

Fatigue Behavior of Tublar K-Joint in HSS

Manoj Pal (Asst. Professor)

Department of Mechanical Engineering
SRM University, NCR Campus
Ghaziabad, India

Gulfirdaush Kalam, Manas Srivastava, Abhishek Raj

Department of Mechanical Engineering
SRM University, NCR Campus
Ghaziabad, India

Abstract—It involves the fatigue behavior of K-Joint made of HSLA material using numerical analysis. Use of High Strength Steel instead of using normal grade steel makes the structure lighter and increases the strengths. The fatigue crack growth initiation is always observed at the chord crown toe location irrespective of the stress whether compressive or tensile. Stress distribution plays an important role in the geometry of K-Joint under fatigue stress. As observed, location around the weld has the highest hotspot stress are highest still chord crown toe are at most critical fatigue risk, hence stress investigation have to be performed. The stress in the region surrounding the weld is computed using numerical simulation and the component with the highest residual stresses are discussed.

Keywords – *Fatigue, Chord crown, stress, Hotspot stress*

I. INTRODUCTION

K-Joints are mainly used in steel bridges and in offshore structures. Where it is subjected to cyclic loading and may experience crack in due to fatigue. There are high stress concentrations due to traffic loading and other factors that develop fatigue crack. This study evaluates the stresses developed due to cyclic loading of K-Joint geometry.

Finite element Simulations are used in these papers for cyclic loading of S-355 Steel. To get accurate result for the joint result a detailed 3-D finite element model of the geometry was formed and simulated. Different material properties were assigned to HSS material. The use of rectangular hollow structural shapes(HSS) has been used because of a strongly effective cross section against compressive and torsional stresses, material cost is reduced also it provides high strength to weight ratio.

II. PROBLEM FORMULATION

In designing mechanical component and structures that satisfy the mechanical strength criteria may fail due to the phenomenon of fatigue. Fatigue mainly occurs due to cyclic loading of the material or structure. The nominal maximum stress value does not have to be more than the strength of the material (ultimate tensile stress or yield stress limit) for the

damage to be done constant loading and unloading cycles are the main reason for fatigue failure. When the loads are above a certain limit cracks begin to form at the stress concentrators. The crack will grow in size and reach a critical value then the crack will propagate suddenly and structure will fail hence fatigue strength, fatigue limit and endurance limit are one of the major factors governing the design of structure and the selection of material.

So the shear of damage done by a fatigue is large, so we have to manage fatigue related problems in an appropriate way.

III. OBJECTIVE

In this study we are concentrating on the fatigue life and different aspects of fatigue of K-Joint that are used in offshore structures rampantly. The fatigue behavior of structure is determined by 3-points; loads, material and geometry.

In this study the loads are defined, the material is high strength steel and the geometry is defined as per the AISC code so in here we are have used HSLA(high strength low allow steel) S-355 rather than normal grade steel and give loading criteria and study their behavior. In this Project HSLA Steel (S355) is used as the strength of the steel is more also high Strength to Weight Ratio ensures the Weight of the structure is reduced which reduces the intake of raw material hence less consumption of energy.

IV. FINITE ELEMENT ANALYSIS

A Three dimensional mechanical finite element analysis is performed to simulate the stress generation and the no of cycles during cyclic loading of the K-Joint,so a validated finite element model was developed and results were obtained that can be compared with future experimental data.

A. Geometry of the connection

In this phase a finite element model was created using CATIA V5 and was analyzed.

For the chord member a circular HSS is used with following dimension

- (i) Diameter: 168.3 mm
- (ii) Thickness: 20 mm.
- (iii) Length: 420 mm

For two branch Members Following dimensions are used

- (i) Diameter: 88.9 mm
- (ii) Thickness: 8 mm
- (iii) Length: 80 mm

Therefore a branch to chord Diameter ratio of 168.3 mm was set up. And there was a gap given to the branches of 38.4 mm. And the Braces are attached to the chord at an angle of 60°. In the AISC Manual, Section K-2, criteria and limits have been set up for Circular HSS connections.

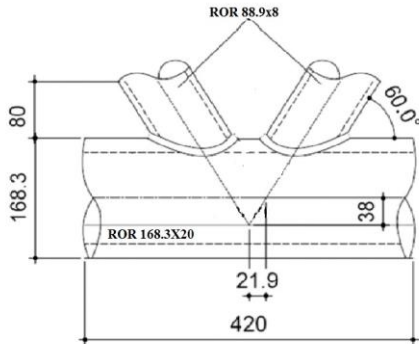


FIGURE 1. Schematic of K-Joint

B. Physical Parameters

TABLE 1. Physical Properties of S355 Steel

Elastic Modulus (Mpa)	200
Thermal Expansion Coefficient (10 ⁻⁶ /K)	17
Density (kg/m ³)	8000
Tensile Strength(Mpa)	1158
Yield Strength(Mpa)	1034
Poisson's Ratio	0.3
Ultimate Tensile Strength(Mpa)	510
Specific Heat (J/kg-K)	477

C. Finite element modelling

ANSYS was used for the finite element analysis of the K-Joint that was made using CATIA V5. Here 3-D solid elements are used for the element type of the K-Joint. 20 node solid element, solid 186 is used it has 20 nodes having 3 degree of freedom per node. Free meshing is used for meshing the desired geometry. Boundary conditions are-

- (i) Load on branch = 930000
- (ii) Load on chord = 2800000

One of the faces of chord is fixed whereas rolling connection is applied to the other face of the chord in its axial direction.

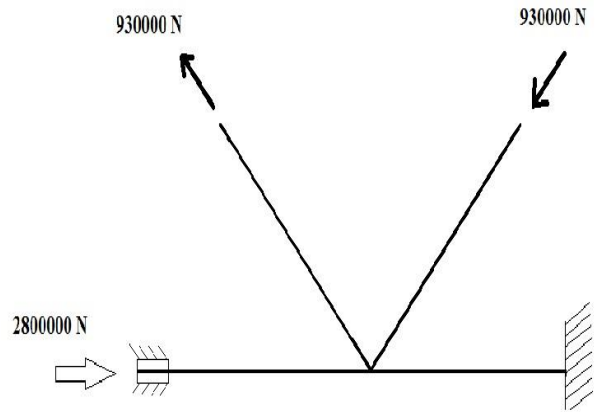


FIGURE 2. Loading Of K-Joint

V. ANALYSIS PROCEDURE

ANSYS Workbench 14.0 is used for the analysis part of the project all the forces and loading concentrations are applied in ANSYS workbench 14.0,using this existing stress are also determined for the solid element and the factor of safety life is calculated.

A. Equations for fatigue analysis

Three Basic equations are as follows-

$$\sigma_t = P/A \quad (1)$$

$$\sigma_b = My/I \quad (2)$$

$$\tau = Mr/J \quad (3)$$

Stress Concentration Factor

$$K_t = \frac{\text{Actual Stress}}{\text{Nominal Stress}} \quad (4)$$

Endurance

$$S_e = 0.5S_{ut} \quad (5)$$

$$S_e = K_a K_b K_c K_d S_e \quad (6)$$

Number of cycles

$$\log_{10} N = \text{Fatigue Life} \quad (7)$$

Fatigue Strength

$$\log_{10} S' = \text{Number of cycles} \quad (8)$$

Factor of safety

$$\text{FOS} = \frac{\text{Maximum Allowable Stress}}{\text{working Stress}} \quad (9)$$

Where S_e=endurance

N = no of cycles

S' = bending stress

S_{ut} = ultimate tensile strength

K = modification factors

VI. RESULTS

The stress measurements were conducted using ANSYS workbench 14.0 although the structural displacement and results derived from the results of finite element analysis are not 100 % precise but are accurate enough. The strain stresses etc. are derived from the nodal displacement solutions

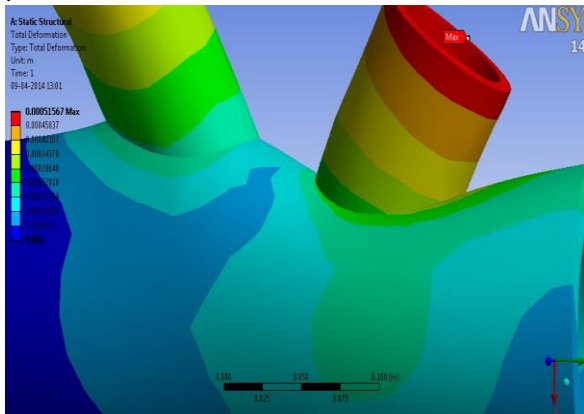


FIGURE 3. Displacement solution of K-Joint

The shown Figure3 describes the displacement of nodes under loading as specified before. In this the large displacement vector is shown in the branch HSS also the displacement in compression end was larger than the tension end also the chord member deformation followed the behavior of branch member. Large displacement appeared in the top surface of the chord member.

The maximum displacement occurred at weld connection at the heel part of HSS hence weld connection faces a maximum displacement also the stress exceeds the yield point. Therefore the failure is defined as the base metal tear out at the chord member. This large deformation is also the reason of connection failure.

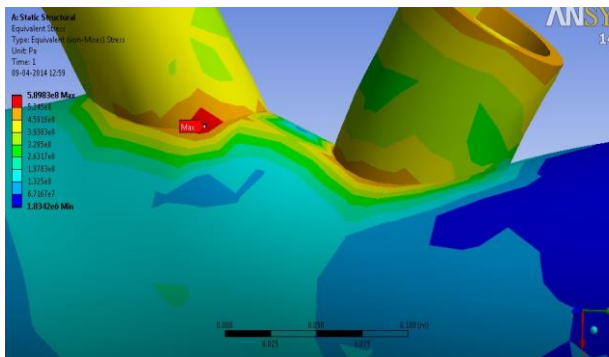


FIGURE 4. Stress Distribution of K-Joint

A. Stress Result

Stress distribution is also investigated to find out the correct failure model and location. Von Mises stress gives the most meaningful stress map (σ_v) it is calculated by combining stresses in 3-D and 2-D so the stress result are typically Von Mises form as shown in the Figure4 the stress maps conforms the failing criteria discussed above .

B. Life

This result contour plots shows the available life for the given fatigue analysis. Here the loading is of constant amplitude

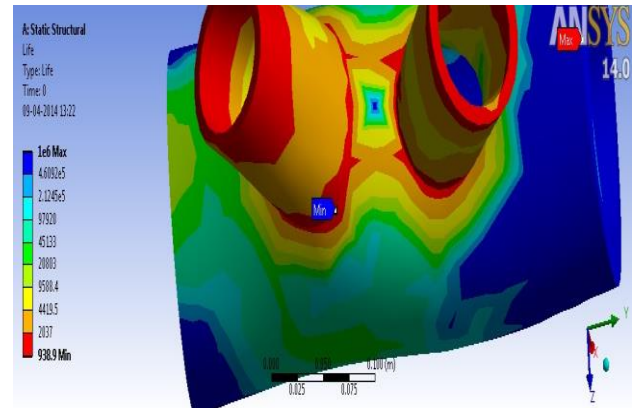


FIGURE 5. Fatigue Life Distribution of Nodes

Here the number of cycles is represented as 989 before the part will fail due to fatigue .

B. Biaxiality Indication

In this analysis we have carried out biaxiality assessment to calculate the stress strain response it also affects the type and severity of fatigue damage.

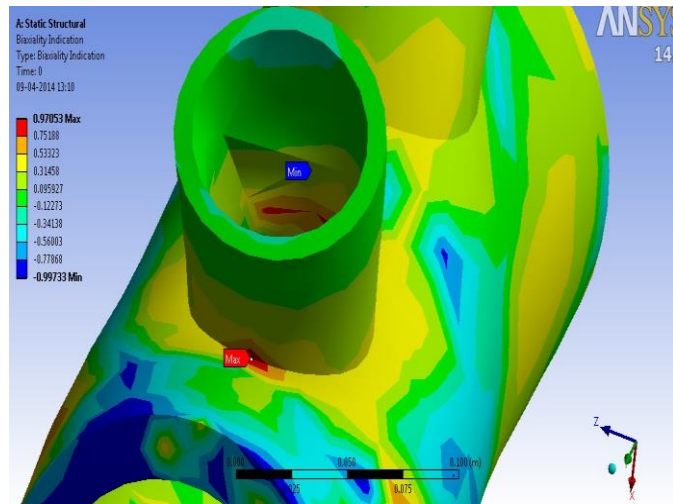


FIGURE 6. Biaxiality Distribution

Here the minimum result is -0.997 showing the maximum shear plane where cracks tend to initiate also the maximum

biaxial stress is +0.970 showing that the heel part is more susceptible to damage for the same level of shear strain.

C. Fatigue Sensitivity

There is a plot for showing the fatigue result as a function of loading at the critical location of scoped loading. Different life for different loading condition of the K-Joint is shown in the Figure 7.

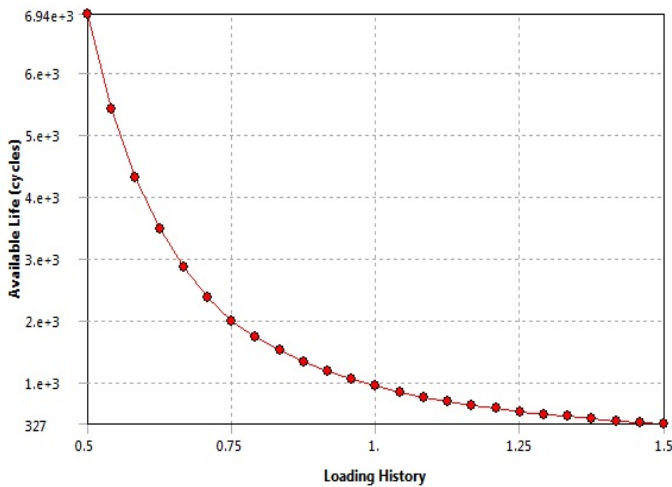


FIGURE 7. Life Cycle vs. Loading History Graph

VII. CONCLUSION

The simulation of fluctuation load is based on ANSYS analysis and the following conclusions are derived from the results of Finite Element Analysis. In this the mainly the chord member base metal tear out and branch member base metal tear out were found as the failure modes of the K-Joint. Here the stresses at the chord region bounded with a brace by a welded joint is maximum ($5.89 \times 10^8 \text{ Pa}$) here the stress are less than the yield strength of the selected S-355 steel hence the breakage as discussed above is less possible while using this steel in comparison with other steels Biaxiality index result shown above gives us a comparative areas where the damage done to the structure by Fatigue will be maximum as we can see the maximum result is 0.97 at the heel of the braces the region is more susceptible to Fatigue damage hence steps should be taken to increase the fatigue life of the area. Life of the K-Joint under the load is shown the result (989 cycles) which can be increased by inducing surface compressive stresses, protection from atmosphere, improvement in welding like avoiding severe stress concentrations ,modifying weld notches, use TIG dressing to improve weld profile.

REFERENCES

[1] Zhao,Gang. Finite element analysis and design of welded HSS connections.2006

[2] Hajro Ismari,Zijah Burzic,Nenad J.Kapor,Tugomir Kokelj Experimental Investigation of High-Strength Structural Steel Welds
 [3] C. acevedo, J. M. drezet and A. Nussbaumer.Simulation of welding stresses for fatigue design of welded tubularconnections
 [4] C. Acevedo, J.M. Drezet, J.P. Lefebvre, L.D'alvised, A.Nussbaumer.Residual stresses in as-welded joints: finite element modeling and neutron diffraction stress measurements
 [5] Specification for Structural Steel Buildings ,AISC 2005 Section k-2
 [6] A.Chattopadhyay,G.Glinka. Stress Analysis and fatigue of welded structure.
 [7] Yuzhuo Jiaa, Zhaohua Ruana.Numerical Analysis on Residual-Stress in Direct Welded Joint of Long Span Transmission Steel Tower.
 [8] Dragi Stamenković, MSc (Eng) Ivana Vasović, BSc (Eng). Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates