American Journal of Computational Mathematics, 2024, 14, 291-304
https://www.scirp.org/journal/ajcm
ISSN Online: 2161-1211
ISSN Print: 2161-1203

Finite Element Analysis of Tubular KK Joint under Compressive Loading

Mohammad Tanzil Hasan[1*], Md. Jobayer Mia[2], Md. Syful Isalm Bipul[2]

[1]Department of Mathematics, Bangabandhu Sheikh Mujibur Rahman Maritime University, Dhaka, Bangladesh
[2]Department of Naval Architecture and Offshore Engineering, Bangabandhu Sheikh Mujibur Rahman Maritime University, Dhaka, Bangladesh
Email: *tanzil.math@bsmrmu.edu.bd, jobayer.naoe@bsmrmu.edu.bd, syfulbipul@gmail.com

How to cite this paper: Hasan, M.T., Mia, Md.J. and Bipul, Md.S.I. (2024) Finite Element Analysis of Tubular KK Joint under Compressive Loading. *American Journal of Computational Mathematics*, **14**, 291-304.
https://doi.org/10.4236/ajcm.2024.143013

Received: June 3, 2024
Accepted: July 19, 2024
Published: July 22, 2024

Abstract

KK tubular joints are used to build jacket-type offshore structures. These joints are mostly made up of structural steel. These joints can withstand yield, buckling, and lateral loads depending on the structure's design and environment. In this study, the Finite Element Model of the KK-type tubular joint has been created, and analysis has been performed under static loading using the Static Structural analysis system of ANSYS 19.2 commercial software and structural mechanics module of COMSOL Multiphysics. The KK tubular model is analyzed under compressive load conditions, and the resulting stress, strain, and deformation values are tabulated in both graphical and tabular form. This study includes a comparison of the outcomes from both commercial software. The results highlight that maximum stress, strain, and deformation values decrease as joint thickness increases. This study holds significant relevance in advancing the understanding of tubular KK joints and their response to compressive loading. The insights derived from the analysis have the potential to contribute to the development of more robust and reliable tubular KK joints in various engineering and structural applications.

Keywords

KK Joint, Brace, Offshore Structures, Chord, Numerical Analysis, Static Analysis

1. Introduction

Offshore structures are installed on the seabed to extract oil and gas from the seafloor. In recent years, the offshore sector has grown rapidly. A tubular KK joint is a type of joint used in the construction of offshore structures such as oil rigs, platforms, and pipelines. It is designed to connect two tubular members together in a way that provides both strength and flexibility to the overall structure. When the tubular KK joint is subjected to compressive loading, it experiences forces that tend to crush or buckle the joint. The ability of the joint to resist compressive loading depends on several factors, including the strength of the material, the geometry of the joint, and the length and diameter of the tubular members being joined. To ensure that the tubular KK joint can withstand compressive loading, engineers typically conduct a detailed analysis using mathematical models and simulations. One common approach to designing a tubular KK joint under compressive loading is to use a combination of axial and hoop stresses to resist the forces. The axial stresses are caused by the direct compression of the joint, while the hoop stresses are caused by the bending of the joint as it tries to maintain its circular shape. Other factors that can affect the performance of a tubular KK joint under compressive loading include the presence of corrosion, fatigue, and other types of damage.

Many theoretical and experimental studies on the structural behavior of unstiffened tube joints have been conducted by the researchers since 1950. Most of this study was devoted to problems like calculating static strength. Zhao *et al.* [1] investigated about experimental study on static behavior of multi-planar overlapped CHS KK-joints. The study included experimental and numerical analyses of various joint configurations. The results showed that the joint capacity is influenced by the chord-to-brace diameter ratio and the brace-to-chord thickness ratio. The authors suggested that the design of CHS KK joints should consider these factors to ensure structural integrity and safety. Chen and Wang [2] studied about flexural behavior and resistance of uni-planar KK and X tubular joints. The joints were subjected to cyclic loading and their load-displacement responses were recorded. The experimental results showed that the KK joint had higher stiffness and strength compared to the X joint. The findings of this study can be useful in designing tubular structures and improving their performance under seismic loading. Analysis of tubular joints of offshore structure was studied by Sawant and Muthumani [3]. The paper discussed the behavior of tubular joints in offshore structures and also conducted numerical simulations using finite element analysis and validated their results with experimental data. They found that the joint's behavior is highly dependent on the loading direction and magnitude, as well as the geometry of the joint. Santacruz and Mikkelsen [4] worked on numerical stress analysis of tubular joints and showed that the presence of weld geometry leads to higher SCFs (stress concentration factors) than ignoring it. The use of solid elements in FEA is found to produce more accurate results than shell elements due to better representation of joint geometry. The authors recommend implementing weld geometry in FEA during design and structural integrity assessment for safe operation of offshore structures. Oshogbunu *et al.* [5] represented an experimental and numerical prediction of hotspot stress for DKK CHS joints in offshore jackets under arbitrary loading. This paper proposed a set of six basic brace load cases and ten unique stress values that accurately represent force distributions in double K (DKK) joints, providing similar accuracy to full 3D FEA analysis and greater accuracy than existing parametric methods. Zhao *et al.* [6] published an experimental and numerical investigation of behavior of overlapped CHS K/KK-joints with different welding situations. The results showed that while the welding situation of the hidden seam affects stress distribution and failure mechanism, it does not significantly affect the ultimate capacity of the joints, except for those with special geometrical parameters. Forti *et al.* [7] studied numerical methodology for analyses of tubular KK multiplanar steel joints. The methodology included considerations such as material properties, mesh generation, incremental load steps, and failure criteria. The proposed criterion identifies the two most common failure modes, and an extension of Lu's deformation criterion is proposed for calculating joint resistance.

Jingxia *et al.* [8] worked on fatigue strength assessment on a multiplanar tubular KK-joints by Scaled model test. A scaled KK-joint was designed using sensitive analysis and similarity analysis, and then subjected to static and fatigue tests under axial loading. The maximum hot spot stress (HSS) and its location were recorded, and the fatigue life was predicted for the original KK-joint using finite element analysis and compared with rule-based results. Aghaei *et al.* [9] studied about new parametric equations for estimating stress concentration factors in tubular KK-joints under axial loading. The S-N curve method is commonly used for estimating the fatigue life of these structures, but there are no equations available for calculating stress concentration factors (SCFs) in tubular KK-joints. A data bank of SCFs for KK-joints under balanced axial loading was produced using finite element method, and new equations were derived using nonlinear regression analysis. Parametric study of unstiffened multi-planar tubular KK-Joints was studied by Kadry *et al.* [10]. A verified ANSYS model was used to generate 172 records for parametric study with different geometry, member sections, and material properties. The developed model consists of two equations based on failure mode and showed an average accuracy of 92.5% when compared to experimental test results. Ahmadi *et al.* [11] worked on the ultimate capacity of stiffened and unstiffened multi-planar tubular kk-joints in offshore structures and a parametric study is performed to investigate the effects of varying parameters and three different methods of reinforcing the chord are studied and compared with API design equations. Mia *et al.* [12] showed finite element model of the XT-type tubular joint and its analysis under static loading using ANSYS 19.2 software. The analysis of the XT tubular joint under different load cases revealed that increasing the joint's thickness reduces the maximum stress, strain, and deformation values. The yield point of the joint was determined to be 30 KN of loading for both tensile and compressive loading.

The present study illustrates the results which have been obtained for static loading conditions using two different commercial software namely the Static Structural analysis system of ANSYS 19.2 and structural mechanics module of COMSOL Multiphysics. The research presents the stress, strain and deformation values obtained from the analysis in tabular and graphical forms and also compares the results obtained from both software systems. The study reveals that increasing the joint thickness leads to a decrease in the maximum stress, strain and deformation values.

The present research is composed of four sections. Section 1 serves as a general introduction, including a brief overview of the motivation for the study, a review of pertinent literature, and the research objectives. Section 2 outlines the methodology employed in the investigation, which includes a comprehensive discussion of the governing equations, problem statement, numerical modeling, mesh generation, and definition of boundary conditions. Section 3 provides a brief overview of the results obtained and subsequent discussions. Lastly, Section 4 offers a conclusion of the entire research endeavor.

2. Methodology

In this study, the Finite Element Method (FEM) is employed to analyze the KK tubular joint. FEM is chosen as it is difficult to apply closed-form solutions to determine stress and strain in tubular structures due to their complex geometry. The numerical analysis of the model is carried out using ANSYS software, following the steps outlined in **Figure 1**.

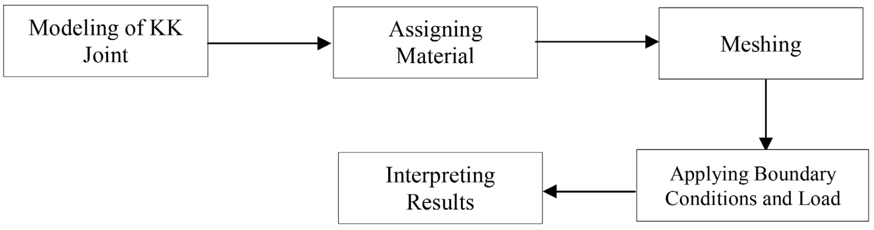

Figure 1. Steps of the model analysis in ANSYS.

For performing the simulation in COMSOL Multiphysics, the whole process is divided into three stages: Preprocessing, processing, and post-processing. The steps of the model analysis in COMSOL Multiphysics are shown in **Figure 2**.

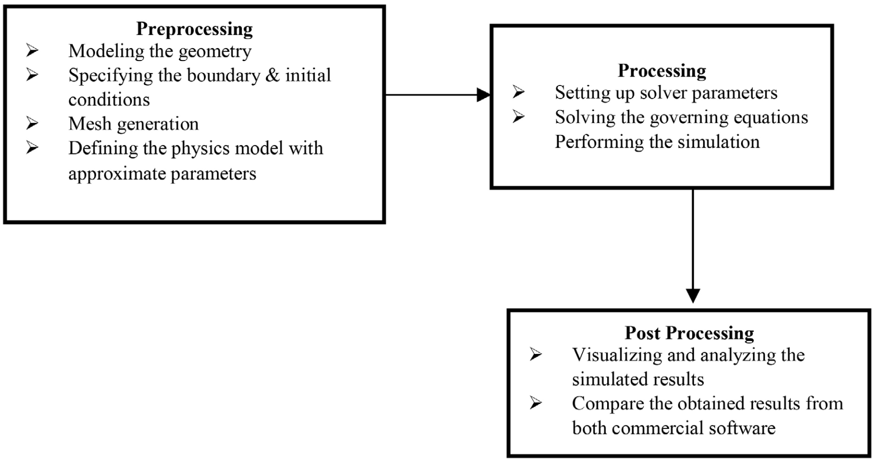

Figure 2. Steps of the model analysis in COMSOL Multiphysics.

2.1. Governing Equations

Stress (σ) is the measurement of applied forces per unit area at a point of a solid body. The stress field is expressed in vector form $\sigma = \{\sigma(x, y, z)\}$ as

$$\sigma = \left[\sigma_x, \sigma_y, \sigma_z, \tau_{yz}, \tau_{zx}, \tau_{xy}\right]^{\mathrm{T}} \tag{1}$$

where σ_x, σ_y, σ_z are normal stresses and τ_{yz}, τ_{zx}, τ_{xy} are shear stresses. Von Mises stress is used as a criterion for determining the onset of failure in materials [13]. The Von Mises stress σ_{VM} is given by [14]

$$\sigma_{VM} = \sqrt{I_1^2 - 3I_2} \tag{2}$$

where I_1 and I_2 are the first two invariants of the stress tensor. For the general state of stress, I_1 and I_2 are given by

$$I_1 = \sigma_x + \sigma_y + \sigma_z \tag{3}$$

$$I_2 = \sigma_x\sigma_y + \sigma_y\sigma_z + \sigma_z\sigma_x - \tau_{yz}^2 - \tau_{xz}^2 - \tau_{xy}^2 \tag{4}$$

"Strain (ε) is the ratio of the change in length caused by the applied force, to the original length" [15]. Strain in vector form is represented as

$$\varepsilon = \left[\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{zx}, \gamma_{xy}\right]^{\mathrm{T}} \tag{5}$$

where, ε_x, ε_y, ε_z are normal strains and γ_{yz}, γ_{zx}, γ_{xy} are shear strains.

If the axial stress is applied in the x-direction, then strain is represented as [16]

$$\varepsilon_x = \frac{\sigma_x}{E} \tag{6}$$

where E is the modulus of elasticity. For small deformation, the strain displacement relation is signified as [13]

$$\varepsilon = \left[\frac{\partial u}{\partial x}, \frac{\partial v}{\partial y}, \frac{\partial w}{\partial z}, \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}, \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}, \frac{\partial u}{\partial x} + \frac{\partial v}{\partial x}\right]^{\mathrm{T}}. \tag{7}$$

2.2. Problem Statement

In this study, the tubular KK joint is considered. **Figure 3** represents a schematic illustration of the problem. The figure shows that there are mainly two parts of the KK joint, chord and brace. The braces are attached to the chord at an angle of 30 degrees. Moreover, the diameter and length of the chord are always greater than the length and diameter of the brace, as indicated in the figure.

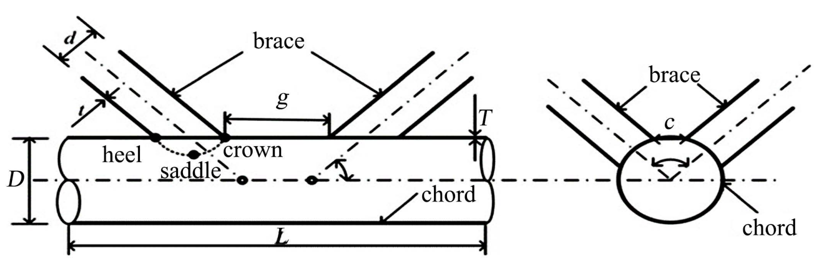

Figure 3. Illustration of the KK model geometry.

2.3. Numerical Modeling

For this study, the numerical model is created using the design modeler of the ANSYS STATIC STRUCTURAL package and COMSOL Multiphysics. The dimensions of the model are chosen based on previous research. Figure 4 and Figure 5 display the developed model. Table 1 lists the dimensions of the model's geometric parameters, and Table 2 provides the engineering properties, including the material used, density, and Young's modulus.

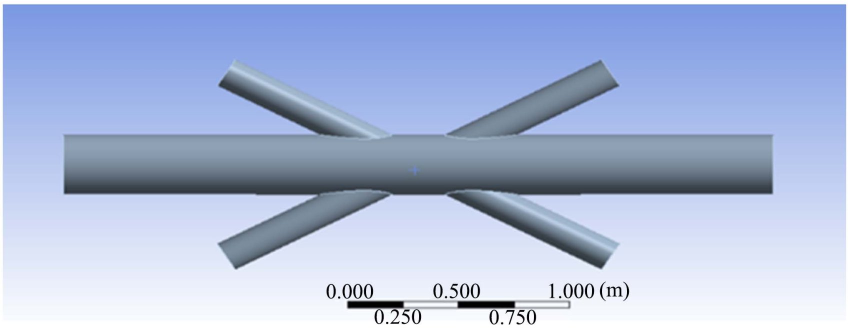

Figure 4. Developed KK joint model in ANSYS.

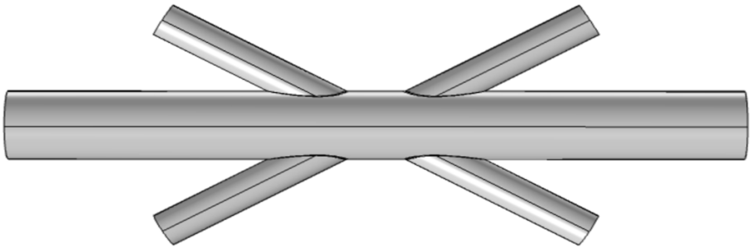

Figure 5. Developed KK joint model in COMSOL Multiphysics.

Table 1. Geometric parameters of the model.

Geometric parameters	Chord (mm)	Brace (mm)
Diameter	325	162
Length	3200	1000
Thickness	5	5

Table 2. Engineering parameters of the model.

Engineering properties	Specifications
Material used	Structural steel
Density	7850 kg/m^3
Young's Modulus	2×10^5 MPa
Poisson's Ratio	0.3
Bulk Modulus	1.67×10^{11} Pa
Shear Modulus	7.69×10^{10} Pa
Yield Stress	250 MPa

2.4. Meshing

Meshing is a crucial step in Finite Element Analysis (FEM) as it involves dividing the entire geometry into smaller elements in order to obtain accurate results. The mesh setup is one of the most important aspects of numerical analysis as it plays a significant role in determining the accuracy of the FEM analysis results. It is estimated that around 60% of the project's time is dedicated to acquiring the proper meshing geometry. By ensuring good and accurate meshing, the numerical model can simulate more realistic outcomes.

For the current study, meshing is done using both the meshing module of the ANSYS STATIC STRUCTURAL workbench and the default meshing tool of COMSOL Multiphysics. A customized element size type meshing is selected to avoid numerical errors. An effort is made to design a regular mesh with the correct element size, and the unstructured mesh type is chosen in both cases. To make the simulation process easier and achieve good results, the model is divided into 47,981 nodes and 23,750 elements using the meshing module of the STATIC STRUCTURAL workbench. On the other hand, using the COMSOL meshing tool, the model is divided into 142,828 boundary elements and 5402 edge elements. The generated mesh for this study is illustrated in **Figure 6** and **Figure 7**.

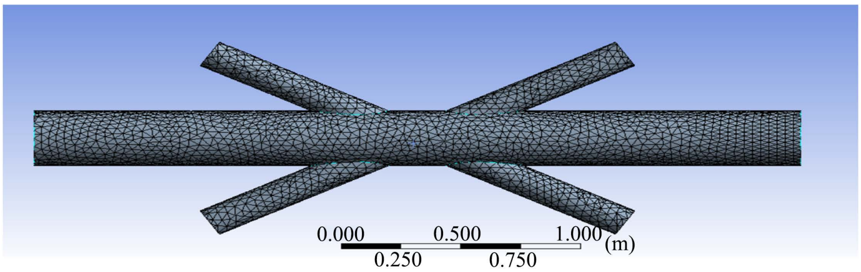

Figure 6. Generated mesh in ANSYS.

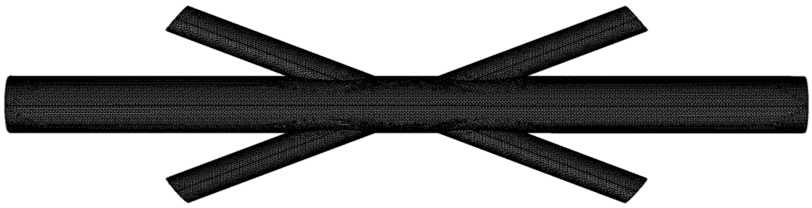

Figure 7. Generated mesh in COMOL Multiphysics.

2.5. Boundary Conditions

In this study, specific boundary conditions are applied to the chord and brace of the joint to analyze their behavior under compressive loading. The ends of the chord, which are welded into the jacket of the offshore structures, are fixed in place by setting fixed boundary conditions at both ends. This means that both ends of the chord are constrained in all degrees of freedom. In terms of compression, the force is distributed evenly across the free ends of the braces, with the downward direction of loading indicating compression. A total load of 100 N is applied to the braces, with 25 N of load applied to each brace. Additionally, in both the ANSYS and COMSOL simulation modules, the compressive loading is applied as remote forces to ensure that the forces are distributed uniformly across the braces. The loads are also applied gradually, allowing for the corresponding stress, deformation, and strain values to be obtained clearly. The boundary conditions and loading are illustrated in Figure 8.

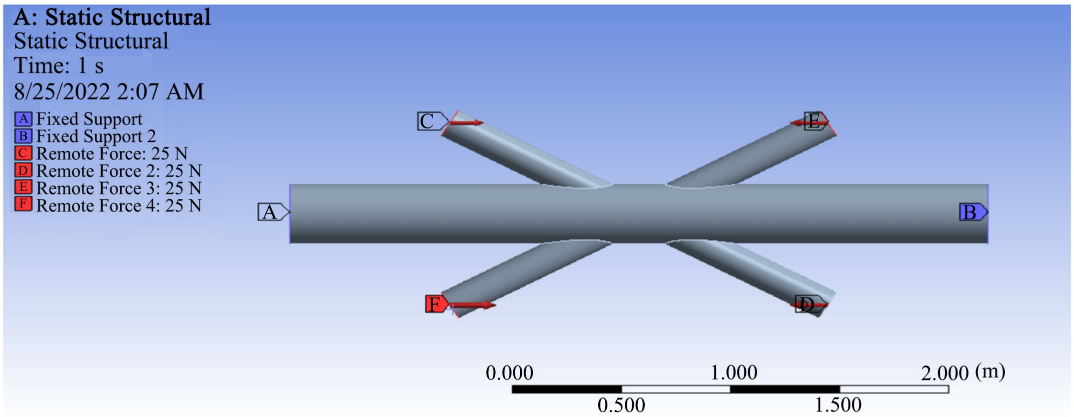

Figure 8. Boundary conditions of the problem.

3. Results and Discussions

Figure 9 and Figure 10 illustrate the deformation of the joint. It is seen that maximum deformation occurs in the braces where the compressive load is applied. When compressive loading is applied to the brace of a tubular KK joint, it will experience deformation as a result of the applied load. The deformation can be described in terms of the change in length, shape, or both. The amount of deformation that occurs will depend on the strength and stiffness of the brace, as well as the magnitude of the applied load. From the figures, it is seen that the value of maximum deformation found in the ANSYS simulation is relatively larger than the COMSOL Multiphysics simulation.

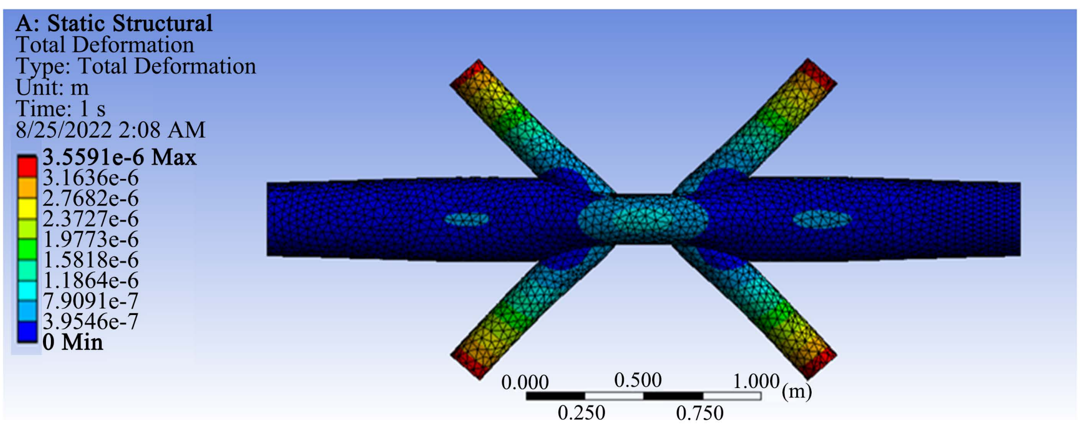

Figure 9. Total deformation of the joint in ANSYS.

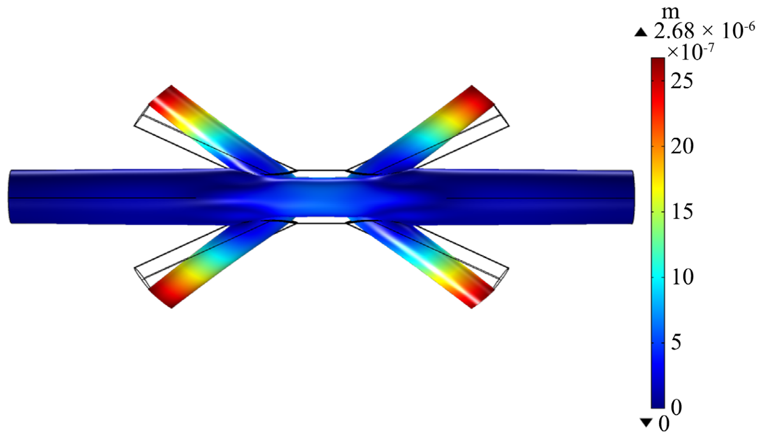

Figure 10. Total deformation of the joint in COMSOL Multiphysics.

Figure 11 and Figure 12 represent the distribution of Von-Mises stress in the KK joint. Von-Mises stress is a measure of the combined effect of normal stresses and shear stresses on a material. It is used to determine the maximum stress that a material can withstand before it fails. In the context of a tubular KK joint for compressive loading applied in the braces, Von Mises stress is used to determine the maximum stress that the joint can withstand before it fails.

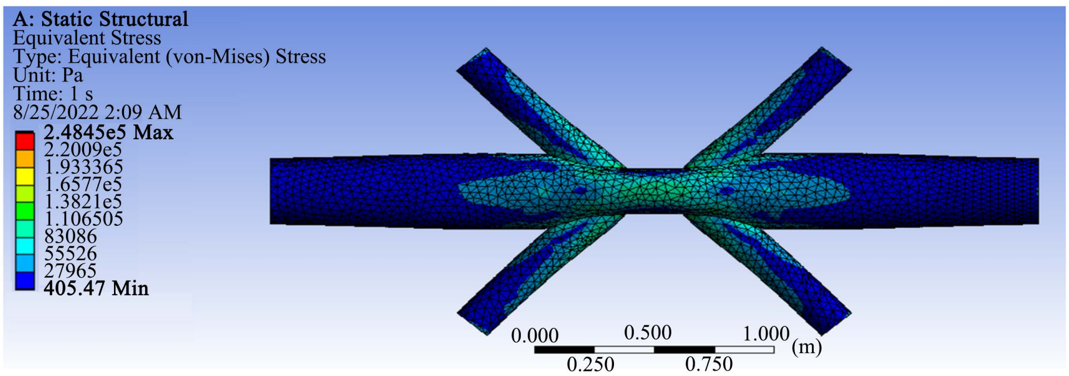

Figure 11. Von-Mises stress distribution of the joint in ANSYS.

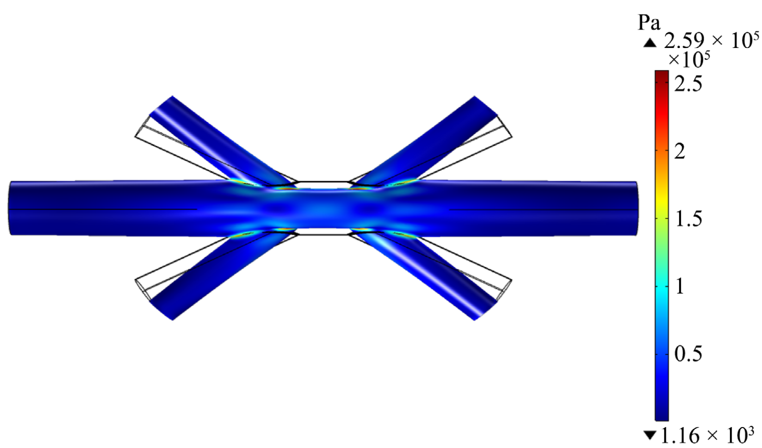

Figure 12. Von-Mises stress distribution of the joint in COMSOL Multiphysics.

In this study, when a compressive load is applied to the braces of the tubular KK joint, the stress state in the joint is complex. The loading conditions in the joint include both axial and torsional stresses, as well as bending stresses. The Von-Mises stress is used to evaluate the overall stress state in the joint and determine if the joint is safe and stable.

From the figures it is seen that the Von-Mises stress is higher near the brace connections, where the complex loading conditions are greatest. And the maximum value of Von Mises stress is pretty similar in both ANSYS and COMSOL Multiphysics simulations.

Figure 13 and Figure 14 illustrate that when compressive loading is applied in the braces of the KK joint, the strains are primarily concentrated in the brace-to-chord connection. The strain in the chord itself is relatively small and is primarily caused by the displacement of the brace-to-chord connection. The strains in the brace-to-chord connection are primarily caused by the applied compressive load and by the bending of the brace caused by the displacement of the chord.

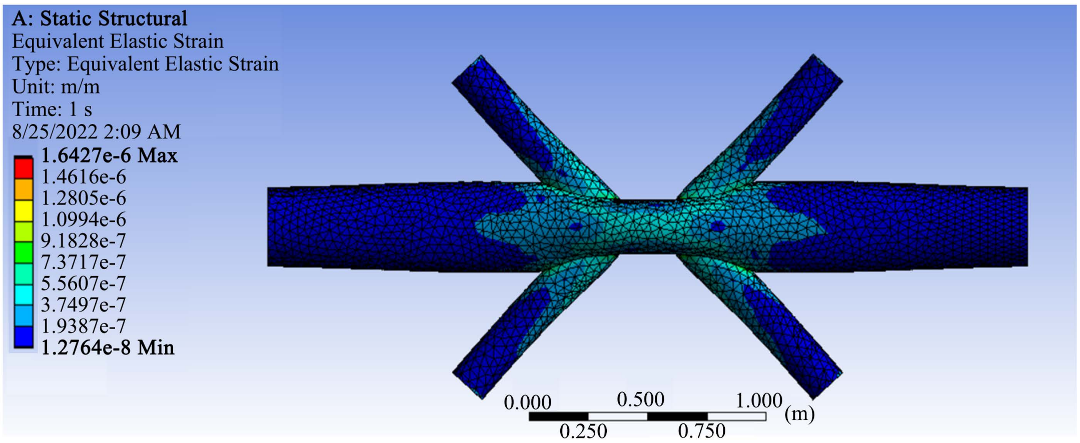

Figure 13. Total strain distribution of the joint in ANSYS.

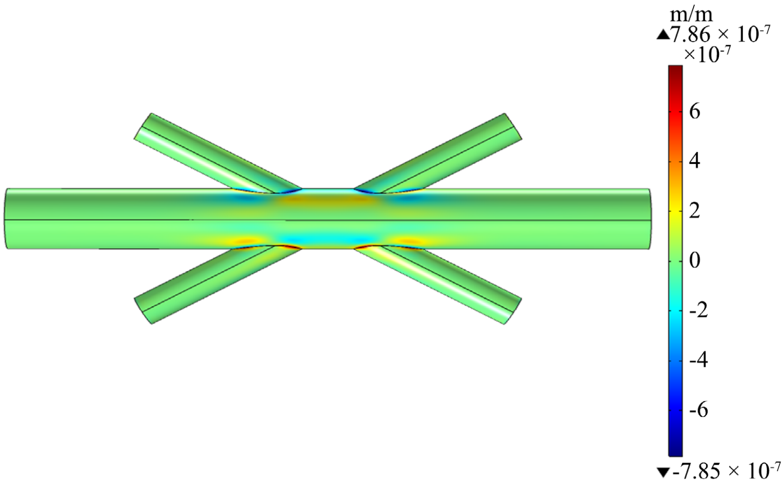

Figure 14. Total strain distribution of the joint in COMSOL Multiphysics.

It is also found that the strains in the brace-to-chord connection are highest at the inner surface of the brace, where the brace is in contact with the chord. The strains at the outer surface of the brace are relatively small due to the thinness of the brace material. The strain in the chord is highest at the point where the brace is in contact with the chord and decreases as the distance from this point increases. Moreover, the strains in the brace-to-chord connection are relatively higher in the circumferential direction due to the circular shape of the tubing. The strains in the radial direction are relatively small due to the thinness of the brace material.

Figure 15 shows the maximum deformation vs. load curve for different thicknesses. It is seen that the curve starts at the origin and increases as the load increases. In the initial stages, the curve is relatively linear, indicating that the joint is behaving elastically. As the load increases, the curve becomes non-linear, indicating that the joint is undergoing plastic deformation. At some point, the curve reaches a peak, indicating that the joint has reached its maximum load capacity. Beyond this point, the curve begins to level off, indicating that the joint is becoming increasingly stiff. Besides, for both simulations, it is seen that as the thicknesses of the joint increase, the value of maximum deformation decreases.

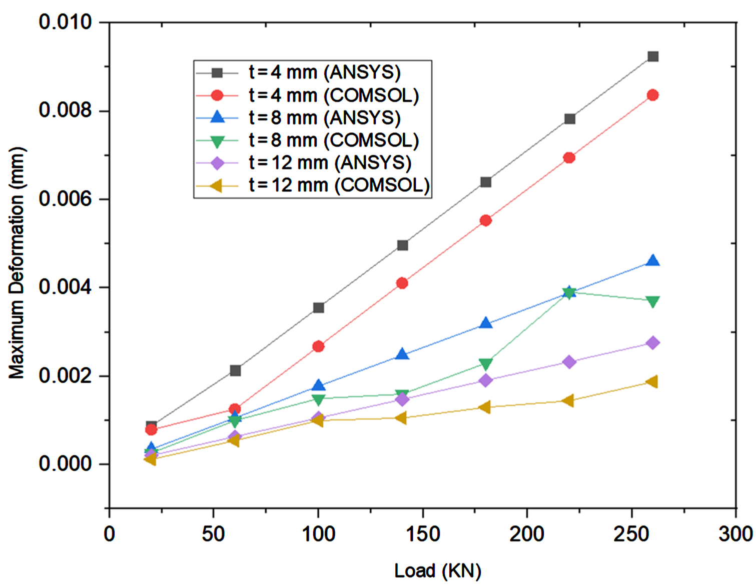

Figure 15. Variations in maximum deformation for different thicknesses.

Figure 16 illustrates the maximum stress vs. load curve. From the figure, it is seen that as the load is increased, the maximum stress experienced by the joint also increases. The slope of the curve in the initial stages is relatively steep, indicating that the stress increases rapidly as the load increases. As the load increases, the slope of the curve becomes less steep, indicating that the stress is increasing at a slower rate. Moreover, as the load approaches the yield strength of the material, the maximum stress reaches a peak and then levels off. Besides, the maximum value of stress decreases as the thickness of the joint increases.

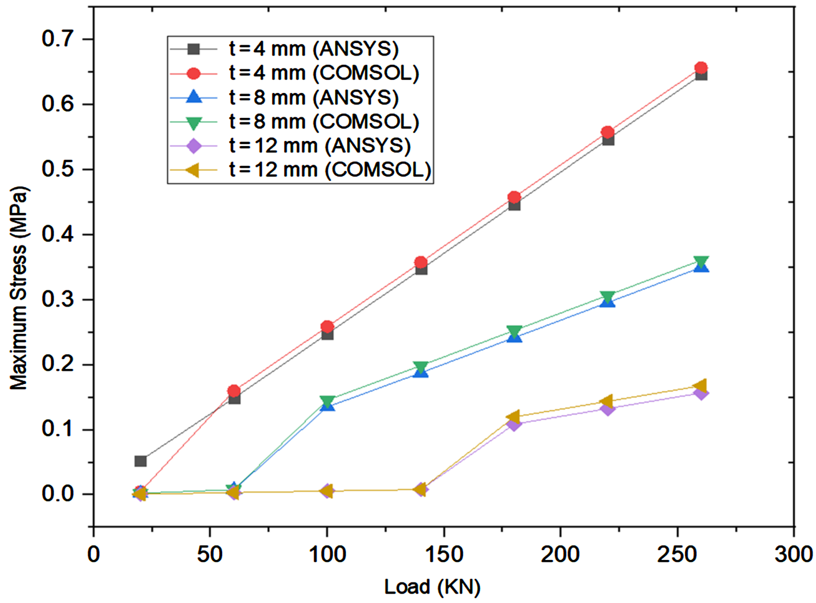

Figure 16. Variations in maximum stress for different thicknesses.

The maximum strain vs. load curve for different thicknesses is shown in **Figure 17**. It is seen from the figure that the maximum strain value decreases as the thickness of the joint increases. The curve typically starts at the origin, with zero load and zero strain. As the load increases, the strain in the joint also increases. The slope of the curve in this initial region is known as the elastic modulus of the material. The curve reaches a peak, known as the yield point, where plastic deformation begins to occur, and the slope of the curve becomes less steep. Beyond the yield point, the curve becomes relatively flat, and the joint can withstand larger loads without significant increases in strain.

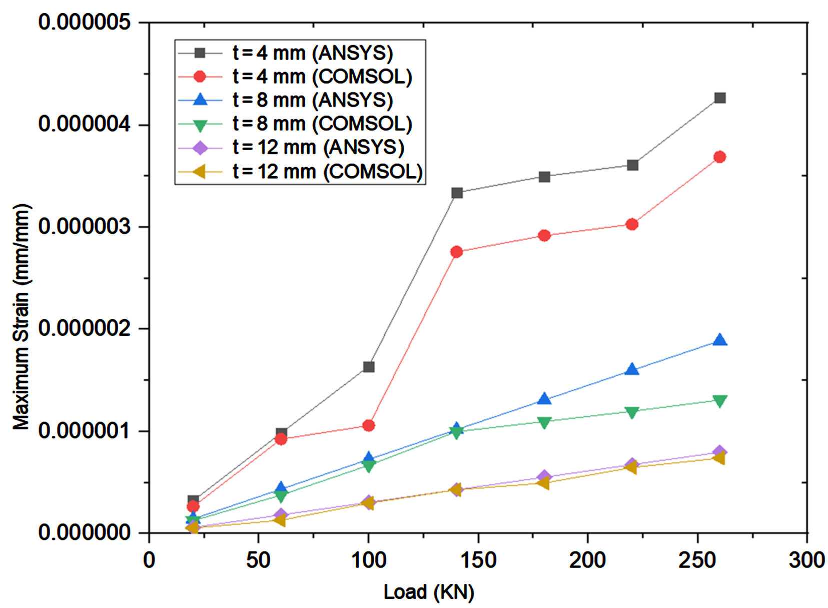

Figure 17. Variations in maximum strain for different thicknesses.

4. Conclusion

The study has analyzed the behavior of a tubular KK joint under compressive loading using the finite element method. The results of the analysis indicate that the joint is able to withstand the applied compressive loading without failure. The study has also highlighted the importance of considering the geometric nonlinearity of the joint when analyzing its behavior under loading. The study's results can be used as a reference for future work on the design and optimization of tubular KK joints for use in various applications. Additionally, the study's results can also be useful for those who are interested in understanding the behavior of tubular joints under compressive loading and the effect of different parameters on the joint's performance. Overall, this study is important in understanding tubular KK joints and their behavior under compressive loading. From this study, it can be concluded that for an equivalent stress distribution, maximum stress is found near the joint over the chord. Besides, minimum stress is found between the joint and the top of the chord. On the other hand, maximum deformation is found on the top of a brace. Again, minimum deformation is found at the top edge of the chord and maximum strain is found near the joint over a brace. Moreover, maximum stress, strain, and deformation values decrease as joint thickness increases. The insights derived from this study can serve as a valuable reference point for future research and development endeavors focused on enhancing tubular KK joint designs for diverse applications. In addition, the findings are pertinent for individuals and professionals seeking to gain a deeper comprehension of how tubular joints behave when subjected to compressive loading and the impact of different variables on their functionality.

Conflicts of Interest

The authors declare no conflicts of interest regarding the publication of this paper.

References

[1] Zhao, X., Han, S., Hu, K., Chen, Y. and Wu, A. (2015) Experimental Study on Static Behavior of Multi-Planar Overlapped CHS KK-Joints. In: de Miranda Batista, E., Vellasco, P. and de Lima, L.R.O., Eds., *Tubular Structures XV*, CRC Press, 505-512. https://doi.org/10.1201/b18410-75

[2] Chen, Y. and Wang, W. (2003) Flexural Behavior and Resistance of Uni-Planar KK and X Tubular Joints. *Steel and Composite Structures*, **3**, 123-140. https://doi.org/10.12989/scs.2003.3.2.123

[3] Santosh Sawant, S. and Muthumani, K. (2020) Analysis of Tubular Joint of Offshore Structure. *Journal of Physics*: *Conference Series*, **1716**, Article ID: 012012. https://doi.org/10.1088/1742-6596/1716/1/012012

[4] Santacruz, A. and Mikkelsen, O. (2021) Numerical Stress Analysis of Tubular Joints. *IOP Conference Series*: *Materials Science and Engineering*, **1201**, Article ID: 012032. https://doi.org/10.1088/1757-899x/1201/1/012032

[5] Oshogbunu, E.O., Wang, Y.C. and Stallard, T. (2021) Reliability and Applications of a New Design Method for Calculating Hot-Spot Stress in CHS Double K-Joints un-

der Arbitrary Combined Loading. *Structures*, **29**, 1610-1626. https://doi.org/10.1016/j.istruc.2020.11.057

[6] Zhao, X.Z., Qiu, S., Chen, Y.Y., Xu, X.B., Wu, X.F., Hu, K.H., *et al.* (2017) Behavior of Overlapped CHS K/KK-Joints with Different Welding Situations. In: Heidarpour, A. and Zhao, X.-L., Eds., *Tubular Structures XVI*, CRC Press, 647-655. https://doi.org/10.1201/9781351210843-80

[7] Forti, N.C.D.S., Requena, J.A.V. and Forti, T.L.D. (2017) Numerical Methodology for Analyses of Tubular KK Multiplanar Steel Joints. *REM—International Engineering Journal*, **70**, 157-165. https://doi.org/10.1590/0370-44672014700205

[8] Jingxia, Y., Yuliang, L., Chi, Z., Qi, Z. and Zhifan, D. (2016) Fatigue Strength Assessment on a Multiplanar Tubular KK-Joints by Scaled Model Test. *Journal of Offshore Mechanics and Arctic Engineering*, **138**, Article ID: 021602. https://doi.org/10.1115/1.4032158

[9] Aghaei, A., Gharabaghi, A.M. and Chenaghlou, M.R. (2006) New Parametric Equations for Estimating Stress Concentration Factors in Tubular KK-Joints Under Axial Loading. https://www.researchgate.net/publication/267758227

[10] Kadry, A.A., Ebid, A.M., Mokhtar, A.A., El-Ganzoury, E.N. and Haggag, S.A. (2022) Parametric Study of Unstiffened Multi-Planar Tubular KK-Joints. *Results in Engineering*, **14**, Article ID: 100400. https://doi.org/10.1016/j.rineng.2022.100400

[11] Ahmadi, H., Alinezhad, R. and Alizadeh Atalo, A. (2021) Stress Concentration Analysis of Internally Ring-Stiffened Two-Planar Tubular KK-Joints. *Ships and Offshore Structures*, **17**, 2203-2217. https://doi.org/10.1080/17445302.2021.1979922

[12] Mia, M.J., Islam, M., Kabir, A. and Islam, M. (2022) Numerical Analysis of Tubular XT Joint of Jacket Type Offshore Structures under Static Loading. *Bangladesh Maritime Journal*, **6**, 2520-1840.

[13] Chandrupatla Tirupathi, R. and Belegundu, A.D. (2009) Introduction to Finite Elements in Engineering. Pearson Education Taiwan Ltd.

[14] Gazi Karakoc, S.B. and Neilan, M. (2014) A C^0 Finite Element Method for the Biharmonic Problem without Extrinsic Penalization. *Numerical Methods for Partial Differential Equations*, **30**, 1254-1278. https://doi.org/10.1002/num.21868

[15] Singer, F.L. and Pytel, A. (1980) Strength of Materials. Harper& Row.

[16] Budynas, R.G. and Nisbett, J.K. (2014) Shigley's Mechanical Engineering Design. McGraw-Hill Higher Education.