CASE STUDIES ON FINITE ELEMENT MODELING OF WELDED JOINTS

BY

PATRICK HEINEMANN*, DORINA ISOPESCU and SEBASTIAN GEORGE MAXINEASA

„Gheorghe Asachi” Technical University of Iași, Faculty of Civil Engineering and Building Services, Iasi, Romania

Received: March 04, 2021
Accepted for publication: May 16, 2021

Abstract. Joints made out of tubular steel members are often used in industrial constructions, like offshore, trusses or tree-shaped columns. The joint area is the weakest point in a truss structure. Beside the normal static behaviour many non-linearities due to the geometry or to the welding process have to take into account. Steel hollow sections are often used. Much research was made on welded nodes so far, but mostly it is valid for limited geometries or load cases. A review of the main aspects in designing welded nodes are given.

The second part is about a study of two aspects. The first aim is to investigate the influence of the joint angle on the resistance of a T-joint made out of Circular-Hollow-Sections or Squared-Hollow-Sections under a tensional load. To archive the most realistic results, the second aspect, a numerical study on the finite element types and the element shape functions will be made.

Keywords: Hollow Sections; Numerical Simulation; Joint Angle; Tubular T-Joint; Steel

*Corresponding author; e-mail: patrick.heinemann@tuiasi.ro
© 2021 Patrick Heinemann et al.
This is an open access article licensed under the Creative Commons Attribution-NonCommercial-NoDerivatives 4.0 International License (CC BY-NC-ND 4.0).
1. Introduction

For industrial buildings or extravagant structures, the main focus is on their stability and resistance. At the same time, it must be a very low weight to reach the requirements to the height or length (e.g. at widespan constructions). This is the reason why trusses, especially together with hollow profiles are often used. Rectangular (RHS) or square hollow sections (SHS) can be found in truss structures, bridges and high-rise buildings. Mostly for offshore platforms circular hollow sections (CHS) are used. To calculate the nodes of these structures is the more complex part. There are different types of joints. Fig. 1 shows the types which are listed in the standards. The most common type in industry is the K-joint (Espinosa, 2017).

![Types of joints with hollow profiles (extract from standard)](image-url)

Fig. 1 - Types of joints with hollow profiles (extract from standard)
In tubular joints where two pipes have different diameters the larger pipe is called chord and the smaller one brace. The angle between brace and chord is called inclination angle. The joint area with a small inclination angle is called brace crown heel, with large brace crown toe (Fig. 2b). Figure 2a shows the geometrical parameters and the ratios which will be need for simulations (Azari-Dodaran, 2019) (Espinosa, 2017):

- $D$, $T$: Diameter and thickness of the chord
- $d$, $t$: Diameter and thickness of the brace
- $g$: Gap between the braces
- $e$: Distance between the lines of influence of the braces
- $\beta = (D/d)$: Brace to chord diameter ratio
- $\gamma = (D/2T)$: Chord wall slenderness ratio
- $\theta$: Brace inclination angle
- $\tau = (T/t)$: Brace to chord wall thickness ratio
- $\zeta = (g/T)$: Relative gap between the braces
- $\alpha = (2L/D)$: Ratio between the length and the diameter of the chord

![Fig. 2](image)

Fig. 2 - a) geometrical parameters of a K-joint, b) notation for the brace crown (Espinosa, 2017)

2. Literature Review

In structural analysis the connection of brace and chord uses hinges. In reality there are moments, because of the welding line which is around the brace (Garifullin, 2016). It is necessary to do research on normal-forces and on bending moments. In simulations in relation to joints axial loads, in-plane and out-of-plane (or anti-plane) bending moments can be calculated. In nature different
loadings can appear. There are static loadings, like dead load or cyclic loads, like wind, wave ice or traffic. Also, other loadings like fire-induced elevated temperatures were studied (Azari-Dodaran, 2019). Even when a single load is applied, we can generally observe a multiaxial state of stress/strain in the volume of material surrounding a notch (Marulo, 2018). For nodal points in trusses the fatigue resistance is one of the biggest issues. Many papers and a lot of research were done to that topic. The welding joint is the weakest point in tubular nodes. One reason is the mechanical heterogeneity. On one hand there are different shapes, which are coming together in one point, on the other hand there is the welding line, which is always a weak point. Zamzami, 2019, explained that by comparing the strength of the welded and non-welded components, made of the same material, that there is a significant reduction in the fatigue strength of the welded components. Azari-Dodaran, 2019, stated it more precisely and concluded a difference of nearly 10% in the ultimate strength. Also, there is a difference in load displacement of less than 10%. Azari-Dodaran, 2019, made experimental and numerical research on two- and three-dimensional joints. In normal trusses, for example K-joints are just used in a two-dimensional way. For offshore constructions there is a need of three-dimensional nodes where more than two braces are orientated in different directions. A significant difference could be occurred between the ultimate load values in uniplanar KT-joints and corresponding values in two-planar connections (Azari-Dodaran, 2019). Sometimes because of weight reduction the industry uses different materials. Zamzami, 2019, did some research on hybrid joints, where the chord is made of steel and the braces are made of aluminium. Next to stress addition due to material differences researches have to take geometrical conditions into account. Joints with relatively small $\beta$ and $\tau$ ratios can cause local buckling in the braces (Azari-Dodaran, 2019). When there is a higher $\beta$ the rotational stiffness is also higher. Times now no analytical method exists to calculate initial rotational stiffness of welded tubular RHS joints (Garifullin, 2016). In many papers, authors conclude that a lot of research has to be done to the joints with tubular members.

A very big issue is the resistance of the welding line itself. There are three zones in a welding area with different resistance. Firstly, there is the welding line. Mostly the welding steel is combined with some chemical additions. Secondly there is the Heat-affected-zone (HAZ). This is a small area between the welding line and the welded metal. Because of the high temperature induced by the welding process this area gets some additional stresses. And the third aspect is the welded metal. Many papers are about the welding lines itself and especially about the microstructure in the HAZ. In general, there are different forms of welding lines, like butt-welded, cruciform-welded, lap-welded, tee-welded and fillet welds (Zamzami, 2019). Depending on the form of the welding line various stabilities of the welding joint can be reached. By using fillet welds instead of butt welds the stiffness of the joints can increase, especially for tubes with little sections (100-120 mm), in mean 1.5 times for S355 and 2 times for S700
There are different methods to model or idealize the nodes. Some authors omit the width of the welding line and idealize it as part of the brace. Garifullin, 2016, elaborated for fillet welds an equivalent width of the brace. He added the diameter of the brace with a part of the weld, because the welds’ material can’t be neglected.

2.1 Stress Distribution

Stress is the main criteria in stability and resistance of structures. When there is a constant stress distribution in the cross section it is less complex to calculate. But due to different geometries or heat influence like in the welding process stress concentration in a very small area can happen. Stress concentration is a complex problem, because it depends on many factors like: weld-size effect, thickness of brace and chord, fixing the point of extrapolation, loading conditions in the brace and chord or the type of material (Saini, 2016).

There are three different types of stresses (Saini, 2016):

1. Nominal stress

The nominal stress is the stress which is induced by axial loads or bending moments. The stress can be calculated by using the simple beam theory. The physical notation is \( \sigma_{\text{nom}} \) (Eq.1). The nominal stress does not include geometric discontinuity or welding effects.

\[
\sigma_{\text{nom}} = \frac{P}{A} \pm \frac{M}{I} y
\]  

2. Geometric stress

The geometrical stress is the stress caused by geometrical differences between chord and brace. Effects are different diameters, inclination angle, shapes or welding radii. Some literature name it structural- or hot-spot-stress. The geometric stress is used to calculate the fatigue life of the structure. The physical notation is \( \sigma_{G} \).

3. Local stress

The local stress can have some reasons. It depends on the quality of the welding. It can occur because of the notch of the welding toe. It is very difficult to calculate this effect. Experimental tests include the microstructure of the welding line. The physical notation is \( \sigma_{L} \). Most of the calculation methods neglect this effect.
2.2 Method for Calculating Fatigue Life

Many structures in the offshore purpose are under influence of static and cyclic loading. As a result, fatigue damage can occur. There are some more or less complex methods to calculate the fatigue resistance of a joint. Every method has got advantages and disadvantages. In the following a little summary of the common methods can be found.

The **Hot-spot stress method (HSS)** (Saini, 2016) (Espinosa, 2017) is calculated at the location where a crack is possible. The HSS is computed as linear extrapolation to the weld toe from stresses at positions near by the welding toe. In general, there are three components of notch stress, which can be seen in Fig. 3. First is the membrane stress, which is constant. The second is the shell bending stress, which varies through the thickness of the material. And thirdly the non-linear stress part which is neglected in this method. The fatigue life is defined in S-N curves. Where S is the stress range and N is the number of cycles to failure. The HSS depends on the material thickness. Because of that it is necessary to multiply the stress range with a thickness correction factor. In this method the ratio between hot-spot stress and nominal stress is called **Stress concentration factor (SCF)**.

\[
SCF = \frac{\sigma_{HSS}}{\sigma_{nom}} \tag{2}
\]

Eq. (2) is valid for a one-load case. With other words when the brace or the chord is induced by a force. In a multi-load case, Eq. (3) has to be used in where k is the loading type.

\[
HSS' = \sum_k (SCF)_k \Delta \sigma_{nom}^k \tag{3}
\]

In the evaluation of an experimental test by the HSS, Eq. (4) and Eq. (5) is needed, with ν Poisson’s ratio, \( \xi_n \) the nominal strain, \( \xi_{||} \) the hot-spot strain perpendicular and \( \xi_{\perp} \) the hot-spot strain parallel to weld toe.

\[
SCF = \frac{1+\nu}{1-\nu^2} \frac{\xi_{||}}{\xi_{\perp}} \tag{4}
\]

\[
SNCF = \frac{\xi_{||}}{\xi_{\perp}} \tag{5}
\]

In the experiments the hot-spot strains can be measured in both directions by strain gauges. The fatigue damage of steel tubular joints is proportional to \( \Delta S^3 \), given by the recommended Wöhler exponent \( m = 3 \) tabulated in guidelines (Espinosa, 2017). Espinosa et al. also explain, that an uncertainty of 20% on the
SCF yields deduces approximately a 70% uncertainty in fatigue life (Espinosa, 2017).

The Mesh insensitive structural stress method (SSM) (Saini, 2016) is a robust method based on the mesh size. In this method structural stress is calculated by nodal forces of Finite Element Method (FEM). A master S-N curve is established for a wide variety of joints including typical tubular joints. The structural stress is the sum of membrane stress Eq. (6) and bending stress Eq. (7). Fig. 3 shows graphical version of the equations.

\[
\sigma_{\text{mem}} = \frac{1}{t} \int_{x=0}^{x=t} \sigma(x) \, dx \tag{6}
\]

\[
\sigma_{\text{ben}} = \frac{6}{t^2} \int_{x=0}^{x=t} (\sigma(x) - \sigma_{\text{mem}}) \left( \frac{t}{2} - x \right) \, dx \tag{7}
\]

\[
\sigma_{\text{nlp}} = \sigma(x) - \sigma_{\text{mem}} - \left( 1 - \frac{2x}{t} \right) \sigma_{\text{ben}} \tag{8}
\]

(Saini, 2016) found out, that the structural stress method is far more effective than conventional hot-spot stress method.

![Graphical representation of stress distribution](image)

Fig. 3 - Components of the stress distribution through the thickness of the weld plate (Saini, 2016)

The Extrapolation methods (Saini, 2016) is an addition for the HSS method. The HSS does not include stresses caused by the welding. For experimental tests it is not easy to measure the stress in the welding toe, because there is no possibility to fix the strain gauge at the welding toe and get realistic results. The strain gauge has to be fixed close to the welding joint in a definitely distance and combines the results with a mathematical extrapolation method. In the literature there are different recommendations for the maximum extension. Most of them are in the range of 6 mm to 0.1 \cdot \sqrt{t} \cdot t. In General, there are two extrapolation methods, the linear and the quadratic. The linear can be used for square and rectangle hollow profiles, the quadratic for circular hollow profiles which will be the interesting part for this thesis. Like Fig. 4 shows, the linear method just needs two measurement points. These points should have a distance of 0.4t and 0.6t from the welding toe. In this \( t \) is the thickness of the tubular member.
To simulate a joint with the extrapolation method a couple of FEA parameters are needed. An explanation can be found at 3.1. Besides this parametric equations are needed to determine the SCFs. There are different equations, the most common ones are the: Kuang, Wordsworth/Smedley, UEG, Efthymiou/Durkin, Hellier, Connolly and Dover, Lloyd’s register, Morgan and Lee equations.

The **Peak-stress method (PSM)** (Meneghetti, 2018) is next to the HSS and SSM a method to calculate the fatigue design in welded joints. It is based on the **Notch Stress Intensity Factor (NSIF)**. With the PSM it is possible to estimate the mode 1 SIF of a crack emanating from an ellipsoidal cavity. The NSIF can be defined by Eq. 9.

\[
K_i = \sqrt{2\pi} \cdot \lim_{r \to 0} \left[ \left( \sigma_{jk} \right)_{\theta=0} \cdot r^{1-\lambda_i} \right]
\]  

where \( i = 1,2,3 \) and \( \sigma_{jk} = \sigma_{\theta\theta}, \tau_{r\theta}, \tau_{\theta z} \)

The stress components along the notch bisector line \( \theta = 0 \) can be calculated by FEA. \( \lambda_i \) is the stress singularity for mode 1, 2 or 3. Cracks appear mainly in a semi-elliptical shape (Djokovic, 2018). This approach should have better accuracy than the other methods. In some literature a more ‘exact’ definition of the K1-K3 NSIFs values are given with Eq. 10-12, which is derived from Eq. 9:
where \( d \) is a global element size to input e.g. in Ansys software. The PSM estimates the NSIF from the singular, linear elastic, opening, sliding and anti-plane FE peak stresses, referred to the V-notch bisector line. This approach seems to be the most complex one in comparison to HS and SSM.

### 2.4 International Standards

The next part is about international standards. It is just a short look into it.

The European Standard EN 1993-1-8 is about connections in steel structures. A summary of the joint types can be found in 1. By focusing the loads, the Eurocode contains only equations for the moment resistance of joints where the angle between the brace and chord is 90 degrees.

Jurčíková, 2012, describes the design methods given by the Eurocode for joints are “complicated, difficult to check and offer a limited scope of use” (geometric conditions, restrictions on material characteristics, certain types of joints of special types of loads).

By considering the current practice outside the standard the common types for braces are RHS (Rectangular Hollow Sections) and H-profiles for the bottom chord. Mostly the inclination angle is smaller than 30° (Jurčíková, 2012). The Eurocode 3, 2010, considers three types of failures for CHS (Circular Hollow Section) or RHS braces connected to I- or H-cross section chords which are shown in Figure 5: The first is the failure of web plate (a), the second is the chord shear failure (b) and the third, the brace failure can be seen in Fig 5c.

Fig. 5 - Failure types (Eurocode 3, 2010)
Like picture 5 shows, the Eurocode does not take into account forces or tensions occurring in individual bars. There is only the load case where one brace is compressed while the second brace gets tension. The chord is always without a load case. With other words, when there is tension force in both braces or tension force in the chord it is automatically offset the standard. The Code only considers joint’s geometry, profile type and yield stress (Jurcikova, 2012).

Osage, 2018, took a look at the American standard. He summarised the fatigue analysis methods of the ASME (American Society for Mechanical Engineering). The ASME defines mainly three methods similar to the methods which were introduced in 2.1:
- Level 2 – Method A: Equivalent Stress Range and Smooth Bar Fatigue Curve (including Hot-Spot Stress method)
- Level 2 – Method B: Equivalent Strain Range and Smooth Bar Fatigue Curve, same as Type A, but with strains and not stress
- Level 2 – Method C: Equivalent Structural Stress Range

3. Case Studies on CHS T-Joint

The numerical study is about the analysis of a T-joint with one brace. In the first step the influence of the joint angle on the equivalent stress and deflection will be examined. Beside this there will be an analysis of different mesh element types, which can be seen at 3.3. The profile type is a CHS which is described in 3.1.

3.1 Description of the Finite Element Model

To simulate the models the Ansys 2019 Software is used. The welding line is excluded, so the cross section of the brace is directly connected to the surface of chord.

Analysed is a T-joint with a 30, 45, 60, 75 and 90° inclination angle between brace and chord. The length of the brace is independent of the joint angle. The braces length is 175 mm and the length of the chord is 350 mm.

The boundary conditions are chosen as fixed at both ends of the chord. It is hard to find a uniform condition for the BCs in literature. In some of them the chord is pivot-mounted like a simple beam, in other papers the chord is fixed. The most realistic model has got a pivot-mounted chord with a torsion spring bedding which is equal to the stiffness of the connected compression member, for example in a tree model.

The load is implemented as a tension force. 10,000 N (10 kN) are affected to the brace cross section with the direction of normal-force.
This paper’s research can be seen as one part in a greater future research project. In this step, to analyse the T-joint, one type of a profile type is picked. It belongs to the family of Circular-Hollow-Section. The analysed profile for the CHS have a cross section of 60.2 x 1 mm, which is similar to a DN 50. In every simulation model the brace and the chord have the same profile type. In further studies a variation of cross sections can be done.

The material is similar to the construction steel S 235. The main properties are given with the density $\rho_s = 7850$ kg/m$^3$, the Young’s modulus $E = 210$ GPa, Poisson’s ratio $\nu = 0.3$ and the yield stress $f_y = 250$ MPa.

### 3.2 Element Mesh

In this study four different three-dimensional element types are used for the T-joint. The first is the linear 4-node Tetrahedral element, the second is the linear 8-node Hexagonal element, the third is the quadratic 10-node Tetrahedral element and the fourth is the quadratic 20-node Hexagonal element.

| Element   | 2D Triangle    | 2D Quadrilaterals | 3D Tetrahedrons | 3D Hexahedrons |
|-----------|----------------|-------------------|-----------------|----------------|
| Linear    | PLANE 42       | SOLID 92          | SOLID 45        | SOLID 185      |
|           | PLANE 182      | SOLID 187         |                 | SOLID 185      |
| Quadratic | PLANE 2        | PLANE 82          | SOLID 92        | SOLID 95       |
|           | PLANE 183      | SOLID 187         |                 | SOLID 186      |

In the software Ansys the different element types have got own names which are listed in Table 1. In this step the mesh of the brace and the chord are...
the same. The element size for the coarser part is 10 mm and 4 mm for the round parts in the edges. The finer mesh in the area of the joint is limited to 1 mm which can be seen in Fig. 6. Beside this the tetrahedral element shapes can be seen.

3.3 Analysis and Discussion

In this section the circular hollow section profile DN 50, which is defined by the international standard ISO with an outer diameter of 60.2 mm, is analysed.

Fig. 7 – Results von-Mises stress

Fig. 7 shows an example for the von-Mises stress distribution on the surface. The highest stress value occurs where the welding line is located in the normal case.

Fig. 8 – CHS von-Mises stresses for the two tetrahedral element types
Fig. 8 shows the results of the tetraedal elements with the linear and quadratic shape functions. The graphs aren’t striking each other. There is a huge difference between the results. It seems that the 30° Tetra-Quad value is unrealistic. Because of this value there isn’t a complete increasing trend (Fig. 9). Beside this the value is very high.

![Fig. 9 - CHS - Mean value of the von-Mises stresses](image-url)

The second analysed aspect is the deflection of the CHS profile. Due to the thickness of 1 mm the deflections are greater than at the standard DN 50 with a 2 mm thickness (Fig. 10). The side panels of the chord’s surface are moving together because of the 10 kN tension force. Because of that a deflection of the
brace happens, too. Taking Fig 11 and 12 into account a very smooth increasing behaviour can be seen.

Fig. 11 - CHS - deflections for the two tetrahedral element types

Fig. 12 - CHS - Mean value of the deflections

A quotient smaller or equal “2” describes the difference between the linear and quadratic Tetra element solution. The steady quadratic-increasing behaviour is valid for the CHS profile type. The stress hasn’t got such a smooth behaviour, but the trend is nearly the same.
4. Conclusions

In the paper a review of the research to T-joints under normal forces and bending moments was given. By designing these joints many intricacies, like fatigue life, have to take into account, beside the static behaviour. The different stress distributions in the welding line area due to welding process or geometrical differences were explained.

In the case study a T-joint made out of Circular-Hollow-Section under a tensional load was analysed. Mainly four conclusions can be drawn:

1. There is a great influence of the joint angle to the von-Mises stresses. By increasing of the inclination angle, the stresses are getting greater, too.
2. If the joint angle increases from 30 to 90°, the deflection will increase nearly by the factor 2.
3. The element shape functions have got a great influence on the von-Mises stresses. The tetrahedral finite element with a quadratic shape function creates greater von-Mises stresses than the linear one.
4. By the use of the quadratic shape function, there will be higher deflections than by the linear shape function. The difference between both types are getting stable while changing the joint angle.

REFERENCES

ANSYS Inc. Products 2019 R3 © 2006-2019
Neda Azari-Dodaran, Hamid Ahmadi. Static behaviour of offshore two-planar tubular KT-joints under axial loading at fire-induced elevated temperatures. Journal of Ocean Engineering and Science 2019; S2468-0133(19)30079-8
DIN EN 1993-1-8, Eurocode 3: Bemessung und Konstruktion von Stahlbauten – Bemessung von Anschlüssen. Deutsches Institut für Normung (2010-12)
Jelena Djokovic, Ruzica Nikolic, Branislav Hadzima, Dusan. Arsic, Libor Trsko. Working life estimate of the tubular T-joint by application of the LEFM concept. Procedia Structural Integrity 13 (2018) 334-339
Marsel Garifullin, Maria Bronzova, Timo Jokinen, Markku Heinisuo, Bostjan Kovicac. Effect of fillet welds on initial rotational stiffness of welded tubular joints. Procedia Engineering 165 (2016) 1643-1650
A. Jurcikova, M. Rosmanit. FEM model of joint consisting RHS and HEA profiles. Procedia Engineering 40 (2012) 183-188
Giuseppe Marulo, Francesco Frendo, Leonardo Bertini, Ali Fatemi. On the application of a critical plane approach to the life assessment of welded joints. Procedia Engineering 213 (2018) 448-458
Jorge Mendoza Espinosa, Sebastian Schafhirt, Michael Muskulus. *Mesh Parametric Study for Fatigue Assessment of tubular K-joints using Numerical Methods*. Energy Procedia 137 (2017) 514-522

Giovanni Meneghetti, Alberto Campagnolo. *The Peak stress method to assess the fatigue strength of welded joints using linear elastic finite element analyses*. Procedia Engineering 213 (2018) 392-402

David A. Osage, Pingsha Dong, Daniel Spring. *Fatigue assessment of welded joints in API 579-1/ASME FFS-1 2016 – existing methods and new developments*. Procedia Engineering 213 (2018) 497-538

Dikshant Singh Saini, Debasis Karmakar, Samit Ray-Chaudhuri. *A review of stress concentration factors in tubular and non-tubular joints for design of offshore installations*. Journal of Ocean Engineering and Science 1 (2016) 186-202

Ibrahim Al Zamzami, Buick Davison, Luca Susmel. *Experimental fatigue curves to perform the fatigue assessment of aluminium-to-steel thin welded joints*. Procedia Structural Integrity 18 (2019) 255-261

**STUDII DE CAZ PRIVIND MODELAREA CU ELEMENT FINIT A ÎMBINĂRILOR SUDATE**

(Rezumat)

Îmbinările execute prin utilizarea elementelor tubulare din oțel sunt folosite în mod uzuial în cazul construcțiilor industriale pentru realizarea fermelor sau a stâlpilor în formă de copac. În cazul structurilor de tip ferme, zona de îmbinare între elementele componente este considerată ca fiind cel mai slab punct. Pe lângă comportamentul static, în cazul acestor îmbinări trebuie luată în considerare și analiza neliniară influențată de diferite aspecte precum geometria nodului sau modul de sudare a elementelor din oțel. Pentru realizarea acestor noduri se folosesc în general elemente liniare de tip țeavă. Până în prezent, au fost realizate mai multe studii cu scopul de a studia îmbinările sudate, însă acestea sunt valide doar pentru un număr limitat de tipuri de geometrie sau de cazuri de încărcare. În prima parte a articolului au fost prezentate principalele aspecte care trebuie luate în considerare în proiectarea nodurilor sudate.

Cea de-a doua parte a fost dedicată unei analize amănunțite a două aspecte principale. În primul rând, s-a urmărit investigarea influenței unghiului de îmbinare asupra unui nod sub formă de T realizat din țevi din oțel cu secțiune circulară sau pătrată solicitate la întindere. Cel de-al doilea aspect urmărit a fost calibrarea rezultatelor pentru a obține o comportare care să descrie fidel situațiile reale prin realizarea unui studiu numeric a formei și tipului de element finit utilizat în modelare.