International Journal of Offshore and Coastal Engineering
Vol.10 | No. 1 | pp. 15 – 21 | May 2026
e-ISSN: 2580-0914

Submitted: December 22, 2025 | Revised: February 23, 2026 | Accepted: March 16, 2026

Responses Analysis of Tubular Joints in Offshore Substation Jacket due to Settlement During the Roll Up Process

Daniel M. Rosyid [a)], Wildan Malika Candra Alfatih [a)*], and Rudi Walujo Prastianto [a)]

[a)]Department of Ocean Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, Indonesia
*e-mail: wildanalfatih@oe.its.ac.id

ABSTRACT
The Offshore Substation plays a crucial role in ensuring the success of Offshore Wind Farm systems in the clean energy transition. During the fabrication stage, the jacket structure used as the Offshore Substation undergoes a fabrication process known as roll-up. During this roll-up process, ground settlement at the fabrication yard may occur, causing the jacket structure to tilt and increasing the risk of structural failure, which can reduce the effectiveness of the jacket structure's integrity. In this study, a roll-up analysis of the jacket structure was conducted using SACS software with variations in roll-up angles of 0°, 30°, 45°, 60°, 75°, and 96.4°. The analysis results indicate that during the roll-up process under settlement conditions, several critical areas were identified, particularly at the K-joint and X-joint, with maximum unity check (UC) values of 0.994 and 0.982, respectively. Further investigation was carried out through a local analysis using finite element software. The local stresses obtained were 264.6 MPa at the X-joint and 323.9 MPa at the K-joint. Since the stress at the K-joint exceeded the allowable stress, a thickness increase of 10 mm was applied to the brace experiencing the maximum stress. As a result, the stress was reduced by 172.29 MPa and was brought below the allowable stress limit.

Keywords: *Jacket, Roll up, Settlement, Tubular Joint, Finite Element Methods.*

1. INTRODUCTION
The development of Offshore Wind Farms has become one of the main solutions for meeting global renewable energy demands. Compared to onshore wind farms, Offshore Wind Farms offer advantages due to more consistent and stronger wind resources, which enable a significant increase in energy production [1]. The construction of offshore structures involves several fabrication stages, one of the most critical aspects requiring accurate consideration is the heavy lifting roll-up jacket process. Jacket roll-up is a stage in which the jacket structure, initially positioned horizontally, is lifted and erected into a vertical position for each row of the jacket structure, as illustrated in Figure 1.

In this study, the author conducted a response analysis of tubular jacket joints under ground settlement conditions with settlement variations of 5 cm, 10 cm, and 15 cm. The analysis was performed to evaluate the stress distribution occurring in the jacket structural joints during settlement, in order to ensure that the induced stresses remain below the allowable stress limits. Given the critical importance of offshore platform projects, accurate and reliable analysis is therefore required.

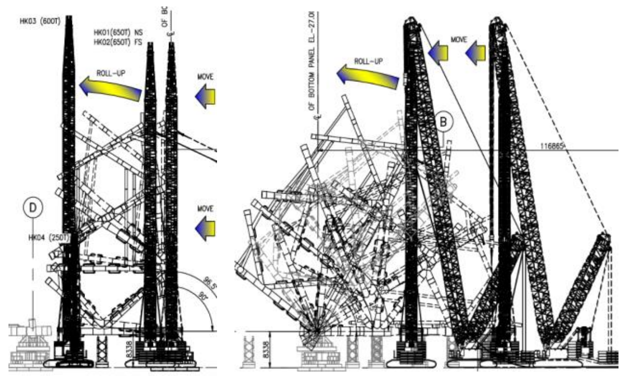

Figure 1.Roll Up Jacket Structure

To minimize the risk of jacket structural failure due to settlement during roll-up, an assessment of the structural capacity and response at each roll-up angle is required. This study aims to analyze the stress distribution in tubular joints of an offshore substation jacket under settlement conditions to ensure that stresses remain below allowable limits.

2. METHODOLOGY
2.1 Literature Study
A literature review was conducted by collecting relevant journals, textbooks, and previous final-year project reports

related to offshore jacket structures, roll-up processes, settlement effects, and tubular joint behavior. This review supports the theoretical framework applied in this study.

2.2 Data Collection

The structural data used in this study correspond to an Offshore Substation Platform (OSS) jacket structure. The OSS is responsible for receiving power from wind turbine radial cables, transforming it into high voltage, and exporting it to the onshore substation via export cables. The jacket is a four-legged structure equipped with pile sleeves at each outer leg, as shown in Figure 2.

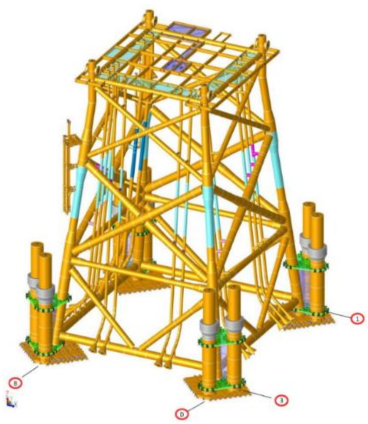

Figure 2. Offshore Jacket Substation Model

Table 1. Weight and COG Jacket

Description	Berat Struktur (mT)	Center of Gravity X (mm)	Center of Gravity Y (mm)
Row-B Structure	1291.27	21.411	16.900
Anodes	34.65		
Scaffolding	100		
Total Gross Weight	1486.66		

Table 2. Data *Rigging Equipment Jacket Substation*

The jacket weight and center of gravity (COG) data are presented in Table 1, while the rigging equipment data used during the roll-up process are summarized in Table 2.

Sling	*Shackle*	*Crane*
26,4 m *x* Ø115 mm *Wire Rope Sling* 12 m x Ø78 mm *Cable Laid Grommet*	*Crosby Bolt Anchor Type Shackle*	*QUY 650 Crawler Crane*

2.3 Modelling of jacket Structure

Structural modeling was performed using general arrangement drawings as input for SACS software. The complete jacket model was initially developed, followed by partial cutting at Row-D, as shown in Figures 3 and 4, in accordance with detailed drawings for roll-up analysis.

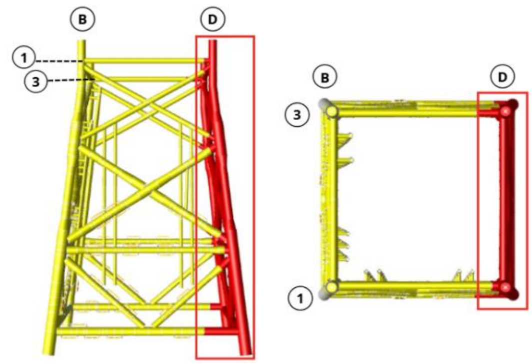

Figure 3. Jacket Row D Cutting Area Model

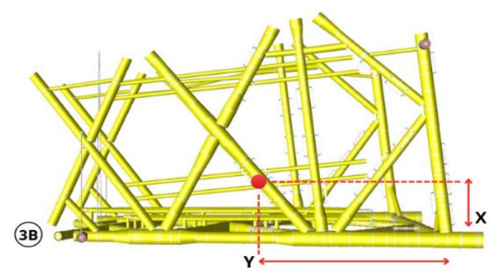

Figure 4. Side View COG Location

Table 2. Berat *Jacket Row* B dan CoG *Position*

Keterangan	Weight (Ton)	COG (m) X	COG (m) Y	COG (m) Z
Anode J-tubes & Support Jacket	1495,64	-38.14	-0.51	17.99

2.4 Model Validation

Model validation was conducted by comparing the total structural weight obtained from the SACS model with the actual structural weight based on the Material Take-Off (MTO). According to ISO 19901-5, the allowable error for validation is less than 5% between the model and the Weight Control Report (WCR) [6]. The comparison results demonstrate that the difference between the modeled structural weight and the actual weight from the MTO falls This indicates that the SACS model has been appropriately developed and can be considered valid for subsequent analyses, including roll-up, settlement, and local stress evaluations. The detailed validation results are summarized in Table 4.

Table 3. Validation of Model Weight with Structure Weight

Keterangan	Weight Control Report (Ton)	Weight SACS (Ton)	Difference (%)
Anode J-Tubes & Support Jacket	2488,09	2521,94	1,36 %

3. ANALYSIS AND DISCUSSION

3.1 Roll-Up Configuration and Static Analysis

At a roll-up angle of 0°, three hook points are used, each supported by three crane units, as shown in Figure 5. When the roll-up angle reaches 60°, hook point 3 (HK03) is released to allow the relocation of the hook point at the subsequent angle in accordance with the lifting procedure, leaving only hook point 1 (HK01) and hook point 2 (HK02).

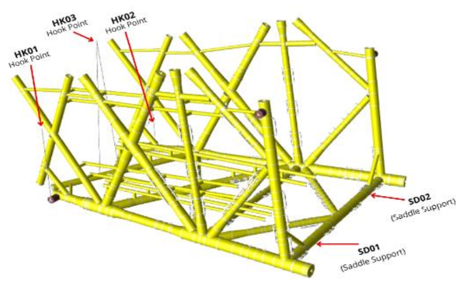

Figure 5 *Roll-up angle 0-60 degree*

In the roll-up angle configuration from 60° to 90°, two additional hook points are introduced, while one hook point at hook point 3 (HK03) from the previous configuration is removed. Consequently, the roll-up process at this stage utilizes four hook points, each supported by four cranes, as illustrated in Figure 6.

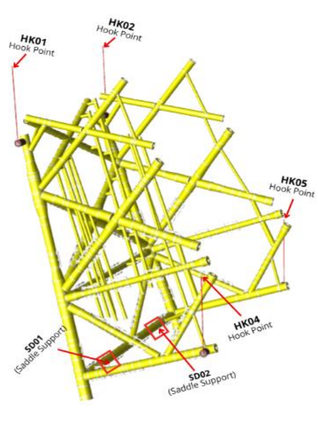

Figure 6. *Roll-up angle 60-90 degree*

When the roll-up angle reaches 96.4°, the lifting operation employs a tandem crane system, in which four hook points—HK01, HK02, HK04, and HK05—are used. Each hook point is directly connected to the upper and lower trunnions, as shown in Figure 7.

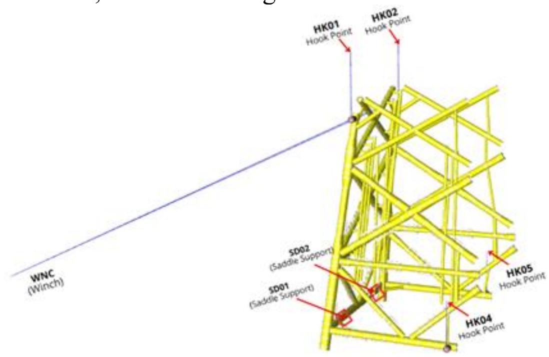

Figure 7. *Roll-up angle 96,4 degree*

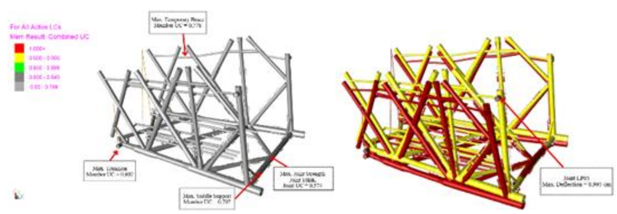

Figure 8. *Roll-up angle 0 degree*

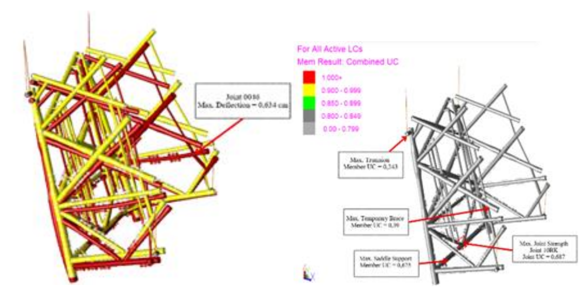

Figure 9. *Roll-up angle 45 degree*

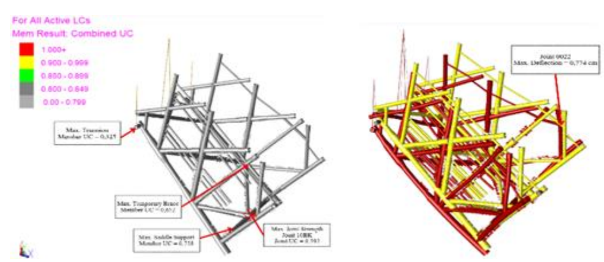

Figure 10. *Roll-up angle 75 degree*

The roll-up angle of 96.4° represents the final and most critical stage of the jacket erection process, where the structure has passed the vertical position and is fully supported by the lifting system. At this stage, the jacket experiences the highest combination of self-weight, lifting forces, and internal force redistribution due to geometric nonlinearity as in figure 11.

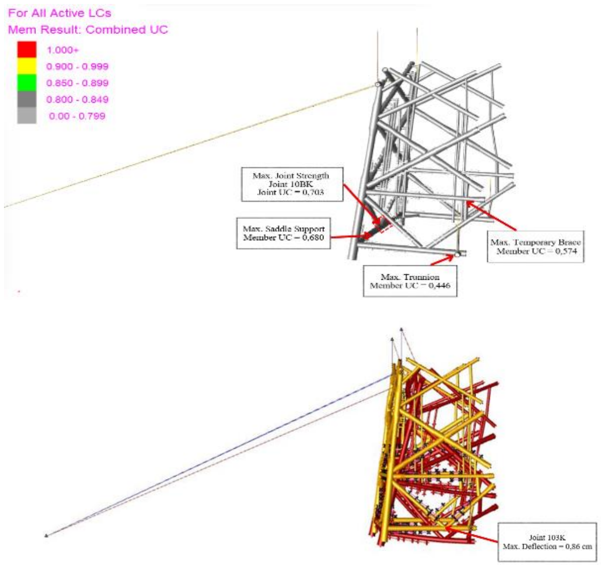

Figure 11. *Roll-up angle 96,4 degree*

The overall results of the static roll-up analysis based on the lifting angles are presented in Figures 8, 9, 10, and 11. In addition, the unity check (UC) result summary graph is shown in Figure 12, which illustrates the increase and decrease in member and joint strength utilization of the jacket structure at each lifting angle, ranging from 0° to 96.4°. The maximum member unity check (UC) occurs at a roll-up angle of 0° in the temporary brace area, with a UC value of 0.778, and at a roll-up angle of 30° in the saddle support area, with a UC value of 0.784. Meanwhile, the maximum joint strength unity check (UC) is observed at a roll-up angle of 96.4°, with a UC value of 0.703. The variations in unity check (UC) values are influenced by changes in the lifting configuration and load distribution throughout the jacket roll-up process.

3.2 Jacket Response Due to Settlement

In the analysis conducted to obtain the reactions acting on the jacket structure due to ground settlement, a modeling approach was applied in which one of the saddle supports was displaced downward from its original position. The ground settlement was varied at 5 cm, 10 cm, and 15 cm, as illustrated in Figure 12.

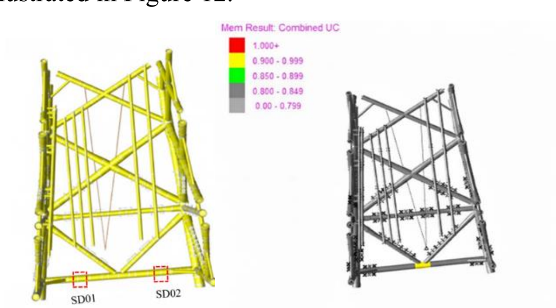

Figure 12. Illustration of jacket 15 cm settlement

Based on the roll-up analysis under various settlement conditions, the results indicate that the member unity check (UC) values of the jacket structure reach critical levels, as they approach the allowable criteria limits. As shown in Figure 13, the tubular K-joint area is identified as a critical region, exhibiting a UC value of 0.994 and experiencing the maximum stress. Therefore, further evaluation is required through a local analysis using finite element analysis (FEA) software to investigate the stress distribution occurring in the joint area in greater detail.

3.3 Local Analysis of Tubular K joint

Tubular joints with unity check (UC) values approaching the maximum allowable limit are modeled using finite element software to evaluate the stress distribution occurring in the joints, particularly in the welded regions. The joint selected for detailed modeling in this study is the tubular K-joint. In a tubular K-joint configuration, the joint consists of one chord element acting as the main structural member and two brace elements connected to the chord at specific angles, forming the characteristic K-shaped geometry. Between the intersections of these connections, a joint can is present, which serves as a critical component in the load transfer mechanism of the joint.

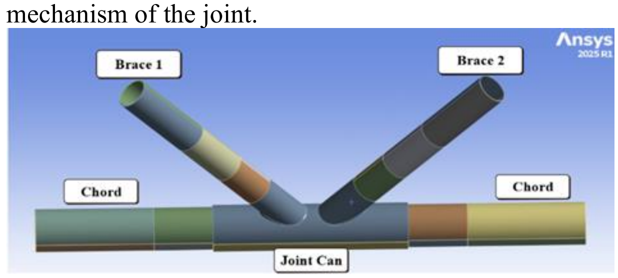

Figure 13. Local Geometry of Tubular K joint

A study conducted by Zavvar et al. (2021) indicates that the inclusion of weld modeling in local analysis can significantly influence the Stress Concentration Factor (SCF) and increase stiffness in hotspot stress regions. Therefore, in this study, the welds at the connections between the brace and chord are explicitly modeled, as shown in Figure 15. The weld modeling adopted in this final project refers to the American Welding Society standard (AWS D1.1, 2004).

Table 4. Weld Size on Tubular K-Joint Connections

Member Type	Wall Thickness (mm)	Tinggi Las (mm)	Panjang Las (mm)
Brace 1 (K *joint*)	25	25,49	26,14
Brace 2 (K *joint*)	25	25,49	26,14

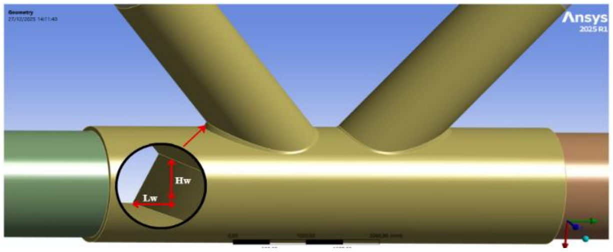

Figure 14. Weld Area in Tubular K joint

3.4 Boundary Conditions

Application of boundary conditions is carried out to simulate structural behavior that closely represents the actual conditions. In this model, both ends of the chord member are assigned fixed support boundary conditions, considering that the deformation or deflection along the chord element is relatively small. In addition to the supports, loads are directly applied to each brace member in the form of axial force (Fx), in-plane bending moment (My), and out-of-plane bending moment (Mz). These three loading components are obtained from the member end force output of the global structural analysis performed using SACS software. Consequently, the developed local model is capable of representing the actual loading conditions acting on the joint. The analysis is focused on the structural response in the joint region between the chord and the brace; therefore, fixed support conditions at the ends of the chord are considered adequate to isolate the influence of local behavior around the joint zone. The applied loading configuration is illustrated in Figure 15.

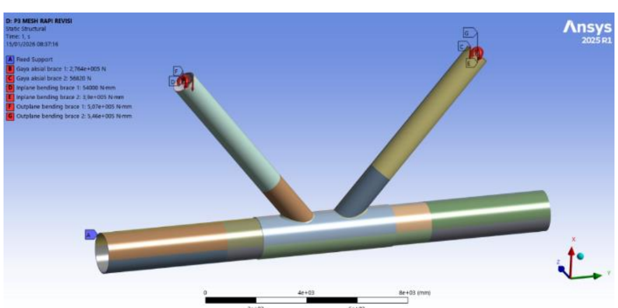

Figure 15. Boundary Conditions K joint

3.5 Meshing and Local Analysis

The mesh for the chord and brace members is modeled using hexahedral elements, while tetrahedral elements are applied in the weld region. The use of hexahedral elements for the chord and brace is based on their geometric characteristics, where the chord in particular has the most complex geometry and the largest dimensions. Hexahedral elements are able to represent complex geometries more effectively due to their greater number of corner nodes. Meanwhile, tetrahedral elements are employed in the brace–chord intersection and weld regions because these areas are divided into multiple zones. Due to the geometric singularity at the joint intersection, the mesh elements tend to become more irregular; therefore, tetrahedral elements are more suitable and recommended for accurately capturing the stress distribution around the joint. The mesh configuration is shown in Figure 16.

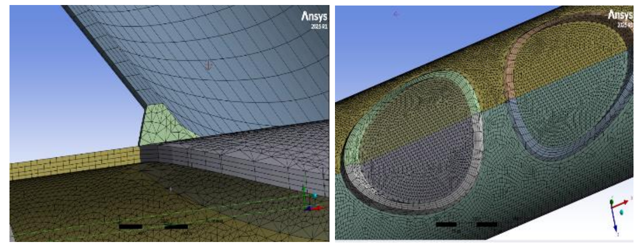

Figure 16. Mesh Model in Cutting Area Connections

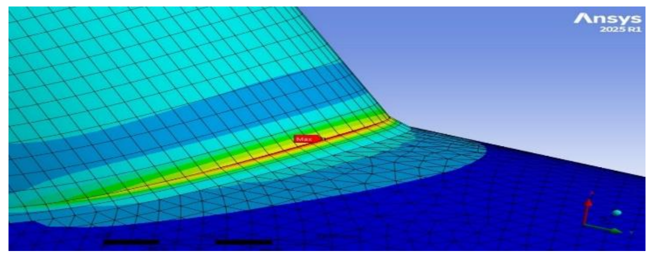

Figure 17. Stress Maximum in Tubular K joint

The stress obtained for the K-joint connection, as shown in Figure 17, is 323.9 MPa. According to API RP 2A-WSD 22nd Edition (2014), the maximum allowable stress for a K-joint is 0.6 times the yield stress, which is 213 MPa. Although the resulting stress does not exceed the yield stress of 355 MPa indicating that the deformation remains within the elastic range—it does exceed the allowable stress limit. Considering the importance of allowable stress in ensuring design safety, mitigation measures are therefore required. In this study, mitigation is carried out by increasing the thickness of the brace (joint stub) in the region where the brace is connected to the chord.

3.6 Thickness Addition for Critical Joints

As a mitigation measure against the exceedance of allowable stress in the tubular K-joint connection, a design modification is implemented by increasing the thickness of the brace. Based on the initial finite element analysis results, the highest stress concentration is identified at the intersection between the brace and the chord, particularly around the weld toe region. Increasing the brace thickness, the stress distribution in the joint is expected to become more uniform, as the enhanced stiffness of the brace reduces excessive deformation and stress concentration near the weld region. Moreover, the added thickness contributes to lowering the stress intensity at the weld toe, which is particularly important for maintaining compliance with allowable stress limits specified in API RP 2A-WSD. This modification does not alter the global structural configuration but effectively improves the local joint performance, ensuring that the induced stresses remain within acceptable limits and enhancing the overall safety and reliability of the tubular joint design.

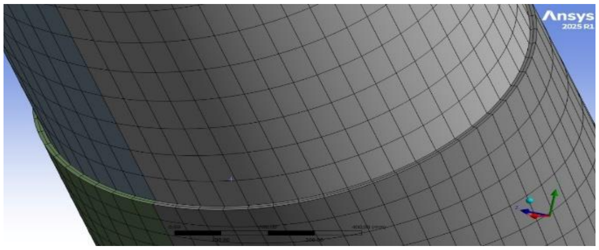

Figure 18. Thickness Addition 10 mm in Stub

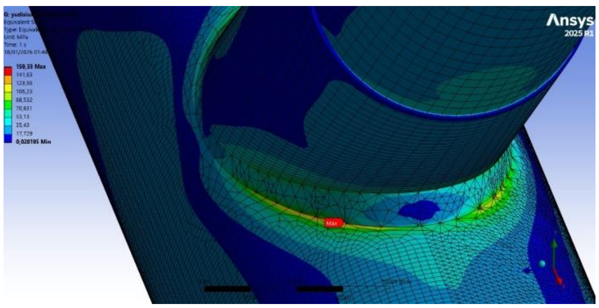

Figure 19. Stress maximum After Addition Thickness’

The von Mises stress in the tubular K-joint connection after the addition of a 10 mm thickness is observed to decrease, with the resulting stress value falling below the allowable stress limit, reaching 159.33 MPa. The reduction in stress indicates an improvement in the load-carrying capacity of the joint, thereby enhancing its structural performance. As a result, the structural capacity of the connection increases, and the risk of plastic failure can be effectively minimized.

3.7 Hot-Spot Stress

The hot-spot stress calculation in this study is conducted in accordance with the guidelines issued by the International Institute of Welding (IIW), specifically referring to document IIW-XV-E developed by A. Hobbacher (2008). These guidelines are widely adopted in fatigue and local stress analyses of welded joints, particularly for tubular connections in offshore structures, due to their ability to provide a consistent and reliable definition of structural hot-spot stress. By applying linear extrapolation between the stresses obtained at these two reference points, the structural hot-spot stress at the weld toe can be estimated more accurately. This approach allows the separation of the structural stress component from the local notch stress. The use of the IIW-recommended method ensures that the calculated hot-spot stresses are consistent with international standards and can be reliably used for subsequent fatigue assessment and structural integrity evaluation of the tubular K-joint connection.

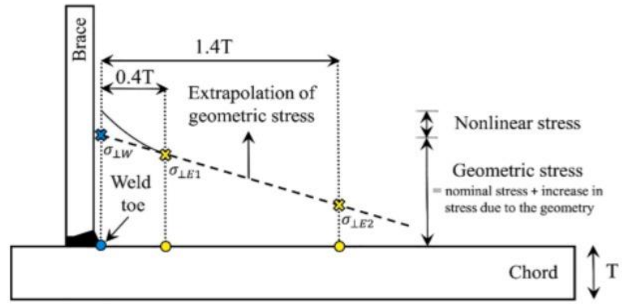

Figure 20. Extrapolation in Weld Toe area

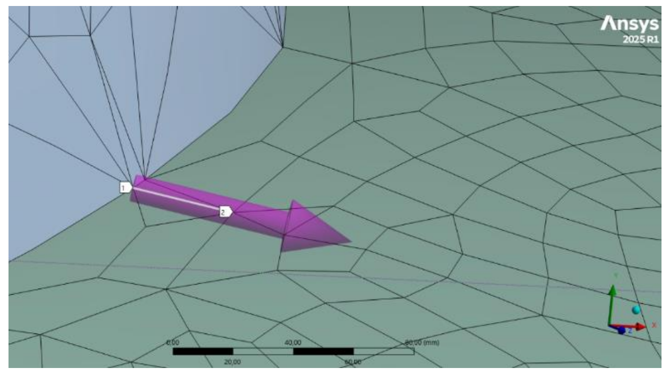

Figure 21. Path Extrapolation in Weld Toe

Table 5. Node Stress Extrapolation After Addition Thickness

Titik tinjau		Jarak (mm)	Stress (MPa)
Notch	0		
A	0,4t	12	159,33
B	1t	30	146,37

Based on the linear equation established in Figure 22, the hot-spot stress of the tubular K-joint connection after the implementation of a 10 mm thickness increase is found to decrease to 172.29 MPa. This value is obtained through the linear extrapolation process applied to several reference stress points located at specified distances from the weld toe**.** The observed reduction in hot-spot stress following the thickness increase indicates that the mitigation measure effectively improves the local stress behavior of the joint. The enhanced stiffness resulting from the added thickness leads to a more gradual stress gradient and a lower extrapolated stress value at the weld toe. Consequently, the joint exhibits improved resistance to fatigue crack initiation, and its overall structural performance is enhanced. This result confirms that the thickness modification strategy is effective in reducing critical stresses and improving the safety margin of the tubular K-joint connection.

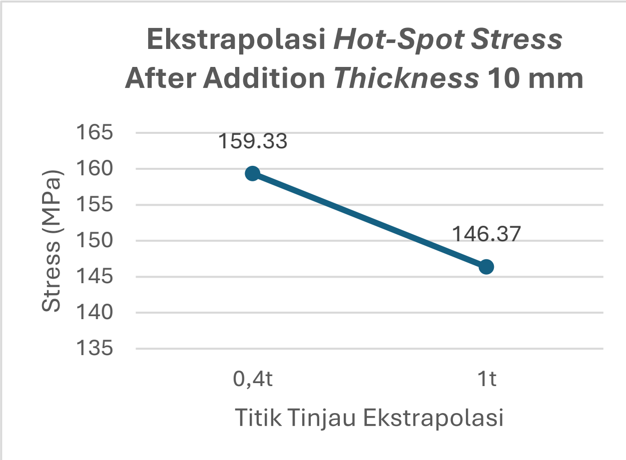

Figure 22. Extrapolation Grapics in Hot Spot Stress

After determining the hot-spot stress in the tubular K-joint with the addition of a 10 mm thickness, a stress concentration factor (SCF) assessment is subsequently carried out. In the tubular K-joint, the axial load used for the nominal stress corresponds to the load acting on Brace 1, amounting to 6,462 kN, since the hot-spot stress is located at the connection between the chord and Brace 1. Once the hot-spot stress and nominal stress are determined, the stress concentration factor can be obtained, as summarized in Table 6.

Table 6. Stress Concentration Factor

Titik tinjau	Jarak (mm)	Stress (MPa)
172,29	12,58	13,37

6. CONCLUSIONS

Based on the analyses conducted, the following conclusions can be drawn from this study:

1. The roll-up analysis under settlement conditions identifies several critical areas. The most critical condition occurs at a settlement of 15 cm during the roll-up at an angle of 96.4 degrees, with a unity check (UC) value of 0.994 and a joint strength UC of 0.984. These critical responses are located at the tubular K-joint connection.
2. In the local analysis of the tubular K-joint, a maximum stress of 323.9 MPa is observed at the brace weld toe region. This stress exceeds the allowable stress limit of 213 MPa, indicating a critical condition at the joint. To mitigate this issue, a design modification is implemented by increasing the thickness of the tubular brace by 10 mm. After the thickness increase, the von Mises stress is reduced to 159.33 MPa, while the hot-spot stress at the joint area is 172.29 MPa.
3. The resulting stress concentration factor (SCF) is 13.7 relative to the nominal stress of the tubular K-joint. Therefore, the 10 mm thickness increase is considered effective and can be applied during the fabrication roll-up process under settlement conditions.

ACKNOWLEDGEMENTS

The author would like to express sincere and deepest gratitude to all parties, including parents and friends, who have provided support throughout this research, both materially and through knowledge, motivation, and prayers, enabling the successful completion of this study.

REFERENCES

1. Timoshenko, S. and Goodier, J.N.: *Theory of Elasticity*. 2nd Ed., Mc Graw-Hill, New York, 1951
2. Misra, S.C.: *Design Principles of Ships and Marine Structures*. CRC Press, Boca Raton, USA, 2015
3. ABS: *Guide for Spectral-Based Fatigue Analysis for Floating Production, Storage and Offloading (FPSO) Installation.* American Bureau of Shipping, ABS Plaza, Houston, TX, USA, 2010
4. API: *Recommended Practice for Planning, Designing, and Constructing Fixed Offshore Platforms-Working Stress Design*. API RP 2A-WSD, 22nd Ed., American Petroleum Institute, Washington, DC, USA, 2014
5. Kurian, V.J., Ng, C.Y. and Liew, M.S.: An experimental investigation of the responses of classic spar platform subjected to bi-directional waves. *Research Journal of Applied Sciences, Engineering and Technology* 9(1): 11-17, 2015 DOI:10.19026/rjaset.9.1370
6. Chen, X.: Offshore hydrodynamics and applications. *The IES Journal Part A: Civil & Structural Engineering* V.4, N.3, pp. 124–142, 2011 http://dx.doi.org/10.1080/19373260.2011.595903
7. Wichers, J. E.W.: Wave-current interaction effects on moored tankers in high seas. *Offshore Technology Conference*, Paper No. OTC5631, Houston, TX, USA, 1988
8. Fossli, B. and Hendriks, S.: PRD12,000 drill ship; increasing efficiency in deep water operations. *SPE/IADC Drilling Conference and Exhibition*, Orlando, Florida, USA, 2008

For internet documents:

9. Osei-Badu, A.: Building FPSO in Ghana not possible – MODEC. Ghana Chamber of Oil Bulk Distributors, June 22, 2016. Retrieved January 7, 2017 from http://cbodghana.com/building-fpso-in-ghana