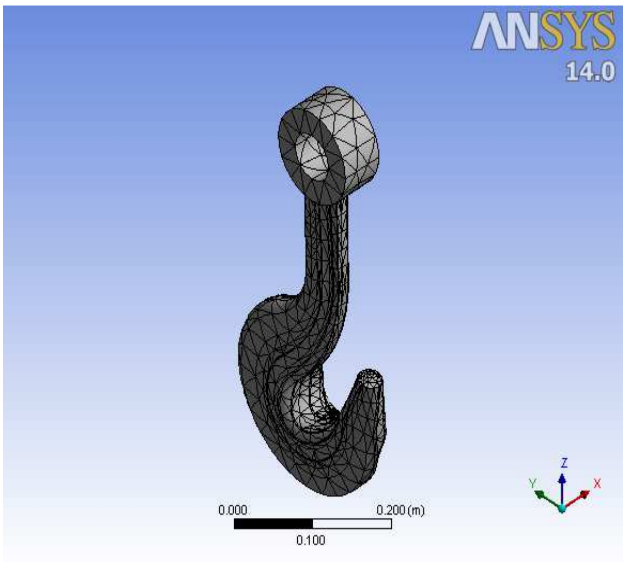


**Fig.4** Mesh with automatic setting is generated


**Fig5.** Fixed support and force of 12500 kg is shown in fig.

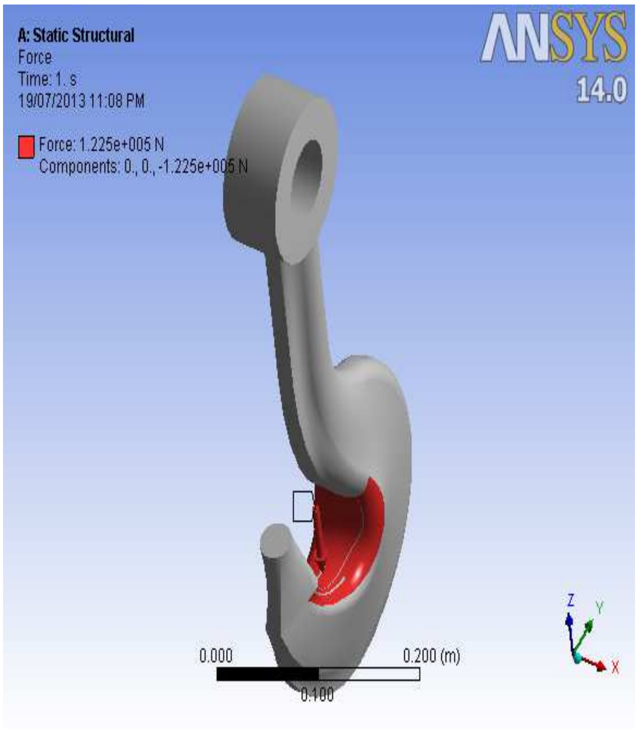


**Fig.7.** Force of 12500 kg is applied at inner portion of principal cross-section

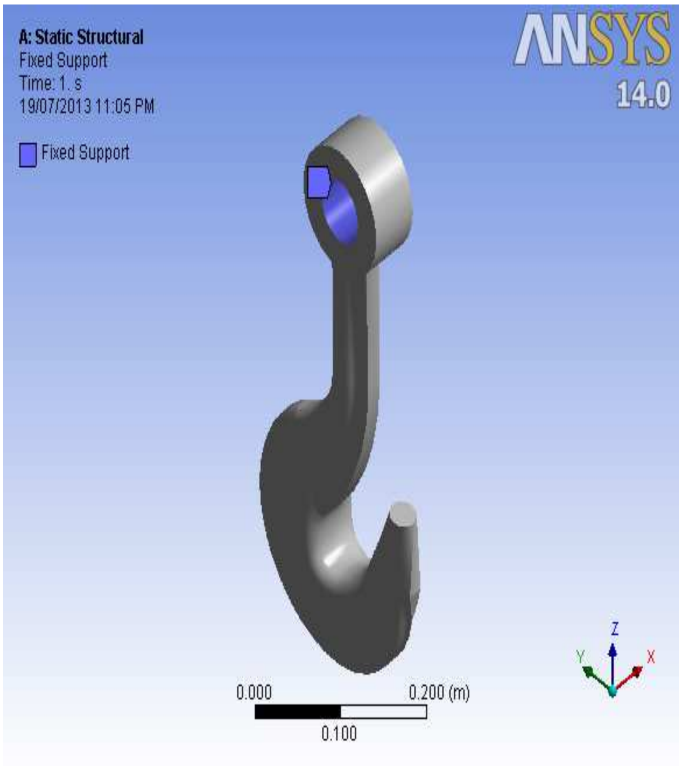


**Fig6.** The fixed support

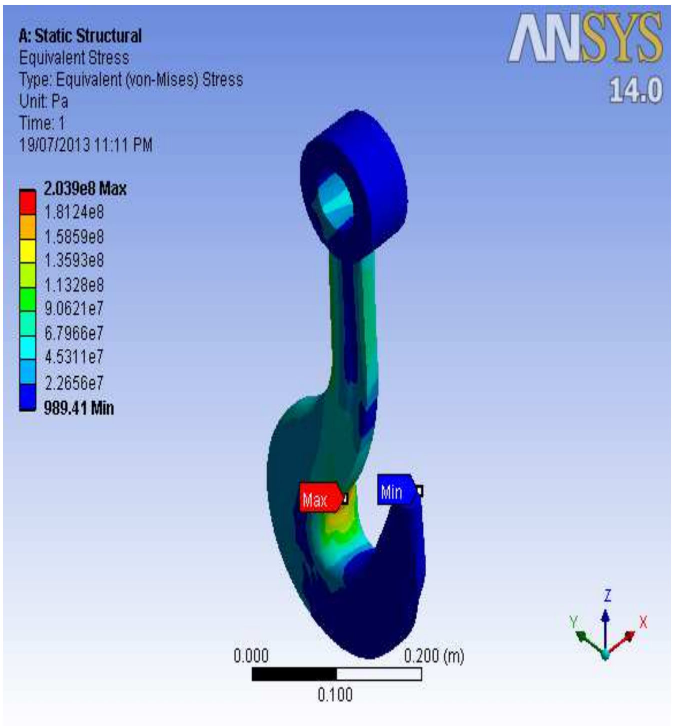


**Fig8.** Equivalent (Von-Mises) stress

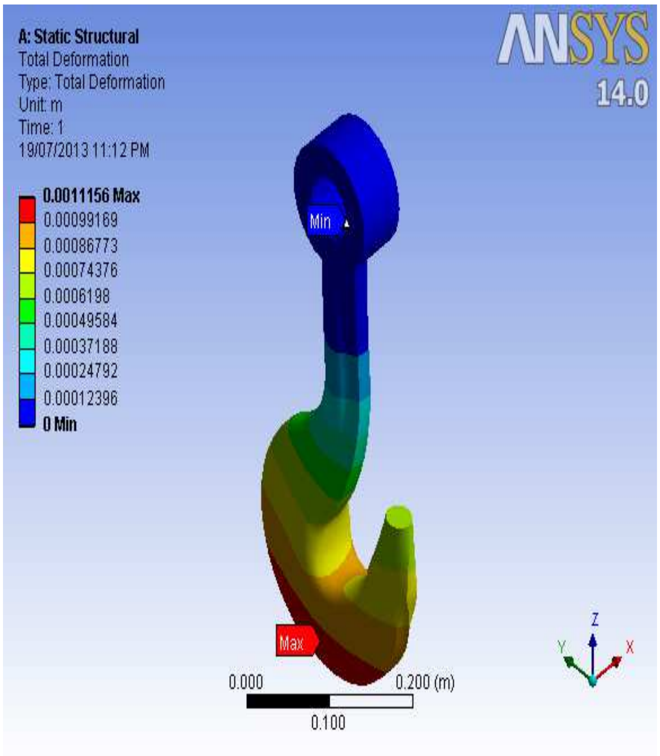


**Fig.9.** Total deformation

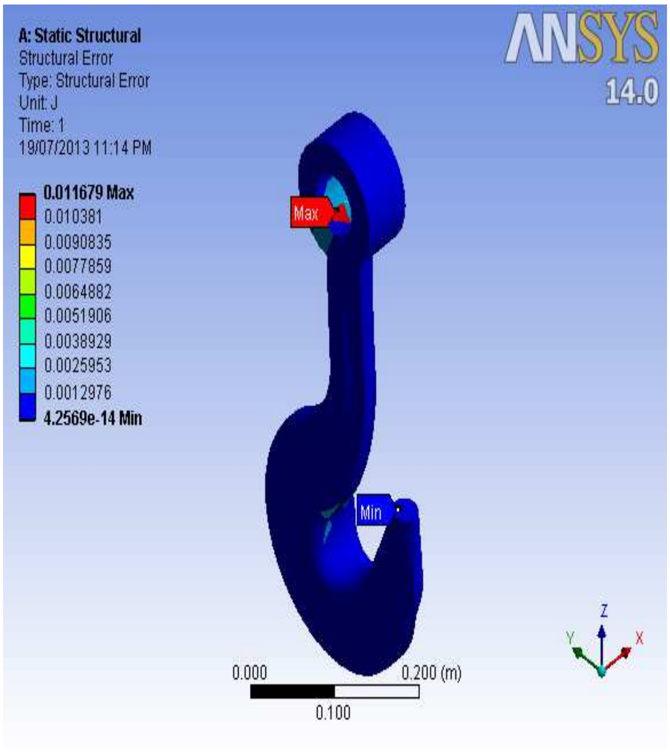


**Fig10.** Structural Error

## 8. COMPARISON OF STRESSES

Stresses obtained for the hook by ANSYS, DIN standard 15 400 [7] and Winkler-Bach theory is tabulated below:

**Table4.** Stress obtained by various methods

| Method | Stress (N/mm^2) |
|---|---|
| ANSYS | 203 (tensile) |
| DIN Standard | 200 (tensile) |
| Winkler-Bach formula for curved beams | 183.93 (tensile) |

Variations in stresses obtained by various methods are tabulated below:

**Table5.** Variation in stress values

| Method | Variation (%) |
|---|---|
| DIN Standard v/s ANSYS | 1.5 |
| DIN Standard v/s Winkler-Bach formula | 8.73 |
| ANSYS v/s Winkler-Bach formula | 10.36 |

## 9. RESULTS AND DISCUSSIONS

The induced stresses as obtained from Winkler-Bach theory for curved beams, explained in the section 5.1, are compared with results obtained by ANSYS software. The results are in close harmony with a small percentage error of 10.36%. Probable reasons for variation might be due to following assumptions (1) loading is considered as point loading in case of Winkler-Bach Formula calculation while it is taken on a bunch of nodes in ANSYS .(2) principal cross section is assumed to be perfect trapezoidal. (3) Assuming sections that are initially plane remain plane after bending. The complete study is an initiative to establish an ANSYS based Finite Element procedure, by validating the results, for the measurement of stress with Winkler-Bach theory for curved beams. This model has an important meaning to design larger tonnage lifting hook correctly.

## 10. FUTURE SCOPE

Further it is advisable to conduct photo elasticity test for the crane hook under investigation in order to get better insight for stress concentration.

## REFERENCES

[1] Yu Huali, H.L. and Huang Xieqing, "Structure-strength of Hook with Ultimate Load by Finite Element Method", Proceedings of the International MultiConference of Engineers and Computer Scientists, 2009 Vol II IMECS 2009, March 18 - 20, 2009, Hong Kong.

[2] “Advanced Mechanics of Materials”, by A P Boresi and O M Sidebottam, John Wiley and Sons, 4th Ed. 1985, ISBN: 0-471-84323-7.
[3] DIN 15 401 Part 1, Lifting hooks for lifting appliances; single hooks; Unmachined parts Deutsche Norm, 1982.
[4] Rashmi Uddanwadiker, “Stress Analysis of Crane Hook and Validation by Photo-Elasticity”, Engineering, 2011, 3, 935-941.
[5] “Strength of Materials”, by Sadhu Singh, 9th Ed. 2008, ISBN No: 81-7409-048-7.
[6] http://www.steel-tube-steel-pipe.com/Prcoducts/StE-355-Youngs-modulus.html
[7] DIN 15 400, Lifting hooks; materials, mechanical properties, lifting capacity and stresses Deutsche Norm, 1990.