Mesh Sensitivity Study of Steel Tubular T-joints for the Computation of Stress Concentration Factors

Althaf Mohammed*, Dasari Sandhya Rani, Yogesh M. Desai

Department of Civil Engineering, IIT Bombay, Mumbai-400076, India

*Corresponding author's e-mail: althaf2901@gmail.com

doi: https://doi.org/10.21467/proceedings.156.1

ABSTRACT

The stress concentration factor (SCF) computation is a key element in the fatigue assessment of offshore tubular joints. Mesh sensitivity research is conducted to identify the optimum mesh controls needed to create an extensive FE model library to develop an AI prediction model for the SCF prediction. Nine finite element models with varying mesh controls are analysed for the mesh sensitivity study. The optimum mesh controls for the Finite Element Model for estimating the SCF of steel tubular joints were identified. A python code for an automated mesh generation for tubular T joints with identified mesh controls using APDL commands in ANSYS is developed.

Keywords: Finite Element Analysis, Tubular Joints, Mesh sensitivity controls, Stress Concentration Factors

1 Introduction

Offshore structures are often exposed to cyclic wave loads, which may need to be considered to accumulate fatigue damage. The computation of the SCF is a key element in the fatigue assessment of offshore tubular joints. Welded tubular structures are widely used in Offshore structures due to their more aesthetic appearance and structural performance. Because of the intricacies in the geometry of welded tubular joints, local stresses are not evenly distributed. Nominal Stress is the stress estimated in a structure that only considers the general geometric effect and ignores the stress accumulation brought on by the discontinuities and welding in the joints. The structural stress on a hot spot, or where a fatigue crack is likely to start, is known as hot spot stress. Stress Concentration is the ratio of Hot Spot stress to Nominal stress. SCF may be calculated using experimental, empirical, and numerical techniques. The hot spot locations for steel tubular joints considered for the fatigue life assessment are the chord crown, the chord saddle, the brace crown, and the brace saddle shown in Figure 1.

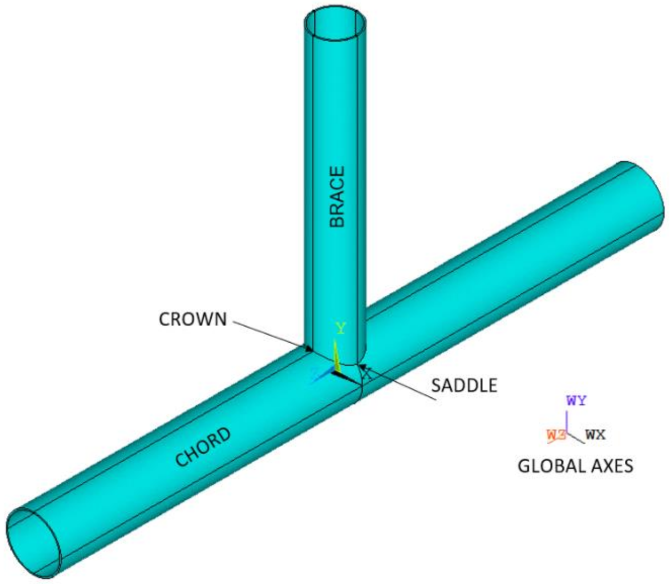


Figure 1: *Typical tubular T-joint showing crown and saddle*

Proceedings DOI: 10.21467/proceedings.156; Series: AIJR Proceedings; ISSN: 2582-3922; ISBN: 978-81-961472-7-3

The computation of SCF using experimental is practically infeasible due to enormous heterogeneity in the design and numerous human errors. Empirical techniques are based on non-dimensional geometric parameters and may fail to predict the actual SCF, which may affect figuring out the fatigue life. All the above drawbacks are handled smoothly using numerical techniques. Additionally, the computational efficiency of current Finite Element software and systems encourages engineers to use FE Analysis to identify SCF precisely.

An extensive FE model library was necessary for research to create an AI prediction model for the SCF value of the steel tubular T joints. Mesh sensitivity research is conducted to identify the optimum mesh controls to save computation time in creating the database without affecting the accuracy of results. This mesh sensitivity analysis uses the finite element model studied by A. Santacruz and O. Mikkelsen [1].

2 Mesh sensitivity study

2.1 Finite Element Modelling

The non-dimensional geometric parameters of the steel tubular T joint considered for the study are shown in Table 1. There is symmetry in geometry, loading and boundary conditions about two planes for axial loading, one plane each for out-of-plane and in-plane bending. So, one-fourth of the geometry is modelled for axial loading and one-half of the geometry is modelled for out-of-plane and in-plane bending in the ANSYS using Ansys Parametric Design Language. Information on the weld profile was taken directly from the publication by Santacruz *et al.* [1]. The FE model is simulated using the pre-written ANSYS-APDL commands as shown in Figure 2.

Table 1: *Non-Dimensional Geometric Parameters*

Parameter	Symbol	Ratio	Value	Range
Alpha	α	2L/D	11.87	4-40
Beta	β	d/D	0.52	0.2-1
Gamma	γ	D/2T	13.69	8-32
Tau	τ	t/T	1	0.2-1

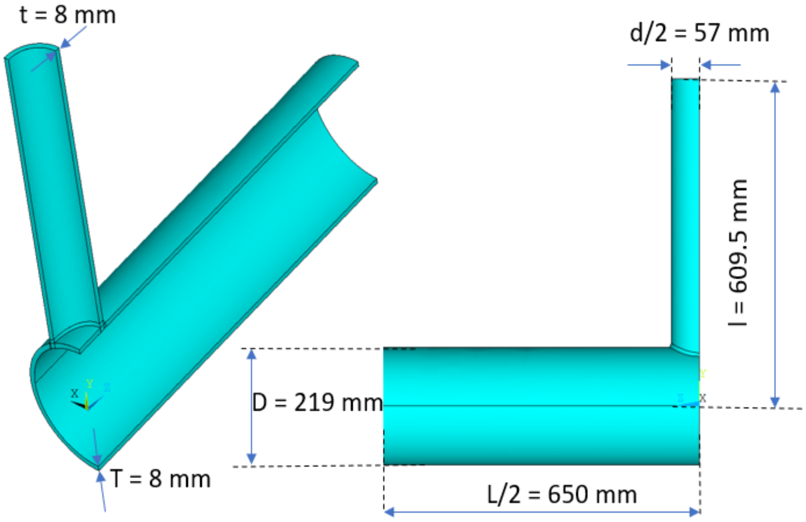


Figure 2: *Simulated FE model using the pre-written ANSYS-APDL commands.*

2.2 Mesh Generation, Applying Load and Boundary Condition

For studying mesh sensitivity, ANSYS's Three-Dimensional 20 Node Brick element (solid 186) is utilised to generate finite elements. In the neighbourhood of the weld region, a finer mesh is adopted and coarser as the elements approach the farther regime. The simulated geometry is partitioned using the identified mesh controls. With these controls, the meshing algorithm is coded with the help of APDL commands for axial, out-of-plane and in-plane bending and Figure 3 shows generated FE models. Figure 4 illustrates the key parameters for mesh sensitivity analysis.

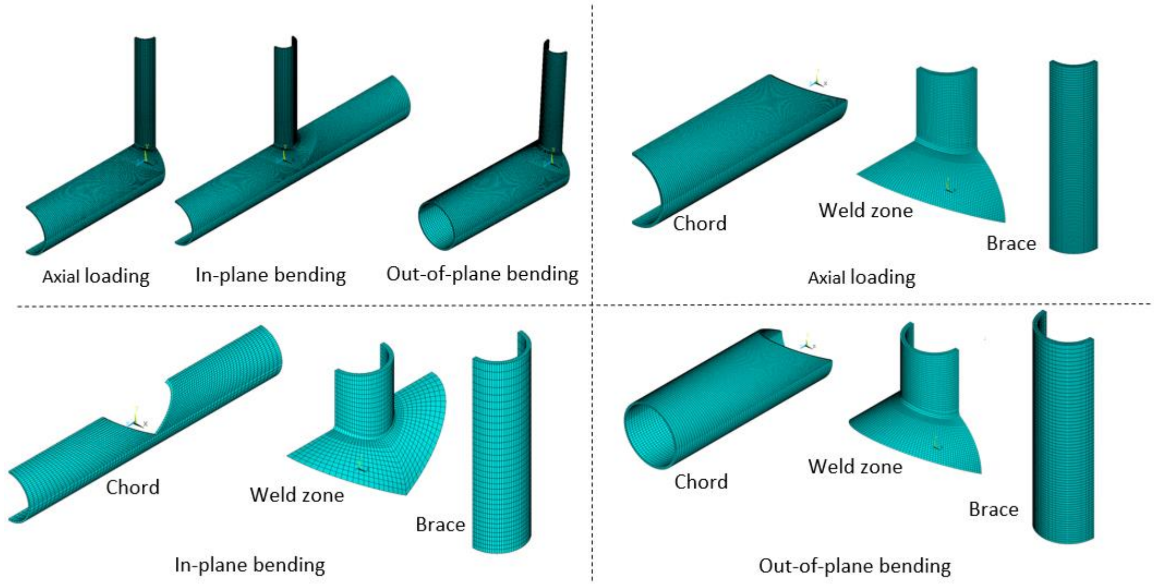
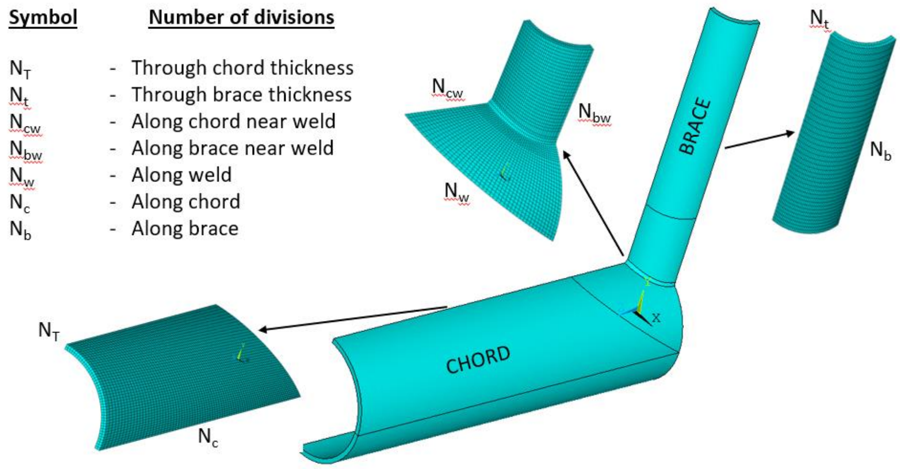


Figure 3: *FE Models for axial, out-of-plane and in-plane bending.*



Symbol	Number of divisions
N_T	- Through chord thickness
N_t	- Through brace thickness
N_{cw}	- Along chord near weld
N_{bw}	- Along brace near weld
N_w	- Along weld
N_c	- Along chord
N_b	- Along brace

Figure 4: *Key Parameters for mesh sensitivity study*

Displacements in all directions are restricted at the chord end to simulate the fixed-fixed boundary condition for axial, out-of-plane, and in-plane bending. For axial loading, unit pressure (1 MPa) is applied at the brace end. By applying the appropriate stresses at associated nodes, a linear stress variation with zero stress at the neutral axis and unit pressure (1 MPa) at the extreme point is stimulated for both out-of-plane and in-plane bending.

Displacements in the x and z directions for all the nodes in the YZ and XY symmetric planes are arrested for simulating the symmetric boundary condition about YZ and XY planes, respectively, for axial loading. Similarly, for the remaining loading conditions, displacement in the x direction for all nodes in YZ symmetric plane is arrested for in-plane bending and displacement in the z direction for all nodes in YZ symmetric plane is arrested for out-of-plane bending. Figure 5 depicts the said loads and boundary conditions for all the FE Models.

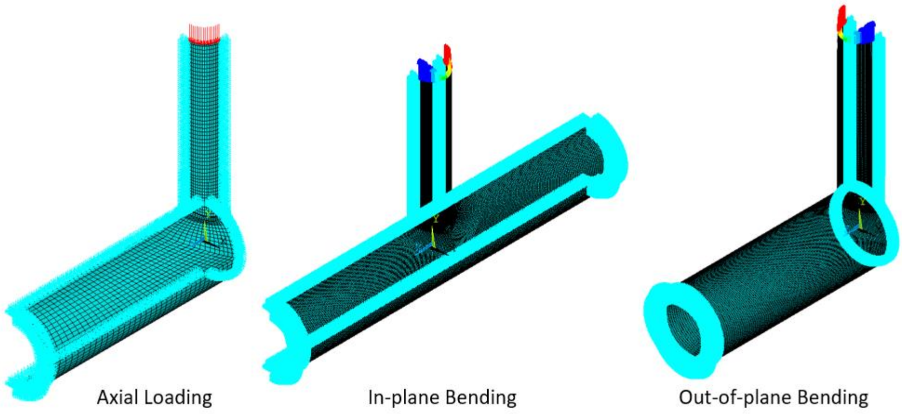


Figure 5: *Loads and boundary conditions for FE Models*

3 Results and Discussion

Nine finite element models with varying mesh controls, as shown in Table 2, are analysed for the mesh sensitivity study. All the elements in meshes a, b, and c maintain aspect ratios of one near the weld and two farther away. The aspect ratio near the weld regime is increased from 1.2 to 1,7 with an equally spaced interval of 0.1 for meshes d to i and a corresponding number of divisions along the chord and brace around the weld zone, as shown in Table 2 to achieve the finer mesh without increasing the number of divisions through the chord and brace thickness.

Table 2: *Mesh Controls for the finite element models*

Mesh Name	N_T	N_t	N_{cw}	N_{bw}	N_w	N_c	N_b
Mesh a	1	1	13	21	16	36	46
Mesh b	2	2	16	27	33	60	80
Mesh c	3	3	26	42	49	90	118
Mesh d	3	3	30	49	49	98	129
Mesh e	3	3	33	54	49	102	133
Mesh f	3	3	35	57	49	105	138
Mesh g	3	3	37	60	49	108	142
Mesh h	3	3	41	67	49	111	145
Mesh i	3	3	43	70	49	114	149

The stress perpendicular to the weld toe is taken for the computation of the SCF so the global z-direction for the chord crown, and global y-direction for the brace crown and saddle. Whereas at the chord saddle, stress transformations are used due to their circular geometry. Using the extrapolation method described in DNVGL-RP-C203 [2], non-linear extrapolation is applied for SCF at the brace crown and chord saddle whereas linear extrapolation is used for other locations. The SCF values with the number of elements for all the corresponding loads are shown in Figures 6-8 respectively.

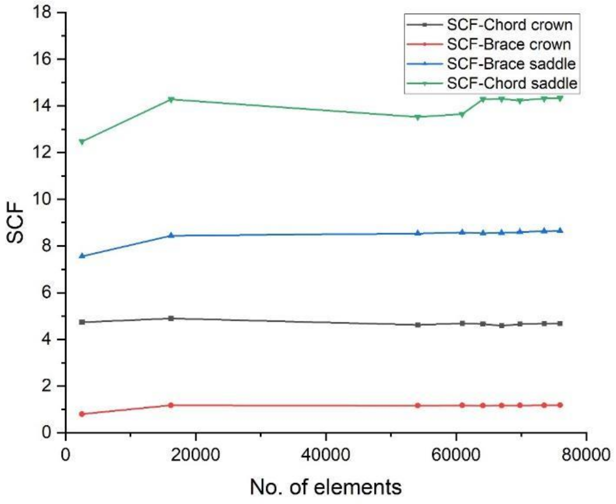


Figure 6: *Variation of SCF vs No. of elements for axial loading*

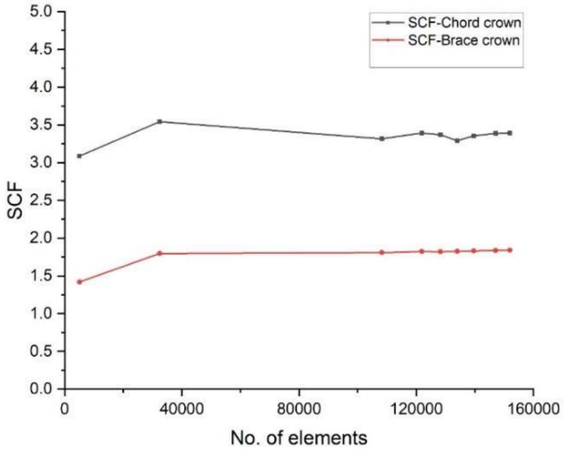


Figure 7: *Variation of SCF vs No. of elements for in-plane bending.*

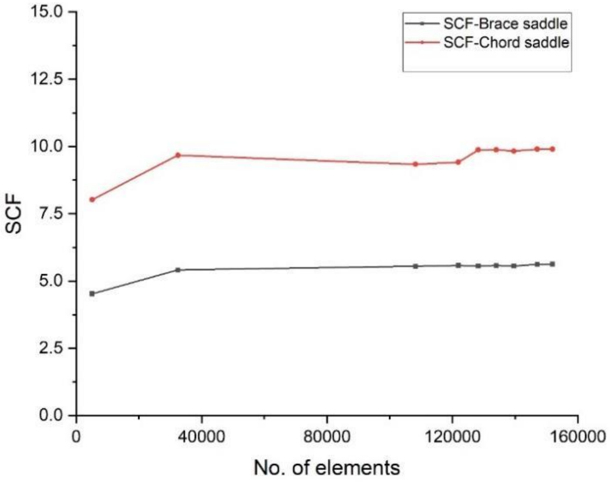


Figure 8: *Variation of SCF vs No. of elements for out-of-plane bending*

SCF values show a declining trend for all the load cases by changing the number of divisions through the chord and brace thickness from two to three, so the optimum number of divisions can be two or three. Except for the chord crown and chord saddle for axial loading and the chord crown and brace saddle for in-plane and out-of-plane bending, the relative percentage error in the SCF values for all load scenarios is determined to be less than 5%. As the mesh switched from Mesh c to Mesh e for axial loading, the SCF value at the chord saddle shot up from 13.5 to 14.3, but at all the other points, it remained relatively constant. While the SCF value at the brace crown is stable, the SCF at the chord crown increases from 3.32 to 3.39 when the mesh changes from Mesh c to Mesh i with a decline of 3.29 for Mesh f for in-plane bending. As the mesh switched from Mesh c to Mesh e, the SCF value at the chord saddle increased from 9.34 to 9.87, however, the value at the brace saddle remained steady for out-of-plane bending since the ratio of brace to chord diameter is 0.52 for this model.

Finite Element Models are run in 32 GB RAM, Intel Xeon processor system, and the time for analysis is also noted and is depicted in Figure 9. The analysis time for axial loading showed a linear trend since the number of elements for all the models varied from 2543 to 75974. However, the analysis time showed a linear trend for out-of-plane and in-plane bending till the Mesh f, which had 133976 elements. An abrupt increase in analysis time is shown for Mesh g, which had 139604 elements, and a linear trend continued until Mesh i, which had 151948 elements.

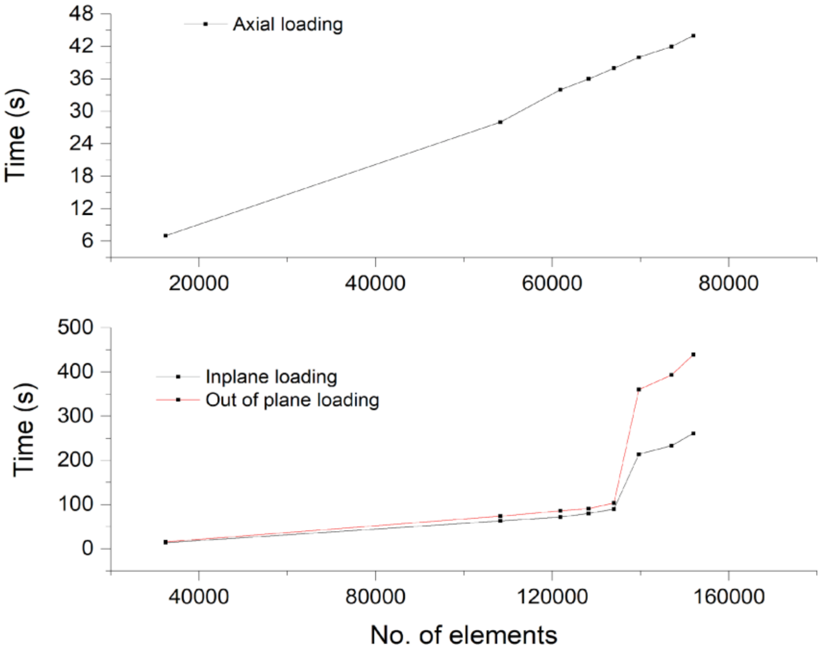


Figure 9: *Computational Time vs No. of elements*

4 Conclusion

The optimum mesh controls for the Finite Element Model for estimating the SCF of steel tubular joints were identified to generate the FE database for training the AI predictive model. It was recognised that the result's precision could be maintained by restricting the number of divisions across the chord and brace thickness to three, and the optimum mesh controls are identified to get finer mesh at the welding phase. Additionally, it was found that using a higher-aspect ratio towards the brace and chord end by employing the St. Venant principle, which was not demonstrated in the study, did not significantly change the results. The outcome of this study can be stated as the development of python code for an automated optimum mesh generation for tubular T joints with identified mesh controls using APDL commands in ANSYS.

5 Declarations

5.1 Acknowledgements

The authors gratefully recognize the resources offered by the Indian Institute of Technology Bombay's Centre for Computational Engineering and Science.

5.2 Publisher’s Note

AIJR remains neutral with regard to jurisdictional claims in institutional affiliations.

How to Cite

Mohammed *et al.* (2023). Mesh Sensitivity Study of Steel Tubular T-joints for the Computation of Stress Concentration Factors. *AIJR Proceedings*, 1-7. https://doi.org/10.21467/proceedings.156.1

References

[1] A. Santacruz and O. Mikkelsen, “Numerical stress analysis of tubular joints,” *IOP Conf Ser Mater Sci Eng*, vol. 1201, no. 1, Nov. 2021, doi: 10.1088/1757-899X/1201/1/012032.

[2] Det Norske Veritas, “DNV-RP-C203 Fatigue design of offshore steel structures,” Apr. 2016. https://www.dnv.com/oilgas/download/dnv-rp-c203-fatigue-design-of-offshore-steel-structures.html (accessed Apr. 12, 2023).