E3S Web of Conferences **483**, 03017 (2024) https://doi.org/10.1051/e3sconf/202448303017
ISST 2023

The geometrical parameters effect on the degree of bending of multiplanar DTKY tubular joint for offshore platforms

Rudi Walujo Prastianto*, *Nur* Syahroni, *Yeyes* Mulyadi, *Ferdita* Syalsabila, and *Alviani Rahma* Rizkhita

Institut Teknologi Sepuluh Nopember, Ocean Engineering Departement, 60111 Surabaya, East Java, Indonesia

Abstract. The service life of an offshore structure is an important variable to consider in the sustainability of an infrastructure. The service life could be assessed through fatigue assesment of the critical tubular joint considering the hot-spot stress near the weld of the joint. Another important parameter in assessing the fatigue is degree of bending of the joint which represents the stress distribution beneath the thickness. Finite Element Method based analysis makes structure modelling more advanced and detailed for complex structures. In this study, a case of jacket offshore structure tubular joint, DTKY-type, is modelled by using FEM-based software. Stresses in the tubular joint model will be evaluated for axial load acting on the brace members in a tension-compression combination mode. Non-dimensional geometry parameters such as β, γ, and τ are varied to determine their effects on the degree of bending of the joint. The study show that stress distribution could be well obtained at the critical points of the joint and β generally decreases the degree of bending of the joint at the position of saddle and crown. Meanwhile, γ and τ both, generally increases the degree of bending at the saddle and crown positions.

1 Introduction

A jacket sub-structure is a part of a jacket offshore structure comprises of members in the form of steel circular hollow sections (CHS), which are circularly hollow-shaped tubes, commonly known as tubular members. A tubular joint, refers to the connection point between tubular components, where another member, called brace, is welded to the main member surface (chord). The brace can be one or more to transfer the structural load [1]. Based on the branching configuration, two types of tubular joint are classified. Uni-planar is the that have branching configuration between chord and braces in one dimensional plane. Meanwhile, multi-planar consists of more than a plane, in general, two or three planes [2, 3]. More

* Corresponding author: rudiwp@oe.its.ac.id

intersections in multi-planar joints lead to more complex behaviour than uni-planar that encourage the researchers in developing more simple way to analyze.

In a jacket platform, the tubular joint is subjected to continous loading from the environmental conditions, especially waves, making it the most vulnerable part in terms of stress concentration. As a result, the tubular joint is a critical component in the occurrence of fatigue and damage, which significantly affects the service life of the jacket platform.

The service life calculation process typically uses the S-N Curve approach, based on hot spot stress, which seeks the highest stress in critical areas located in welded regions. However, according to Ahmadi's 2019 research [4], different geometry in tubular joints give significant difference of failure cycle count although share the same hot spot stress. These variations arise from shifts in crack growth rates, a factor influenced by the stress distribution across the thickness of the tubular structure. Hence, the fatigue life of tubular joints, frequently encountered in ocean structures, is influenced not only by the hot spot stress value but also significantly affected by the stress distribution throughout the tubular thickness, as indicated by the degree of bending (DoB).

As mentioned earlier, it is clear that the parameter hot spot stress is not enough to describe the fatigue failure mode. Thus, the calculation of current hot spot stress is proposed to consider the degree of bending influence. By including the stress through the member thickness, fatigue life prediction could be more accurate [1]. The DoB than expressed as the rasio between overall stress (σ_B) and bending stress (σ_T), which also consider the member stress (σ_M). Equation 1 shows the mathematical function of DoB.

$$\text{degree of bending}=\frac{\sigma_B}{\sigma_T}=\frac{\sigma_B}{\sigma_B+\sigma_M} \quad (1)$$

Based on the explanation above, this research focus about the influence of dimensionless geometric parameter of multi-planar tubular joints on the degree of bending. The DoB along the weld toe subjected to specific loading conditions is mainly affected by the joint geometry. To investigate the performance of a tubular joint and establish a clear connection between this performance and the geometric attributes of the joint, a set of dimensionless geometrical parameters. The geometric characteristics β, γ, τ, pertain to the tubular member dimensions, specifically diameters of chord (D) and brace (d), the wall thicknesses of chord (T) and brace (t), as well as their respective lengths of chord (L) and brace (*l*). These parameters are crucial for determining the critical locations along the weld joint where the calculation of the DoB values is required. These key positions include the inner saddle, outer saddle, toe, and heel of the tubular structure at the intersection between the brace and chord.

2 Materials and methods

2.1 Finite Element Modeling

In this study, finite element-based, was employed for the purpose of modeling and analyzing DTKY-joints with multiple planes under axial loads. This was done to obtain the DoB needed for the parametric analysis. This section provides a comprehensive overview of the finite element modeling and analysis procedures used. The tubular joints to be analyzed are focused on multiplanar DTKY tubular connections. The selection of these connections is based on the highest unity check values from previous analysis, specifically the global analysis.

The DTKY-joint will be modeled using solid element types. In this geometry modeling, the dimensions of the tubular connections used are identical to the ones in the field model. In addition to modeling the geometry, this modeling will also include the geometry of the welds that connect the chord and brace. To expedite analysis, the modeled tubular joints do not utilize the original connection length. Iti is necessary to truncate the members in accordance with the rules established [7], where the minimum cutting length is 6 times the outer diameter of the member. Here is Figure 1 which illustrates the results of the geometric modeling of the tubular joints.

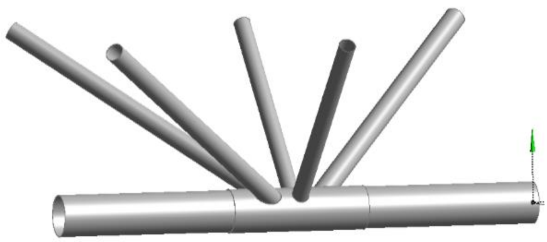
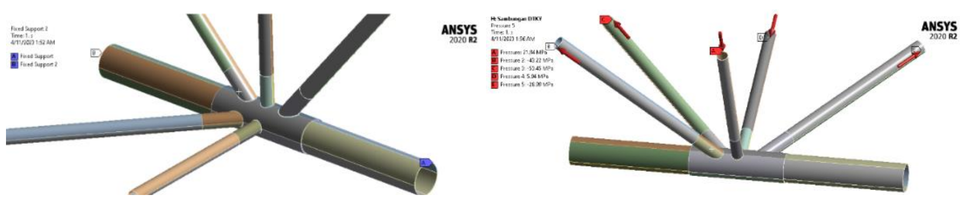


Fig. 1. FEM Model

2.1.1 Load and Boundary Conditions on the Model

The joint model was subjected to boundary conditions designed to replicate the real structure accurately. As shown in Figure 2 (a), the applied loads were a combination of axial forces that consist of three tensile forces and two compressive forces. Loads are applied at the top surface are of each brace from the previous global analysis.

The critical connection model is endowed with boundary conditions to effectively capture the true structural representation. Boundary constraints are imposed at both ends of the chord in the form of fixed supports. In the context of structural FE analysis, the term "fixed support" denotes the comprehensive restriction of all degrees of freedom, thereby precluding any motion of the model elements.



(a) (b)

Fig. 2. (a) Fixed Support; (b) Applied Load

2.2 Mesh Generation

As detailed in the earlier subsection, the author employed solid elements in this finite element analysis to simulate the brace, chord, and welds. This type of element exhibits compatible displacements and is particularly suitable for representing curved boundaries.

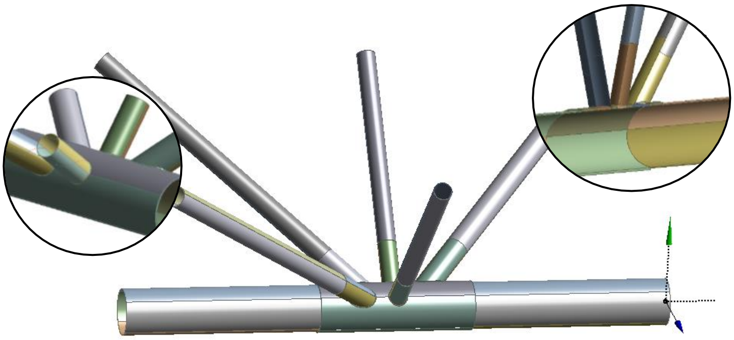


Fig. 3. Mesh Zone

Meshing is a crucial component of finite element based simulation which discritize the model into smaller elements. This method improves the precision of the outputs by breaking down the model into finer subdivisions. The analysis involves augmenting the number of mesh elements in the model and then comparing the results from each stress probe. The addition of the number of mesh elements is continuously carried out until the stress probe reaches a constant value with a maximum error of 2%. Application of mesh in this analysis uses two types of elements, namely tetrahedron and hexahedron elements. Hexahedral mesh elements can be used in sub-zones with more regular shapes or in this model, in areas that are away from welds. Tetrahedral mesh elements are used for the weld modeling. Figure 3-5 show the mesh configuration of the tubular model.

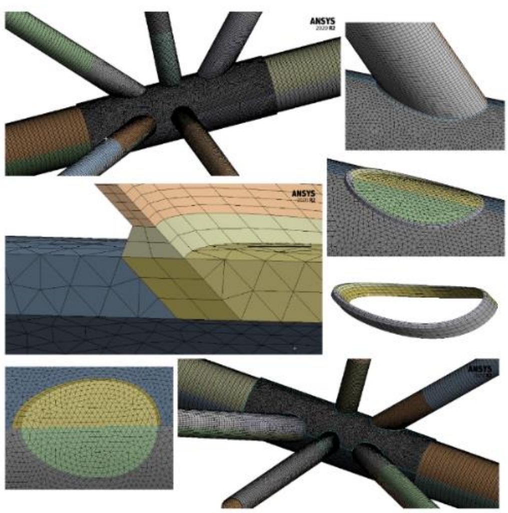


Fig. 4. Generated Mesh of the Model

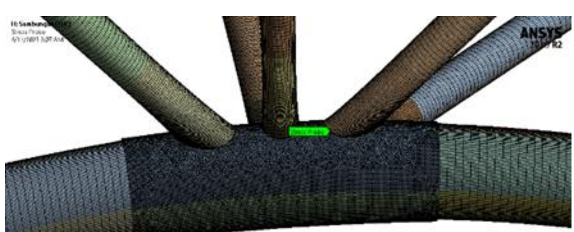


Fig. 5. Probe Stress

2.3 Variation of Geometric Parameters

Variation of geometric parameters is conducted to investigate their influence on the Degree of Bending. The varied parameters are non-dimensional parameters in terms of β, γ, and τ. Each parameter value refers to the range provided by [6], as well as considerations of the main geometry range. The following Table 1 presents the varied geometries applied to the model.

Table 1. Geometrical Parameters

Parameter	Value
β	0.3 ; 0.4 ; 0.5
γ	21 ; 31 ; 41
τ	0.3 ; 0.4 ; 0.5

3 Results and discussion

3.1 Maximum Stress

The analysed output results for the DTKY-joint are based on equivalent stress. Equivalent stress, commonly called von Mises Stress, is the widely used to describe the entire stress tensor (or "state of stress") at any point within the model at a glance. The respective location of maximum equivalent stress is presented in Figure 6 below.

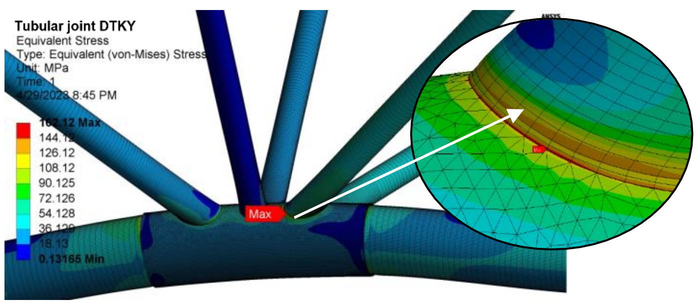


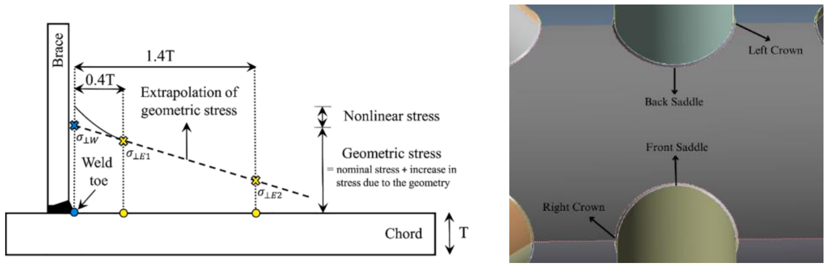
Fig. 6. Maximum Stress

3.2 The Calculation of Hot-spot Stress

Degree of bending require hot-spot stress that happend at the inner and outer tubular member surfaces. The calculation of hot-spot stress in this study is conducted using the linear extrapolation method with 2 reference extrapolation points, namely at the points 0.4thickness and 1.4thickness, where t represents the thickness of the brace [17]. Both of these points have the weld toe as zero point, where the highest stress occurs. The following is an illustration at Figure 7 the hot-spot stress extrapolation points [9].

The calculation of hot-spot stress in this research is exclusively performed on the critical members under the previously described loading conditions. Furthermore, this study focuses solely on stress within four specific regions: Right Crown (RC), Front Saddle (FS), Left Crown (LC), and Back Saddle (BS).

 https://doi.org/10.1051/e3sconf/202448303017



(a) (b)

Fig. 7. (a) Point of Extrapolation Hot Spot Stress (b) Specific Region for Hot Spot Stress

Here are Figure 8 through Figure 11, which present the results of the hot spot stress calculations for each location. Total of nine variation are conducted for each location.

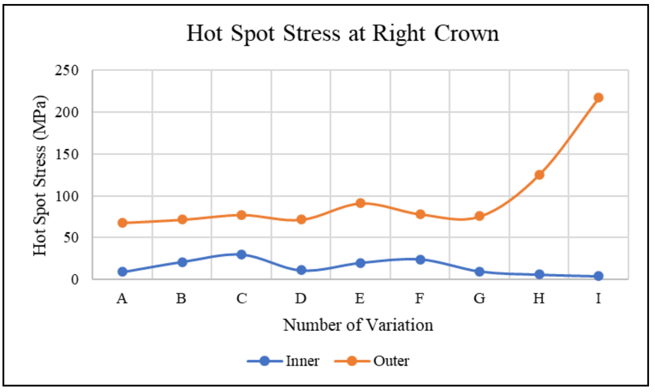


Fig. 8. Hot Spot Stress Result at Right Crown

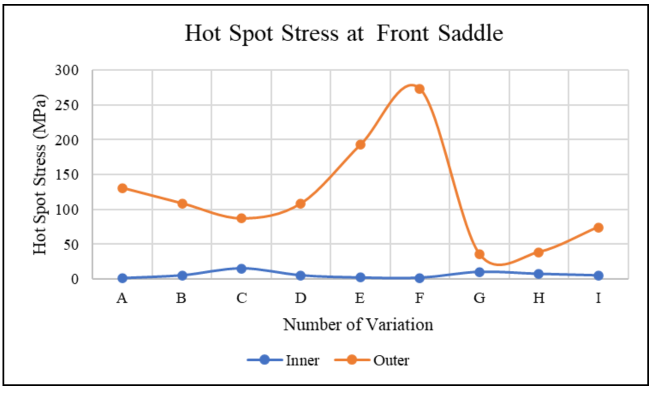


Fig. 9. Hot Spot Stress Result at Front Saddle

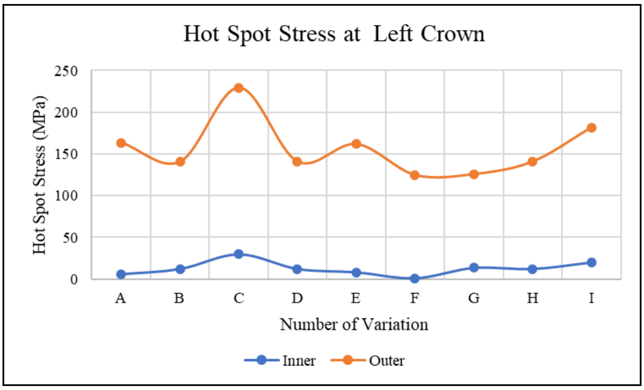


Fig. 10. Hot Spot Stress Result at Left Crown

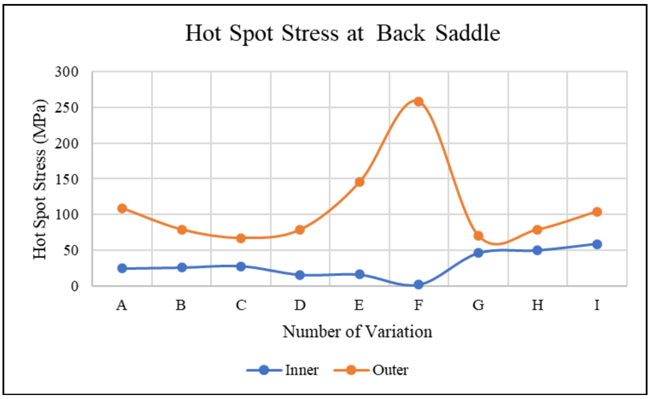


Fig. 11. Hot Spot Stress Result at Back Saddle

At the Right Crown location, the inner hot-spot stress is at variation C with a value of 29.47 MPa, and the outer hot-spot stress is at variation I with a value of 217.34 MPa. Moving on to the Front Saddle location, the inner hot-spot stress is at variation C with a value of 15.02 MPa, and the outer hot-spot stress is at variation F with a value of 273.43 MPa. At the Left Crown location, the inner hot-spot stress is at variation C with a value of 29.80 MPa, and the outer hot-spot stress is at variation C with a value of 229.38 MPa. Finally, at the Back Saddle location, the inner and outer hot-spot stress is at variation I with a value of 58.81 MPa and at variation a with a value of 108.72 MPa, respectively.

3.3 Degree of Bending

Based on the Ahmadi et al. [2], the Degree of bending is obtained from the hot spot stress on the outer and inner surface of the chord as previously shown in equation 1. Then, the equation is derived to Equation 2 with σ_I is the inner hot-spot stres sat the weld toe and (σ_I) is the outer one.

$$\mathrm{DoB}=\frac{1}{2}\left(1-\frac{\sigma_I}{\sigma_O}\right) \tag{2}$$

3.3.1 Effects of Parameter β on Degree of Bending

The parameter β represents the ratio of brace outer diameter with chord outer diameter. Consequently, any increment of this parameter lead to the increasing of brace and chord outer diameters in size. Figure 12 presentes result of beta parameter variation effect on degree of bending. It could be seen that an increase in β generally decreases the value of the degree of bending.

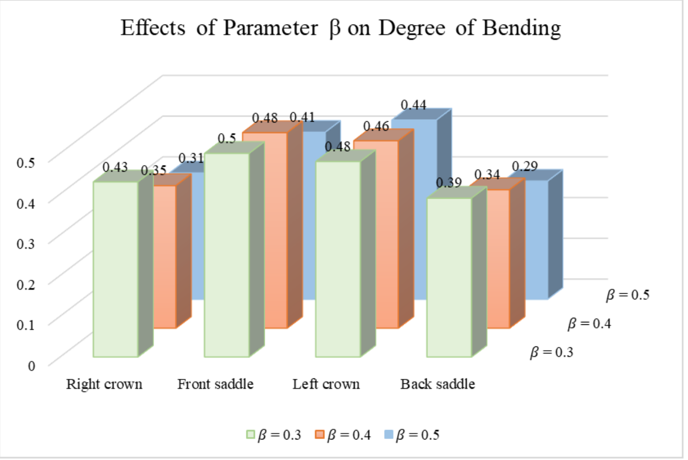


Figure 12. Effects of Parameter β on Degree of Bending

3.3.2 Effects of Parameter γ on Degree of Bending

The parameter γ is a ratio that assesses the slenderness of the chord, with its value comparing the outer diameter of the chord to the chord thickness. Consequently, an increase in the γ parameter will influence both the size of the outer diameter and the thickness of the chord.

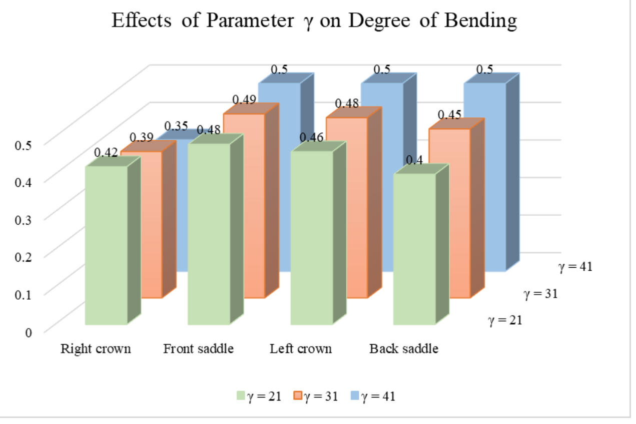


Fig. 13. Effects of Parameter γ on Degree of Bending

Based on the results presented in Figure 13, it can be seen that the increase in γ generally has different effects in each position and loading condition. For regions three areas, namely Front Saddle, Left Crown, and Back Saddle, experience an increase along with the increase in the gamma parameter. However, in the Right Crown area, there is a decrease as the gamma parameter increases.

3.3.3 Effects of Parameter τ on Degree of Bending

The parameter τ represents the ratio of brace thickness to chord thickness. Hence, any increase in the value of parameter τ will impact both the thickness of the brace and the thickness of the chord.

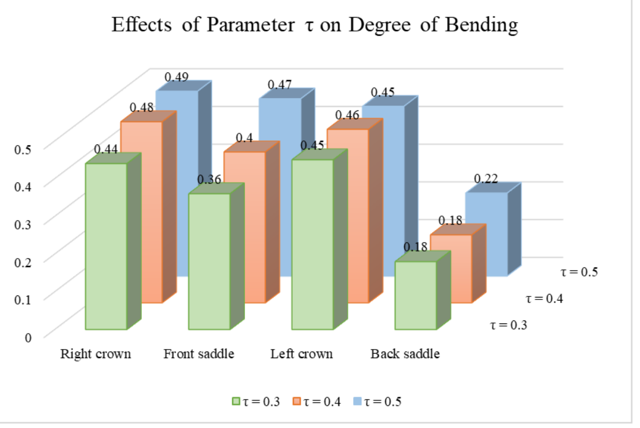


Fig. 14. Effects of Parameter τ on Degree of Bending

The effect of gamma parameter to DoB is presented in Figure 14. From the figure, it can be concluded that an increase in τ generally increases the value of degree of bending. However, most of these increases are not significant in the four positions and three loading conditions. For example, in the Left Crown, the values of each degree of bending fluctuate with very small differences.

4 Conclusion

This research is concluded to several characteristics of DTKY-joint due to geometrical variations. The maximum stress observed in the multi-planar DTKY-joint under axial loads is 162.12 MPa, occurring at the brace toe area. This is attributed to the DTKY-joint experiencing the highest axial load among the other braces. The impact of geometric parameters τ, γ, and β on the degree of bending in DTKY connections shows varied effects. Specifically, the β parameter tends to decrease the degree of bending, while the γ parameter generally increases it. The τ parameter also tends to increase the degree of bending, although the increase is not significant.

Acknowledgements

The authors gratefully acknowledge financial support and facilities from Department of Ocean Engineering, Institut Teknologi Sepuluh Nopember. Deepest gratitude to the experts that have contributed to this study.

References

1. H. Ahmadi, S. Asoodeh, Degree of bending (DoB) in Tubular KT-Joints of Jacket Structures Subjected to Axial Loads. Int. J. Marit. Technol. **4**, 65 (2015)
2. H. Ahmadi, A. Alizadeh Atalo, Geometrical Effects on the Degree of bending (DoB) of Multi-planar Tubular KK-Joints in Jacket Substructure of Offshore Wind Turbines. Appl. Ocean Res. **111**, 102678 (2021). https://doi.org/10.1016/j.apor.2021.102678
3. H. Ahmadi, S. Chamani, A. Kouhi, Effects of Geometrical Parameters on the Degree of bending (DoB) in Multi-planar Tubular XT-Joints of Offshore Structures Subjected to Axial Loading. Appl. Ocean Res**. 104**, 102381 (2020). https://doi.org/10.1016/j.apor.2020.102381
4. H. Ahmadi, V. Mayeli, and E. Zavvar, A Probability Distribution Model for the Degree of bending In Tubular KT-Joints of Offshore Jacket-Type Platforms Subjected To IPB Momen Loadings. Int. J. Coast. Offshore Environ. Eng. **4**, 11 (2019). https://doi.org/10.29252/IJCOE.3.2.11
5. H. Ahmadi, M. A. Lotfollahi-Yaghin, and S. Asoodeh, Degree of bending (DoB) in Tubular K-Joints of Offshore Structures Subjected to In-Plane Bending (IPB) Loads: Study of Geometrical Effects and Parametric Formulation. Ocean Eng. **102**, 105 (2015). https://doi.org/10.1016/j.oceaneng.2015.04.050
6. API, *API Recommended Practice 2A-WSD: Planning, Designing, and Constructing Fixed Offshore Platforms—Working Stress Design* (America Petroleum Institute, Washington D.C., 2014)
7. M. Efthymiou, *Development of SCF formulae and generalized influence functions for use in fatigue analysis OTJ'88*, (Egham, Surrey, 1988)
8. El-Reedy, M. A. *Offshore Strukctures Design, Construction and Maintenance* (Gulf Professional Publishing, Oxford, 2012)
9. A. F. Hobbacher. *Recommendations for fatigue design of welded joints and components* (Springer Cham, Switzerland, 2016). https://doi.org/10.1007/978-3-319-23757-2
10. M. Efthyimou, *Recent Developments in Tubular Joint Technology*. (Surrey, United Kingdom, 1988)
11. I. Noviyanti, R. W. Prastianto, and M. Murdjito, "Stress Distribution Analysis of DKDT Tubular Joint in a Minimum Offshore Structure," *Rekayasa*, vol. 14, no. 2, pp. 191–199, 2021, doi: 10.21107/rekayasa.v14i2.10511.
12. R. W. Prastianto, F. A. Admiral, and Y. S. Hadiwidodo, "Stress Concentration Factor Estimation of a Multi-planar DKT Tubular Joint through Finite Element and Machine Learning Approaches," 2022, doi: 10.4108/eai.11-10-2022.2326418.
13. R. W. Prastianto, I. A. Adha, Y. S. Hadiwidodo, D. M. Rosyid, and Y. Mulyadi, "Hot Spot Stress Analysis of a Two-Planar DTY Tubular Joint Subjected to Axial

Loading," *IOP Conf. Ser. Earth Environ. Sci.*, vol. 1166, no. 1, p. 012036, 2023, doi: 10.1088/1755-1315/1166/1/012036.

14. E. Zavvar, K. Hectors, and W. De Waele, "Stress concentration factors of multi-planar tubular KT-joints subjected to in-plane bending moments," *Mar. Struct.*, vol. 78, no. April, p. 103000, 2021, doi: 10.1016/j.marstruc.2021.103000.
15. AWS D1.1, *Structural Welding Code Steel*. 2010.
16. E. Chang and W. D. Dover, "Stress concentration factor parametric equations for tubular X and DT joints," *Int. J. Fatigue*, vol. 18, no. 6, pp. 363–387, 1996, doi: 10.1016/0142-1123(96)00017-5.
17. A. F. Hobbacher, *Recommendations for Fatigue Design of Welded Joints and Components*, 2nd ed. Switzerland: Springer International Publishing, 2016. doi: 10.1007/978-3-319-23757-2_7.
18. H. Nassiraei and P. Rezadoost, "Probabilistic analysis of the SCFs in tubular T/Y-joints reinforced with FRP under axial, in-plane bending, and out-of-plane bending loads," *Structures*, vol. 35, no. June 2021, pp. 1078–1097, 2022, doi: 10.1016/j.istruc.2021.06.029.
19. I. Noviyanti and R. W. Prastianto, "Numerical Study on the Stress Distribution Analysis of Two-Planar DKDT Tubular Joint Under Variation of Axial Loading Conditions BT - Recent Advances in Mechanical Engineering," 2023, pp. 19–26.