Concentrated stress location areas for welded tubular T-joints under deflected bending load

E. CHOUHA[1], S.E. JALAL[1,2], Z. EL MASKAOUI[1,2], A. CHOUAF[1]

1. Laboratory of Control and Mechanical Characterization of Materials and Structures, High National School of Electricity and Mechanics, Hassan II University, Casablanca, Morocco
2. Team of Materials and Structural Mechanics, High National School of Electricity and Mechanics, Hassan II University, Casablanca, Morocco

elhoussinech@gmail.com, se.jalal@ensem.ac.ma, z.elmaskaoui@ensem.ac.ma, a.chouaf.ensem@gmail.com

Abstract

The welded tubular joints can be used in several industrial fields including, metallic bridges, telecommunication tripods, oil platforms and pipelines. These structures can be damaged by concentrated stresses located in the toe weld vicinity. Semi-parametric formulas, currently available in the literature, can be used to predict the value of the stress concentration factor without given the hot spot location. In this work, a Finite Elements modeling was carried out for a T-joint, in order to have the concentrated stress value around the weld toe. This work was investigated for a series of combined loads.

Keywords: *Tubular Joint, Finite Elements Method, Stress Concentrated Factor, Hot Spot, Linear extrapolation method.*

1. Introduction

Generally, the huge tubular structures are composed of several structural elements which are assembled together by welding process in order to create welded joints called joint. This manufacturing way presents some structural discontinuities that lead to have a fertile area of stress concentration.

Welded tubular joints exist in two categories; simple and complex. The first one is planar joints with no overlap, while the second is multi-planar or overlapped.

Under external forces, these junctions have stress concentrations in the vicinity of the weld toe. This concentration is located in a point called hot spot. If this place is a bit wide, it is called hot zone.

This paper will focus on the hot spot location for tubular T-joints. This work will be carried out for several deflected load cases.

2. Sizes and mechanical properties

Fig. 1 presents the specimen geometry. The Joint's material is S235 steel, E= 210 GPa and υ=0.3 are the Young modulus and the Poisson ratio respectively. ~~In~~ This study is performed in elastic phase that the reason for why the use of yield stress doesn't have any impact.

The yield limit has no impact on the results because this work is carried out in the elastic phase.

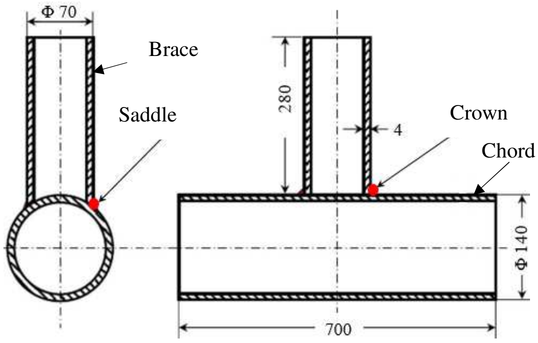

Fig. 1 : Specimen sizes [mm]

3. Numerical modeling

The accuracy of the numerical results by the finite element method depends on several factors; the type of elements used, mesh refinement, particularly in the vicinity of sensitive areas and the integration scheme used.

Several authors have adopted a 2D mesh based on quadrilateral elements, type thin shells, with 4 or 8 nodes. However, this gave results consistent with the experimentation. Chan et al. [1, 2] and Morgan and Lee [3] used this kind of elements.

If the machines allow, the mesh can be made using solid hexahedral elements such hexahedral with 8 or 20 nodes. This second technique is preferred by Karamanos et al. [4] and Chiew et al. [5]. According to these researchers, to use shell elements is inaccurate and linear extrapolation (Fig. 4) specified by the IIW [8] can't be applied to find the concentrated stress.

The mesh of the studied joint is made by using hexahedral solid elements with 8 nodes while instructions based on ARSEM recommendations [7]. Mesh refinement (Fig.2) is performed in the weld toe vicinity, in aim to allow the linear extrapolation method applies.

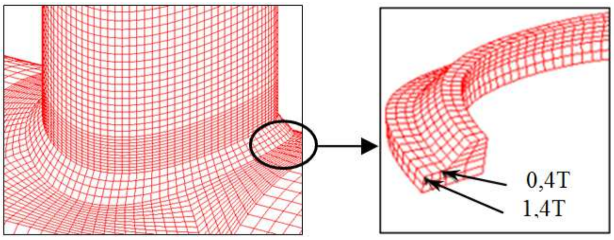

Fig. 2: Mesh refinement and linear extrapolation point

.

4. Numerical model validation

To valid the numerical model, the influence of dimensionless parameter τ (τ = t / T) of the T-junction on the FCC has been carried out. The result was compared with that based on the parametric equations of Efthymiou [9] and Lloyd [10] (Fig.3).

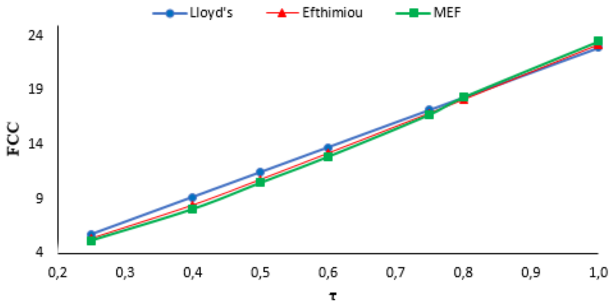

Fig.3: Validation Comparative study

We note that all curves are almost confused, so the model used is validated.

5. Results and Discussion

For each deflected load, the local stress assessment, around the weld bead, is based on the linear extrapolation method (Fig. 4), to avoid the problem of geometrical singularities. The purpose of this normalized method is to take the shape of the stress in a contour spaced from that of the cord toe by a distance of 0.4T and at the contour spaced at 1.4T. Linear extrapolation will be applied at each node of the weld toe. Thus, local constraints are calculated (Fig. 4).

The applying of this method must be taken into account in meshing step (Fig. 2). Otherwise, the results will not be reliable.

Figure 5 shows the extrapolated local stress values at the weld toe for the force equal to 3764.7 N applied at the free end of the brace, with both ends of the chord are fixed. This force was applied in ten angular directions, from 0° to 90° with a step equal to 10° counted from Crown point (ψ= 0°) (Fig.7). The ten curves are presented in the same graph (Fig.5). For the reason of the geometric double symmetry of the T-joint, the angle of the applying force is limited to 90° and all rest can be deduced easily.

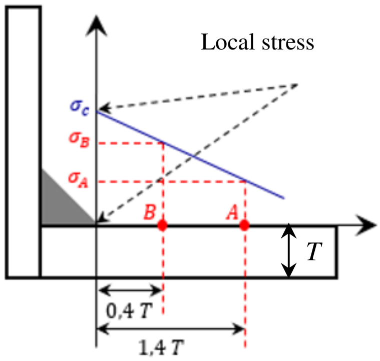

Fig. 4: Linear Extrapolation Method [8]

Three different cases are defined. For ψ = 0°, we have in plane bending case, because of the deformed shape remains always in the plane formed by the axes of purity of the joint. For ψ = 90°, we talk about the out of plane bending. And for an angle comprised between 0° and 90°, we talk about deflected bending.

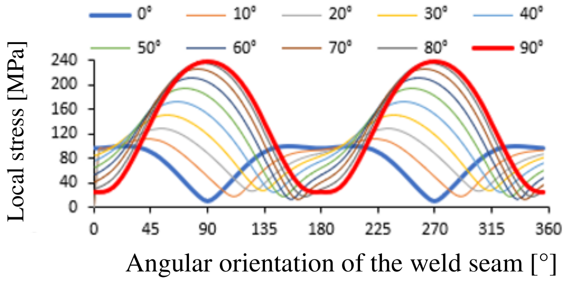

Fig.5: Local Constraint along the weld toe

Figure 5 then shows 8 deflected bends. We note that the hot spot moves towards the Saddle point when stepping from plane flexion to out-of-plane bending. This can be seen clearly in Figure 6, in which we note that the extrapolated local stress is minimal in the case of the in plane bending (ψ= 0° or 180°), and maximum in the case of out-of-plane bending. (ψ= 90°). These results are in agreement with Pang and Lee [6].

From our results, we have represented the maximum local stress as a function of load orientation (Fig. 6).

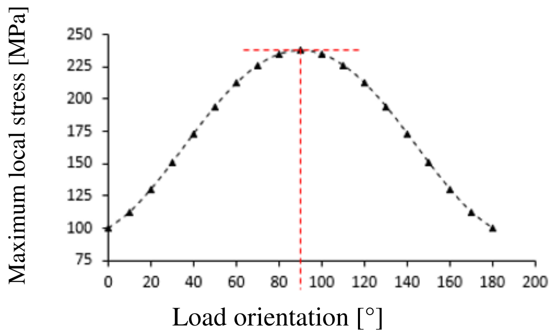

Fig. 6: Maximum local stress variation in function of load orientation

We find that the maximum stress in the case of in plane bending is limited to 99.69 MPa, and is located between Crown and Saddle points. So, we have four locations where the constraint is maximum. In the case of a deflected load, the maximum stress is 273.52 MPa. So, the ratio is equal to 2.38. This stress concentration is located exactly at the two Saddle points.

6. Conclusions and Outlook

In this paper, the response of a T-joint to deflected loads was analyzed. Thus, one could draw the conclusions mentioned below:
- The T-joint resists 2.38 times better in the case of in plan bending compared with the out plane bending case;
- The maximum local stress has a tendency towards the Saddle point when the angle of the applying force tends towards the out-plane bending. The results found show that Crown point is always safe;
- For all the cases studied loads, the most dangerous is the out-plane bending.

In the huge structures, the joints are subjected to a combined loading; tensile, in-plane-bending and out-plane bending. This combined loads, as part of this study, will simulate all the probabilities of loading and will establish a maximum safety limit (red line of stress concentration factors). To do this, we have to:
- Exploit the linear extrapolation method to measure, numerically, the nominal stress and, consequently, the stress concentration coefficient;
- Study the plastic limit states of these structures, based on a material with nonlinear kinematic hardening;
- Consider the study of other kinds of joints with different shapes like Y, K, N, KT, DT, DY, DN, DKT, etc., with and without overlaps.

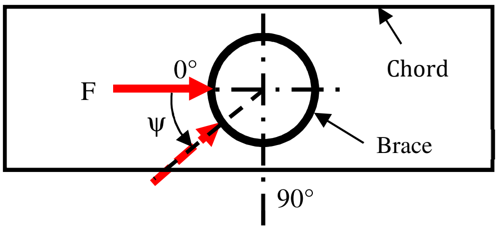

Fig. 7: Force applied according to the inclination angle ψ

Références

[1] Chang E. and Dover W.D., *'Parametric equations to predict stress distribution along the intersection of tubular X and DT joints'*, International Journal of Fatigue, Volume 21, Pages 619-635, 1999.

[2] Chang E. and Dover W.D., *'Stress concentration factor parametric equations for tubular X and DT joints'*, International Journal of Fatigue, Volume 18, Pages 363-387, 1996.

[3] Morgan M.R. and Lee M.K., *'Parametric equations for distributing of stress concentration factors in tubular K-joints under out-of-plane moment loading'*, International Journal of Fatigue, Volume 20, Pages 449-461, 1998.

[4] Karamanos S.A., Romeijn A. and Wardenier J., *'Stress concentration in tubular gap K-joints: mechanics and fatigue design'*, Engineering Structure, Volume 22, Pages 4-14, 2000.

[5] Chiew S.P., Soh C.K. and Wu N.W., *'General SCF design equations for steel multiplanar tubular XX-joints'*, International Journal of Fatigue, Volume 22, Pages 283-293, 2000.

[6] Pang, H. L. J. and Lee, C. W., '*Three-dimensional finite element analysis of a tubular T-joint under combined axial and bending loading*', International Journal of Fatigue, Volume17, Issue 5, Pages 313-320, 1995.

[7] ARSEM, *'Welded tubular joints'*, Paris Technip, 1987.

[8] International Institute of Welding (IIW), '*Fatigue design of welded joints and components*', XIII-1539-96/XV-845-96, Abington Publishing, 1996.

[9] Efthymiou M. and Durkin S., '*Stress concentration in T/Y and gap/overlap K-joints, Proceedings Behavior of Offshore Structures*', Delft, The Netherlands, Pages 429-440, 1985.

[10] Lloyd's Register of Shipping, '*Stress concentration factors for simple tubular Joints*', HSE books, 1997.