Parametric Study of Mesh for Fatigue Assessment of Tubular K-joints using Numerical Methods

Jorge Mendoza Espinosa [a,b], Sebastian Schafhirt [a], Michael Muskulus [a]

[a] Department of Civil and Environmental Engineering, Norwegian University of Science and Technology. Trondheim 7491, Norway

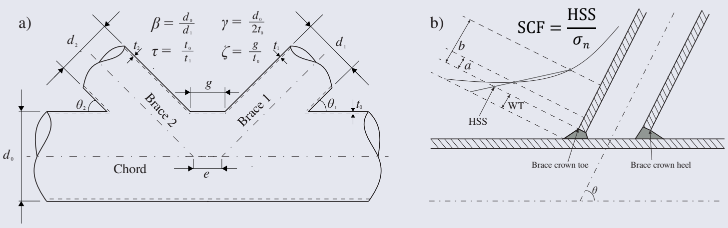
[b] Ramboll Wind. Hamburg 20097, Germany

Abstract

Wind turbine jacket structures are complex structures, whose **joints** design is generally driven by **fatigue**. These joints, along with their complex **welds**, are of special interest in terms of cost reduction. Therefore, a thorough analysis and understanding of the background behind the assessment proposed in guidelines is motivated. The paper presents a study of the **influence of meshing** for the assessment of tubular **K-joints** following the **hot-spot approach** using **numerical methods**. The accuracy of the results is discussed for several mesh layouts. Influence of the **mesh density, element shape** and **element type** are investigated. Furthermore, a parametric study is performed in order to see the variation in the results for different conventional geometry situations. The hot-spot method is proved to be robust regarding mesh regularity. However, the efficiency of irregular mesh models is very low and an asymptotic behavior that tends to a constant solution for increasing number of elements is sometimes found for very high number of nodes. Conclusions can be drawn for which cases it is worth to invest time in semi-automatic meshing. A discussion is done regarding which element size and type is better regarding accuracy and computational time.

Method

K-joint is modelled parametrically using FEM simulations in Ansys©. Hot-spot stress (HSS) is computed as the linear extrapolation to the weld toe as recommended in DNV-GL [1]. Stress Concentration Factor (SCF) is computed at the brace weld toe position. Standard steel and elastic behavior is used in all models.

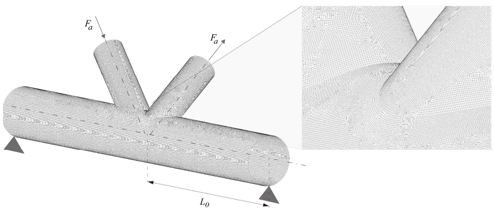


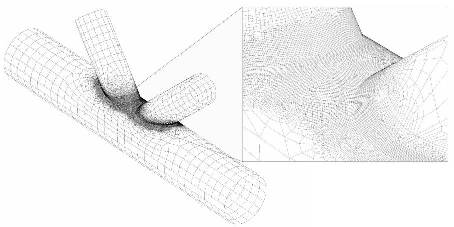
Influence of Element Regularity

Two mesh layouts are compared, i.e. **Automatic** meshing and **Semi-automatic** meshing.

Automatic meshing

Mesh is generated using ANSYS© built in subroutines. Element regularity is quite random at the chord-brace intersection and irregular elements are present

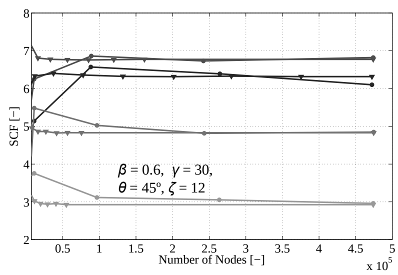




Semi-automatic meshing

Regular elements are present at the joint influenced area. Mesh refinement in this area can be modified parametrically.

44 FEM simulations are run to compare both kind of meshing. SCF is computed at the brace toe position.

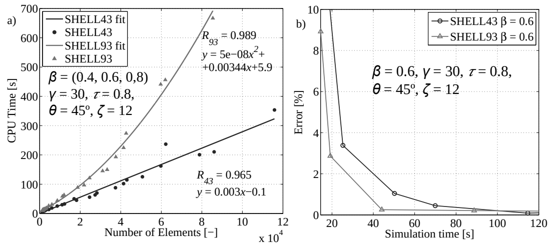


Irreg. and Reg. refer to the automatic and semi-automatic mesh layouts above.

Convergence of the solution to a constant value for increasing number of nodes is clear for the semi-automatic mesh models. An asymptotic tendency is not obtained for the automatic mesh models for all cases until a great refinement is set.
Solutions between both kind of models match for increasing mesh density. This grants the irregular mesh model reliability for a dense enough mesh.

Influence of Element Type

Two element types are compared: 4-node SHELL43 and 8-node SHELL93. **60 FEM simulations** are used for this investigation.



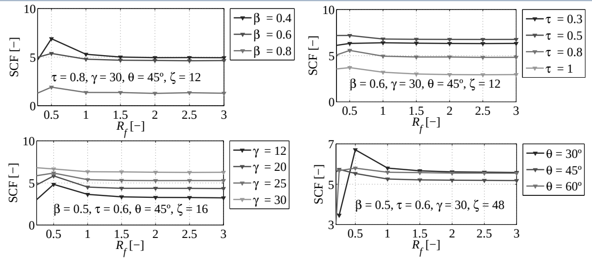
An error of less than 1% for SHELL93 is found for an element size of t_1 x t_1 and t_{CPU} = 35 s. Same precision requires around 55 s for SHELL43.

Elem. size	Element Type	# Elements	# Nodes	t_{CPU} [s]	SCF [-]	Error [%]
$2t_1$ x $2t_1$	SHELL43	3376	3398	13	5.38	16.13
$2t_1$ x $2t_1$	SHELL93	3348	10088	19	4.67	2.87
t_1 x t_1	SHELL43	8654	8672	25	4.79	3.38
t_1 x t_1	SHELL93	8711	26177	44	4.55	0.26
$1/2t_1$ x $1/2t_1$	SHELL43	28249	28264	70	4.65	0.45
$1/2t_1$ x $1/2t_1$	SHELL93	31055	93191	145	4.55	0.16
$1/3t_1$ x $1/3t_1$	SHELL43	59776	59766	162	4.64	0.22
$1/3t_1$ x $1/3t_1$	SHELL93	59693	179055	441	4.54	0.00
$2/7t_1$ x $2/7t_1$	SHELL43	78836	78811	201	4.63	0.00
$2/7t_1$ x $2/7t_1$	SHELL93	87484	262416	688	4.54	0.00

Results for both element type do not match, i.e. a difference of 2% exist. Therefore, it would be unrealistic to ask for an accuracy higher than that. Error in the computation of SCF is done with respect to $2/7t_1$ x $2/7t_1$ results.

Influence of Mesh Density

147 FEM simulations are run varying the refinement factor R_f. Semi-automatic model using SHELL43 is used.



Element size: $A_{El} = \frac{t_1}{R_f}$ x $\frac{t_1}{R_f}$

Guidelines recommend the use of an element size from R_f = 1 up to R_f = ½. For some cases, this may lead to underconservative solutions, e.g. the top-right plot for $\tau = 0.3$

Conclusions

A parametric study to investigate the influence of meshing for the computation of SCF for the hot-spot method was carried out. Several local FEM models are built to investigate the effect of **mesh density, regularity of the elements and element type**.

Generally speaking, **automatically generated meshes** do not provide a good balance between accuracy and computational time. **Great refinement is needed** in order to provide a trustworthy solution. Solutions between the regular mesh model and the automatically generated mesh models match when the number of nodes is increased sufficiently. Thus, their use can be justified for certain cases. They can be a better solution in certain situations since they do not require time to be spent in the manual definition of patterns to create a regular mesh.

8-node elements are more efficient than **4-node elements** for the accuracy required in the hot-spot method. SCF obtained by using both element types do not match, i.e. a difference of around **2%** exist.

Influence of the refinement of the joint influenced area was investigated. For most of the tested geometry situations, the **most efficient element size is t_1 x t_1** However, this is not a general rule. Using a smaller element size could yield underconservative solutions. It is recommended to always perform a mesh density parametric study to ensure that the solution is accurate enough.

Acknowledgements

The present study has been done at the Department of Civil and Environmental Engineering of the Norwegian University of Science and Technology. The authors would like to thank Dr.-Ing. Marc Voβbeck from Ramboll for the original idea here developed.

References

1. Fatigue Design of Oshore Steel Structures. DNVGL-RP-C203; Oslo, Norway: DNV GL AS; 2016.
2. S. A. Karamanos A. Romeijn, J.W.. Stress Concentrations in Tubular Gap K-joints: Mechanics and Fatigue Design. Engineering Structures 2000;22(7):4–14.
3. Romeijn, A.. Stress and Strain Concentration Factors of Welded Multiplanar Tubular Joints. PhD dissertation; Delft University of Technology; Civil Engineering Department; 1994.
4. O. Doerk W. Fricke, C.W.. Comparison of Dierent Calculation Methods for Structural Stresses at Welded Joints. International Journal of Fatigue 2003;25:359–69.

Contact: Jorge Mendoza Espinosa (jorgemendozaesp@gmail.com)

EERA DeepWind'2017
14th Deep Sea Offshore Wind R&D Conference
Trondheim, 18 – 20 January 2017

EERA European Energy Research Alliance

NOWITECH Norwegian Research Centre for Offshore Wind Technology