Adaptive mesh generation procedures for thin-walled tubular structures

Lie, S. T.; Nguyen, T. B. N.; Chiew, Sing Ping; Lee, Chi King

2009

Lee, C. K., Chiew, S., Lie, S., & Nguyen, T. (2010). Adaptive mesh generation procedures for thin-walled tubular structures. *Finite Elements in Analysis and Design*, 46(1-2), 114-131.

<https://hdl.handle.net/10356/102287>

<https://doi.org/10.1016/j.finel.2009.06.011>

© 2009 Elsevier. This is the author created version of a work that has been peer reviewed and accepted for publication by *Finite Element in Analysis and Design*, Elsevier. It incorporates referee's comments but changes resulting from the publishing process, such as copyediting, structural formatting, may not be reflected in this document. The published version is available at: [DOI: <http://dx.doi.org/10.1016/j.finel.2009.06.011>].

Downloaded on 26 Apr 2023 22:53:06 SGT

Adaptive Mesh Generation Procedures for Thin-walled Tubular Structures

C. K. Lee, S. P. Chiew, S. T. Lie and T. B. N. Nguyen*

Division of Structures and Mechanics
School of Civil and Environmental Engineering
Nanyang Technological University
Singapore

* Email: ccklee@ntu.edu.sg

Abstract

In this paper, a family of adaptive mesh generation schemes specially designed for finite element modelling of structural hollow section (SHS) tubular joint is presented. This family of adaptive mesh generation schemes is implemented based on a series of realistic and consistent geometrical models which is founded on measurements obtained from real structures. The underlying geometrical models provide the definitions of different levels of geometrical details and special features that could appear at different stages of the life cycle of the structure. The adaptive mesh generation schemes accompanying the geometrical models are capable of discretizing the SHS tubular joints into different forms of finite element meshes including pure surface meshes, hybrid meshes with surface and solid elements, and full 3D solid element meshes with or without welding and crack details. As a result, a hierarchical adaptive modelling procedure could be developed to assess the performance of the structures for their whole life cycle from quasi-static failure strength analysis to long term fatigue and fracture behaviours under cyclic loadings. In addition, all the mesh generators in this family are adaptive mesh generators such that the discretization error of the corresponding FE models could be effectively controlled by combining them with appropriate adaptive refinement schemes.

KEY WORDS: thin-walled structures; structural hollow section tubular joints; consistent geometrical modelling; adaptive mesh generation; hierarchical adaptive modelling analyses

1. Introduction

In many structural engineering applications, the accuracy of the finite element (FE) analysis is critical to the design and safety of the structures. Hence, error estimation and controls by using adaptive refinement is traditionally a hot topic in the research of the FE methods. As an adequate discretization of the problem domain is the prerequisite for any adaptive analysis algorithm, much effort has been spent on the development of adaptive 2D [1-4], surface [5-8] and volume [9-12] mesh generation procedures.

Traditionally, there is a clear distinction between surface and volume mesh generation procedures on the treatments of the problem domain. In surface mesh generation, the problem domain is idealized and discretized as a surface with no thickness while the thickness of the domain is represented by the nodal thickness without any actual discretization. Two well known examples in solid mechanics are the plate and shell FE formations. In these formulations, successful adaptive algorithms had been developed [13-15]. However, no matter how effective of such refinement algorithms, they could not able to improve the accuracy of the FE solutions beyond the fundamental assumptions of the underlying formulations. For example, no shear stress and normal stress information for a thin-walled structure shall be available if it is modelled by the thin plate formulation.

In volume mesh generation, the problem domain is treated as full 3D solid and discretization is carried out in all dimensions. Theoretically, no loss in geometrical details shall be occurred from dimensional reduction and the models developed should provide the most comprehensive solutions. However, for problems with high solution gradient, in order to obtain accurate modelling results [16-17], full 3D models with huge numbers of degree of freedom are often needed. Furthermore, for applications involving complex domain geometry, the volume mesh generation itself is not a trivial task which is also subjected to intensive research [9-12]. To alleviate such problems, well designed 3D adaptive mesh generators [9-11], error estimation methods and adaptive algorithms [16-19] were developed. However, while adaptive refinement may be an effective tool to cut down the computational cost for problems with high solution gradient, it provides no or little help if the smallest characteristic dimension of the problem domain is one order less than other dimensions. In this case, if no dimension reduction is carried out, to represent the problem domain correctly shall require elements with sizes smaller than the

smallest characteristic dimension. Obviously, such requirement eventually leads to a huge number of elements and hence expensive computational cost.

One classical example for such situation in structural mechanics is the analysis of thin-walled structures: If no dimension reduction is applied to simplify the structures into plate/shell models, a huge numbers of elements are needed to discretize the structure into 3D meshes. However, idealization of the problem domain into shell model may lead to irrecoverable loss of important details such as the complex stress concentration near the junctions of the structures. In fact, in thin-walled structures analyses such a dilemma is faced by structural engineers in their everyday work. One typical example is the modelling of structural hollow section (SHS) tubular joints which are extensively used in offshore engineering. In the design process of many offshore structures, it is a fundamental requirement to predict the responses of the SHS tubular joints in different stages of their service life. Furthermore, in order to evaluate the performance and the safety of a SHS tubular joint during its service life, different levels of response details, from the overall failure strength to the stress distribution along the joint intersections, are needed. Obviously, using a single model with fixed geometrical details (e.g. a full 3D model) could not lead to economic solutions for such problems.

A possible solution for such a dilemma is to develop and adopt an *adaptive* modeling procedure that employs a series of models with different “resolutions” to fulfill the progressive demands in design details. To maintain accuracy and consistency, this series of models should be developed in a *hierarchical* manner such that critical geometrical characteristics could be inherited from the simplest model to the most complex models.

In this paper, such a hierarchical modeling procedure for the analysis of SHS tubular joints is presented. When comparing with the other adaptive refinement mesh generators [2-4, 6-11, 16, 17] which are mainly designed for generating adaptive meshes to reduce the discretization error of the FE solution, the presented modeling procedure extends the concept of adaptivity to the model level. A *family* of adaptive mesh generators which are derived based on a series of consistent and realistic geometrical models with increasing geometrical details and features are developed. While adaptivity in modelling level could be achieved through the proper used of the mesh generator family in a hierarchical manner, each mesh generator in this family itself is an adaptive mesh generator so that the discretization error of the FE model could be reduced by using them with appropriate adaptive refinement strategies.

In the next section, a comprehensive summary on the background information of SHS tubular joints and the modelling requirements for assessing the performance of this type of structures are given. They are then followed by an overview of the modeling and mesh generation concepts employed in this study. In Section 4, the key properties of the series of geometrical models adopted are presented. The algorithms used for generating the surface and solid meshes for FE modelling are presented in Section 5. Examples of models and meshes generating for a real structure using the suggested approach are then presented in Section 6. Finally, conclusions possible extensions of the presented work are given.

2. *Fatigue and fracture modellings of tubular thin-walled structures*

Circular hollow sections (CHS) are widely used in offshore engineering due to their excellent structural and mechanical properties such as high buckling strength, low weight ratio and small friction and drag coefficients. During the construction of offshore structures, SHS members are jointed together by welding along the profiled ends of the smaller section (brace) onto the circumference of the larger section (chord) (Fig. 1) to form a *tubular joint*. In the design of offshore structures, the performance of both the CHS members and the joints require carefully assessments. Since for many offshore structures operating in deep sea locations, repairs or retrofitting works are expensive and difficult to carry out, it is compulsory in all offshore structure design codes that limit state checks are required for both the ultimate strength and the long term fatigue performance of CHS joints. Normally, the following assessments are routinely required for the joints.

- (1) Strength assessments under quasi-static and dynamic loadings.
- (2) Long term fatigue and fracture performance of the joint under cyclic loadings. Fatigue and fracture assessments are needed for both uncracked joints for stress concentration along the welding path and for cracked joints for the crack growth speed and residual life predictions [20].
- (3) Safety and stability of the joint under extreme loading cases such as fire and blast loading.

In order to fulfill the above assessment requirements, the following models with different levels of geometrical and mechanical details are needed to be constructed for safety and performance evaluations.

- (1) A surface model without welding and crack details for ultimate strength estimation.

- (2) A hybrid mode with 3D solid details at the junction of the intact joint for fast fatigue assessment. This normally is applied to those secondary joints at where their failures shall not affect the overall integrity of the structures.
- (3) A 3D solid model with welding details for accurate stress concentration factor (SCF) and hot spot stress (HSS) predictions near the junction [20]. As the fatigue performance of the joint is highly sensitive to the HSS value, checking of the peak HSS is a compulsory procedure for all primary joint at where any failure could affect the overall integrity of the structure.
- (4) A 3D solid model of the cracked joint which includes surface crack details. This model is essential for accessing the residual fatigue life and the safety factor against fracture when surface crack is found in the structure.

It is obvious that for the above models, the geometrical and mechanical assumptions largely determine the *modeling error* and hence limited the application level of the model. Furthermore, if needed, adaptive refinement should be applied to reduce the *discretization error* of the resulted numerical models.

In this studying, the modeling of tubular joint under extreme loading conditions (e.g. high temperature or blast loading) are excluded. For such situations, more complex *evolutionary models* that could reflect the damaged configurations of the structures are needed. Moreover, it should be noted that by combining with a special mesh generation technique [8, 21], the hierarchical geometrical models presented here could be employed to model the early responses of the structure under such extreme conditions.

3. *An overview of the hierarchical modeling and mesh generation procedures*

The main paradigm of the current approach is very simple: Rather than to develop a number of unrelated models to carry out the assessment requirements separately, a more holistic approach to create a series of coherent, hierarchical models is employed. These series of models are linked up by a set of consistent modeling and mesh generation procedures which follow the analysis, design and fabrication sequence of the actual joints. More specifically, for a typical CHS tubular joint, the following analysis and design sequence are used in practice.

- (i) Selection of joint configuration (Fig. 2).
- (ii) Selection of CHS member sizes.
- (iii) Static strength checks.

- (iv) Assessment of long term fatigue performance.
- (v) Re-assessment of strength and fatigue behaviour of the joint if cracks are detected after some years' of service.

In this study, both the geometrical models and the mesh generation schemes are developed hierarchically based on the above order. From Fig. 2, it can be seen that for a tubular joint, the connection details are intrinsically depend on the joint configuration. Toward this end, in order to embrace a wide range of configurations, the partially overlapped CHS joint is selected in this study to demonstrate the model and mesh generation procedures. The partially overlapped CHS joint is selected because it is the most complicated planar joint configuration constructed in practice. Other joint configurations (e.g. T- Y- joints) could be constructed by either simplifying this joint type or by applying a sequence of mirroring, rotation and duplication operations to a simple Y-joint (e.g. V- X-, XT, XX and gap K- joints). A general configuration and notations for a partially overlapped CHS K-joint are shown in Fig. 3. The overall model/mesh generation sequence, which closely resembles the analysis and design sequence, is shown in Fig. 4.

In this study, two different surface models, namely, surface model without welding details (SURF_0W) and hybrid surface-solid model with welding details (HYBR_1W) are used. These models could be applied for static strength analysis and preliminary fatigue assessment. For fatigue assessment of intact joint, two solid models, namely, the SOLID_0W model (without welding details) and the SOLID_1W model (with welding details) are used. These two models are created based on the surface models by adding in the thickness information of the sections and the welding details. In general, with the inclusion of welding details the SOLID_1W model could give a more accurate estimation on the fatigue performance of the joint. However, when no reliable welding information are available, or when the quality of the actual welding dose not compile with the minimum standard, the SOLID_0W model should be used for conservative prediction. For the residual fatigue life estimation of a cracked joint, based on the solid model SOLID_1W, a more complex SOLID_CR model with surface crack details is developed.

One remark that for the overall generation process is that all the models, except the simplest surface model SURF_0W, inherit all the geometrical details from their preceding models (Fig. 5). As a result, a series of hierarchical geometrical models with increasing complexity is formed. Different levels of details are included in successive models by following the fabrication sequence of a real joint. In numerical modeling, this series of geometrical models allows a

consistent family of mesh generation schemes and numerical models to be created in a minimum mathematical complexity. Eventually, the FE models constructed by this approach shall give consistent modeling results with increasing in accuracy.

4. *Geometrical modeling*

In this study, the construction of the hierarchical geometrical model follows closely the fabrication sequence of a real joint. In addition, as the fatigue life of any tubular joint is highly sensitive to the welding details during fabrication [20], when constructing the welding and crack models, extensive references were made to the actual measurements obtained during and after actual full scale tests [22-24]. The whole hierarchical geometrical models could be divided into the following three distinctive levels.

- (i) Model for surface intersection.
- (ii) Model for welding details.
- (iii) Model for crack details.

4.1 *The surface intersection model (for SURF_0W, SOLID_0W)*

If no welding detail is considered, the surface intersection model SURF_0W could be constructed by considering the relatively simple geometrical solution of (circular) tube-to-tube intersection. For a non-overlapped joint (e.g. T- and V- joints in Fig. 2), only one intersection is considered. For an overlapped joint, two intersections following the construction sequence are considered. Obviously, to building a geometrical description for both cases, the fundamental geometrical problem needed to be solved is the intersection between two cylindrical surfaces. In this case, the double mapping method first suggested by Cao et al. [25] which was validated by Lie et al. [26] for a wide range of parameters in actual CHS joints was adopted. The essence of this method is to first unfold one of the circular section (normally the chord) to a flat plane (Fig. 6a). Intersection with the brace is then considered to generate an elliptical curve on the mapped plan (Fig. 6b). This elliptical curve is then further mapped to a circle with the driving angle α as the main parameter (Fig. 6c). The advantage of this double mapping approach is that the exact location of the intersection curve could be defined implicitly by the single parameter α . Furthermore, this procedure (equations listed in Fig. 6) is applicable to the practical range of θ , $[30^\circ \leq \theta \leq 90^\circ]$, even for the special case when $r=R$. Note that Fig. 6 only shows the mapping procedure for a

typical Y- joint. In case that a more complex joint type (Fig. 2) is considered, the above mapping could be repeated with additional coordinate systems. Fig. 7 shows the four coordinate systems employed for a partially overlapped K-joint. In this case, three intersection curves and one common junction point (Fig. 8) are generated in two intersection steps: first between the intersection of the through brace with the chord and then the lap brace with the through brace and the chord.

Since any CHS section can be defined by its inner and outer surfaces, the above intersection approach could be extended to create the solid model SOLID_0W. For non-overlapped joint type such as T/V/Y/X joints, the boundary of the solid model will be simply defined by intersections between the inner and outer brace surfaces with the outer surface of the chord. For a partially overlapped joint, three pairs of surface intersections define the boundary of the solid model. If one follows the fabrication sequence of a real joint (Fig. 9), the intersection curves are formed in the order shown Box 1. Note that the algorithm shown in Box 1 is robust enough for the special case of identical braces and chord diameters.

- *Intersection of through brace with chord*
 1. Outer surface of through brace ↔ Outer surface of chord
 2. Inner surface of through brace ↔ Outer surface chord
- *Intersection of lap brace with chord*
 3. Outer surface of lap brace ↔ Outer surface of chord
 4. Inner surface of lap brace ↔ Outer surface of chord
- *Intersection of lap brace with through brace*
 5. Outer surface of lap brace ↔ Outer surface through brace
 6. Inner surface of lap brace ↔ Outer surface through brace
- *Including the inner surface of the chord to define the solid part for the chord*

Box 1: Intersection sequence to form the solid model SOLID_0W of a partially overlapped CHS Y joint

4.2 The welding details model (HYBR_1W and SOLID_1W)

For any tubular connection, the quality of the welding details is critical to both the static strength and the fatigue performance of the connection [20, 27, 28]. Fig. 10, shows the typical welding procedure for a tubular connection and the associated geometrical parameters. For any load

bearing connection, all design guides [20, 29, 30] specify that *full penetration welding* must be achieved everywhere along the welding perimeter. Furthermore, a certain minimum welding contact thickness T_w must be materialized and compulsory inspections are norms in construction practices.

The welding model used in this study is designed to compile with the industrial standards [20, 29, 30] and its reliability is confirmed by checking models created against measurements obtained from actual full scale joints [22-24, 27]. In most design guides, the minimum welding contact thickness at a given point along the welding perimeter is defined as a *discontinuous* function of the *dihedral angle* γ at that point (Fig. 11). However, in actual full scale measurements [22-24, 27], it was found that for a properly constructed joint, the welding thickness is a *smooth* function of the dihedral angle with value exceeds the minimum requirement. Based on these requirements, the geometrical model for the weld path is created by first considering the pure geometrical intersection between the CHS members (i.e. the SOLID_0W model) which forms two intersection curves when a brace intersects a chord (Fig. 12). The welding profile is then formed by shifting these curves (either fill-out or cut-in) [23, 27] according to the thickness requirements (Figs. 13 and 14). In general, when a brace with thickness t_b intersects a chord, the final contact thickness of the welding obtained from the model can be expressed as

$$T_w = T_1 + T_2 - T_3 \geq T_{AWS} \quad (1)$$

In Eqn. (1), T_{AWS} is the minimum required thickness of the intersection. T_1 is the original contact thickness due to pure geometrical intersection (Fig. 12), T_2 and T_3 are the fill-out and the cut-in of the welding, respectively (Figs. 13 and 14). Note that depends on the value of the *inner dihedral angle* γ_i (Figs. 13 and 14) both the cut-in and the fill-out could locate at either sides of the welding. In this model the values of T_i , $i=1,2$ and 3 are related to t_b in the form

$$T_i = k_i(\gamma_i, \gamma_o)t_b \quad (2)$$

where $k_i(\gamma_i, \gamma_o)$, $i=1,2$ and 3 are three thickness factors that control the welding profile and are functions of the inner (γ_i) and outer (γ_o) dihedral angles. After careful examination of the welding profiles for many full scale and scaled joints and considering the requirements from the industrial standards, the following expressions for $k_i(\gamma_i, \gamma_o)$ are adopted [23]

$$k_1 = \begin{cases} \frac{1}{\sin \gamma_i}, & \gamma_0 \leq 3\pi/4 \\ a\gamma_i + b, & 3\pi/4 \leq \gamma_0 \leq \pi \end{cases} \quad (3a)$$

$$k_2 = \begin{cases} 0.3 \left[1 - \left(\frac{\gamma_i - \pi/6}{5\pi/6} \right)^2 \right], & \gamma_0 \leq 3\pi/4 \\ a\gamma_i + b - \frac{1}{\sin \gamma_i}, & 3\pi/4 \leq \gamma_0 \leq \pi \end{cases} \quad (3b)$$

$$k_3 = 0.25 \left[1 - \left(\frac{\gamma_i - \pi/6}{\pi/3} \right)^{0.4} \right] \quad (3c)$$

$$\text{where } a = \frac{\cos \gamma_i}{\sin^2 \gamma_i} \Big|_{\gamma_0=3\pi/4} - \frac{36}{5\pi}, \quad b = 0.153 + \frac{1}{\sin \gamma_i} \Big|_{\gamma_0=3\pi/4} - a\gamma_i \Big|_{\gamma_0=3\pi/4} \quad (3d)$$

By carrying out a number of numerical experiments for joints covering a wide range of geometrical parameters, it is found that Eqns. 1 to 3 give a smooth welding profile that satisfies the minimum welding standard. In addition, validations against many actual joints confirmed that the resulted thickness is *less* than the thickness achieved in actual welding so that conservative stress concentration modeling results could be obtained.

For a partially overlapped joint, since the final model is constructed by following the fabrication sequence of an actual joint, the welding part between the through brace and the chord (Curve 1 in Fig. 15) shall be further intersected with the welding parts between the lap brace and the through brace (Curve 2, Fig. 15), and between the lap brace and the chord (Curve 3, Fig. 15). Furthermore, as shown in Fig. 16, at the junction between these curves, three intersection points are formed. Eventually, in the HYBR_1W model, only the welding parts are converted to solid elements which are then connected to the surface elements created. For the SOILD_1W and the SOLID_CR models, the solid element obtained from the welding model will be connected to other solid elements in the mesh.

4.3 The geometrical model for crack details (SOLID_CR)

For CHS tubular joints, experimental and numerical studies show that peak HSS always develops at the weld toe (i.e. along the outer weld path) of the welding (Fig. 17). Furthermore, when the joint is subjected to cyclic loading, surface crack is initiated at the peak HSS location. The surface crack will penetrate the affected section in the thickness direction and propagate along the

weld toe of the joint. After a sufficiently large numbers of cycles of loading (up to millions in some cases) are applied, the surface crack may eventually fail the section by fully penetrates it. If the failed section is opened, a smoothed *crack surface* which is formed by many cycles of incremental advancements of the *crack front* could be identified (Fig. 18). The final crack profile usually consists of a through thickness portion and a surface crack portion. Hence, in order to create a useful geometrical model for a cracked joints, it is mandatory to define mathematically the geometries of the crack surface and the crack front.

The geometrical model adopted here to describe the shape of the 3D crack surface is again based on measurements of surface cracks formed in full scale fatigue tests [22-24, 26, 28, 31]. It is found that the shape of the crack surface could be described by considering its cross-section normal to the outer welding path of the joint (Fig. 19). Careful measurements shown that the cross section of the crack surface at any point W_0 along the crack (Fig. 19a) can be represented as a curve W_0D' (Fig. 19b). As shown in Fig. 19b, the curved length of W_0D' and its projection on the plane W_0D are denoted as a' and a , respectively. It is found that the angle $\angle D'W_0D$, denoted as ω is a smooth *monotonically increasing* function of a' . Therefore, it could be represented in a general polynomial form as

$$\omega(a') = c_0 + c_1(a') + c_2(a')^2 + \dots \quad (4a)$$

where $c_i \geq 0$ are constants independent of a' .

Post failure inspections of different tubular joints [22-24, 27, 28, 31] shown that for joints involving rectangular hollow sections (RHSs) [24, 28], the linear term must be included to obtain good modeling results. Hence, for RHS tubular joints, the surface crack should be described by the following equation

$$\omega(a') = c_0 + c_1(a') \quad (4b)$$

In practice, the section is often considered to be failed when $a' = t$ where t is the thickness of the section. Hence, it is more convenient to express the values of the constants c_0 and c_1 in terms of ω_0 and ω_t , the values of ω when $a' = 0$ and $a' = t$, respectively. Thus, Eqn. 4b could be expressed as

$$\omega(a') = \omega_0 + (\omega_t - \omega_0)(a')/t \quad (4c)$$

However, for tubular joints formed by CHS, it is found that the change in the angle ω is very small [22, 23, 27] through the thickness of the section. Hence, for CHS tubular joint modeling, W_0D' could be treated as a straight line. That is, Eqn. 4c could be further simplified to the form

$$\omega = \omega_0 \quad (4d)$$

In the current geometrical model, the general form of Eqn. 4c is adopted. During the implementation of the mesh generation scheme, Eqn. 4d is recovered by setting $\omega_i = \omega_0$ in all crack surface geometry calculations.

One remark regarding Eqn. 4 is that the values of ω_0 and ω_i for different joint types could be obtained by experimental studies and careful measurement of the crack surface shapes after the cracked sections are opened [22-24, 27, 28]. Furthermore, the geometrical model given by Eqn. 4c could also allow the modeler to carry out parametric study on the sensitivity of the values of the stress intensity factors (SIF) to the shape of the crack surface. It turns out that for CHS joints, the values of SIF are not sensitive to the value of ω_0 when it falls within a certain range.

Actual crack front monitoring using the alternating current potential drop (ACPD) technique [22-24] shown that the crack front can be represented as a bi-elliptical curve on the projected plane of the crack surface. In practice, the locations of the two tips and the deepest point of the surface crack relative to a fixed reference point on the weld toe could be obtained by insitu measurements (Figs. 20a and 20b) so that the lengths l_{cr0} , l_{cr1} and l_{cr2} in Fig. 20a are known. Hence, during numerical modeling, the shape of the crack front can be defined as a unsymmetrical bi-elliptical curve on the projected $s'-t'$ plane (Fig. 20a).

$$\text{For } l_{cr1} \leq s' \leq l_{cr0} : \quad \frac{(s' + l_{cr0})^2}{(l_{cr1} + l_{cr0})^2} + \frac{(t')^2}{(a')^2} = 1 \quad (5a)$$

$$\text{For } l_{cr0} \leq s' \leq l_{cr2} : \quad \frac{(s' + l_{cr0})^2}{(l_{cr2} + l_{cr0})^2} + \frac{(t')^2}{(a')^2} = 1 \quad (5b)$$

As a summary, the exact location of the 3D crack front could be obtained by the following steps:

- (i) From the SOLID_1W model, the position of any point W_0 on the weld toe could be located.
- (ii) From Eqn. 4, the shape of the crack surface is defined implicitly for any value of a' .
- (iii) From in-situ measurements (or assumed values during a parametric study), the value of l_{cr0} , l_{cr1} and l_{cr2} and the depth of the crack could be determined.
- (iv) Finally, the crack front equation is defined by Eqn. 5.

5. *Adaptive mesh generations*

Based on the hierarchical geometrical models described in the previous section, an associated family of automatic mesh generation schemes is implemented to discretize the geometrical models into FE meshes for numerical analysis. Since in the geometrical models, different geometrical objects including solids (hollow cylindrical sections and welding parts), surfaces (cylindrical surfaces and 3D crack surface) and curves (welding profile and 3D crack front) are present, it is impossible to use a single mesh generation technique to discretize the target geometrical models into FE meshes. As a result, the family of mesh generation schemes developed here is constructed by combining the following four mesh generation techniques which are specially designed for the discretization of different geometrical objects.

5.1 *Mesh generation technique used*

The surface mesh generation (SMG) technique

As the surface model of a tubular joint is idealized as intersections between different cylindrical surfaces, the joint is naturally divided into different regions (Fig. 21a). A general purpose SMG technique developed by Lee [8, 21] is adopted in this study for the discretization of the surfaces described by the surface model SURF_0W. By default, this general purpose *unstructured* mesh generation technique considers the target domain as a surface patch formed by the union of a number of intersecting surfaces [21]. Furthermore, individual surface of the surface patch could be described in analytical form (in this case, cylindrical surface) or defined implicitly through a surface approximation algorithm. During mesh generation, the boundary of the surface patch will be discretized first in order to maintain the compatibility between different surface regions. Advancing front surface triangulation is then employed to fill the interior of individual region with triangular elements. The mesh generation technique developed is fully adaptive in the sense that it allows user to specify different mesh densities at different locations of the surface [10]. In the case of tubular joint, since in both static strength and fatigue analyses attentions are focused on the accurate prediction of stress concentration near the intersection curves of the joint, small elements are specified and created along all there as shown in Fig. 21b. (A common practice is to use at least 32 segments along the intersection curves between two CHS sections.) It should be mentioned that as the surface model SURF_0W forms the base for all other more complex models (Fig. 5), the SMG technique is the core module in all mesh generation schemes.

The thin-walled structures mesh generation (TWSMG) technique

As the SOLID_0W model is a thin-walled model in which the thickness of the CHS is one order less than its length and diameter, a special solid mesh generation technique is needed to handle this special geometrical characteristic. In this study, the TWSMG technique developed by Lee and Xu [32] is adopted. The basic principle of this technique is shown in Fig. 22. Firstly, the target thin-walled structure is covered by a surface mesh using the SMG technique described in the last section. Secondly, extrusion and refinement are carried out to convert the surface elements into solid elements. Adaptive refinements in both the surface and the thickness directions are carried out independently by combining the SMG technique with the extrusion and refinement technique. This TWSMG technique is specially designed to work with the SMG technique and special algorithms were developed [32] to handle thin-walled structures formed by multiple intersections (Fig. 23). Furthermore, during the SCF and SIF computations of tubular joints, multiple layers of elements are required near the intersection curves. Numerical experiments found that typically four layers of solid elements (Fig. 23b) are needed to capture the stress valuations along the thickness direction while only two layers of elements are sufficient for other parts of the joint. Note that in this TWSMG technique, in order to obtain a smooth transition between different element layers in the thickness direction, four types of solid elements, namely, tetrahedron, prism, pyramid and hexahedron are generated (Fig. 23b). Furthermore, since the solid mesh is formed by extrusion of a surface triangular mesh, most of the solid elements in the mesh are prism elements while hexahedral elements are generation along the intersection between different CHS members. Pyramid and tetrahedral elements are mainly used as transition elements.

The welding parts mesh generation (WPMG) technique

In order to minimize the effect of welding thickness uncertainty which could never be eliminated in actual fabrication of a tubular joint [33], all the design guides [20, 29, 30] for fatigue assessments require analysts to obtain the HSS and the SCF values by using the extrapolation method in which stresses (or strain in experimental studies) at some pre-defined distances from the weld toe are extracted (or measured) and extrapolated for the calculations of the HSS and the SCF values. In order to simplify such extrapolation procedure, a structural mesh for the welding wedge is often preferred. Hence, a special *structured* mesh generation technique is developed to

extract the element size and connectivity information from the corresponding surface mesh (Fig. 21) or solid mesh (Fig. 23). It will then generate compatible size solid elements (prisms and tetrahedrons) to form the welding wedge and the junction point (Fig. 24). The resulted mesh is then combined with the original solid mesh or the surface mesh for the discretization of the SOLID_1W model (Fig. 25) or the HYD_1W model (Fig. 26), respectively. As element connectivity is ensured by the SMG and the TWSMG techniques, the main advantage of this approach is that solid mesh corresponding to the welding wedge could be added with a minimum amount of addition computational cost and mathematical descriptions.

The crack surface and front mesh generation (CSFMG) technique

From the experience obtained in previous numerical modelings [27, 28, 31], in order to obtain an accurate estimation of the SIF along the crack mouth, a carefully designed and locally refined mesh configuration is needed there. From the previous study carried by Lee and Lo [34], it is found that when extracting SIF values from the FE solution by employing the quarter-point displacement technique or the J-integral technique, the local geometry of the element configurations around the crack tip is the most critical factor affecting the accuracy of the extraction results. In particular, when collapsed quarter point elements (CQPE) [35] are placed around the crack mouth for SIF extraction, the accuracy of the extracted SIF value is sensitive to the size of the CQPE and its internal angle at the crack tip [34]. As SIF estimation is the most critical step during the residual fatigue prediction of a cracked joint, in order to ensure the accuracy of the extracted SIF value, a special CSFMG technique is designed to generate locally refined mesh configuration around the crack mouth by using the following steps.

- (1) Based on the surface crack location (position of the reference point O and the deepest point in Fig. 20) and its length (values of l_{cr0} , l_{cr1} and l_{cr2} in Fig. 20) which are specified by the analyst, the block of elements (Fig. 27) that was cut across by the crack is identified. This block of elements and their neighborhood elements are extracted from the solid mesh (Fig. 28a).
- (2) Based on the connectivity information of the extracted block, a connection block (Fig. 28b) which connects the crack surface block (Fig. 28c) with the extracted mesh is created.
- (3) Based on the crack front model, a tube of concentric elements (Fig. 28c) is locally created to enclose the crack front. In order to investigate the variation of SIF along the crack front, this

tube should contain at least 32 segments. (The current implementation would be able to generate up to 64 segments adaptively depends on the crack length). This tube of elements is then connected to the crack surface block by a set of connection elements.

- (4) From the geometrical model of the crack front, CQPEs are inserted along the crack mouth. A “spider-web” configuration of elements (Fig. 29a) is constructed along the crack mouth to facilitate the extraction of SIF after the FE analysis (Fig. 29b). As shown in the zoom in view of Fig. 29b, eight CQPEs are placed around the crack mouth and they are enclosed by three rings of hexahedral elements which are then connected to the crack surface block. Such local pattern of element configuration has been tested and validated against many test results [23, 27, 28, 31] and was found to give reliable SIF estimation for many joint types.

The above CSFMG technique could be applied at any location along any welding path of the joint so long as the user defines the location, the length and the depth of the crack. Furthermore, multiple cracks are allowed so long as they are separate with a minimum distance equal to the thickness of the CHS section.

A final remark on the above mesh generation procedure for crack surface and front is that while only a relatively small volume of the tubular joint is enclosing the surface crack, a relatively large numbers of elements are generated to capture the details of the crack surface and the crack front. In fact, for the typical example shown in Figs. 27 to 29, it can be seen that the extracted mesh, which accounts for more than 97% of the joint volume, is discretized into 15000 nodes and 19000 elements. However, the connection block (Fig. 28b), the crack surface block (Fig. 28c) and the crack mouth block (Fig. 29), which together only account for less than 3% of the joint volume, are eventually discretized into 2600 nodes and 3000 elements.

5.2 Mesh generation schemes developed and application range

As a summary of the mesh generation procedure, by combining the above mesh generation techniques properly, the following of mesh schemes are implemented:

- (1) A surface mesh generation scheme for surface meshes (shell elements) generation for ultimate strength analysis (The SMG technique only).
- (2) A solid mesh generation scheme without welding details for ultimate static strength analysis and stress concentration analysis along the intersection when the welding quality is unknown

or when it does not comply with the minimum welding standard (Combination of the SMG technique and the TWSMG technique).

- (3) A solid mesh generation scheme with welding details for accurate stress concentration analyses for joints with known welding details which comply with the welding standard (Combination of the SMG, the TWSMG and the WPMG techniques).
- (4) A hybrid surface and solid mesh generation scheme for quick estimation of stress concentration and ultimate strength with the effects of welding included (Combination of the SMG technique and the WPMG technique).
- (5) A solid mesh generation scheme with welding and crack details included for fatigue, fracture and residual life assessments of damaged joints (Combination of all four techniques)

By combining the different mesh generation techniques properly, this family of mesh generators is able to generate well shape FE meshes for a wide range of geometrical parameters which cover almost all practical joint configuration used in onshore and offshore applications. Table 1 shows the valid range of the geometrical parameters that the current implementation can handle.

Geometrical parameters (Figure 3)	Valid range of application
Intersecting angle of the joint	$30^\circ \leq \theta, \theta_1, \theta_2 \leq 90^\circ$
Ratio between brace radius and chord radius	$0.1 \leq \beta_1, \beta_2 \leq 1.0$
Ratio between brace and chord thickness	$0.03 \leq l/\gamma \leq 0.25$
Percentage of overlap for partially overlapped joint	$20\% \leq \xi \leq 80\%$
Crack surface angle (Fig. 19)	$-20^\circ \leq \omega \leq 20^\circ$
Location of crack	Anywhere of along the weld toe at either the brace side or the chord side
Number of crack in one joint	Unlimited so long as they are separated by a distance greater than the section thickness
Minimum depth of crack (Fig. 19)	$0.1 \leq a/t \leq 0.9, t = \text{thickness of section}$

6. Mesh generation and modeling examples

6.1 Mesh generation examples

In order to demonstrate the usage of the suggested geometrical models and the family of mesh generation schemes developed, a mesh generation example for a partially overlapped CHS N-joint is given. A N-joint is in fact a particular case of a K joint when one of the brace intersects at

right angle with the chord. In Fig. 30, the surface mesh of a partially overlapped CHS N-joint with the through brace intersects perpendicularly with the chord is shown. The diameters of the chord, the through brace and the lap brace are equal to 300, 250 and 180 units respectively. The overlapping percent of the joint is 25% and the thickness of the chord and the braces are equal to 25 and 19 units, respectively. Note that from the zoom in view near the joint intersection, layers of structured elements corresponding to the welding thickness are added to the surface mesh so that it could be converted to the a hybrid mesh by directly adding the solid elements generated based on the welding model. The corresponding solid mesh with welding details is shown in Fig. 31. These two figures also show that the solid mesh contains more nodes (13778 vs 2974) and elements (18120 vs 2974) than the surface mesh. Thus, solid meshes are only used when details stress concentration effect around the joint is required to be studied.

6.2 *Modelling examples*

The modeled joint

In order to demonstrate the reliability of the geometrical modeling and the mesh generation schemes presented, a full scale partially overlapped CHS K joint was constructed (Fig. 32) and the test results were compared with the numerical results. The joint was tested under both static and cyclic loadings. For static loading, basic loading modes of axial loading (AX), in-plane bending (IPB), out-of-plane bending (OPB) and their combinations were applied. Extensive instrumentations were carried out to measure the strains developed so that the actual variations of the SCF along the joint intersection curves were known. After the static test was completed, the joint was subjected to cyclic loading (AX=200kN, IPB=45kN, 0.2Hz) for fatigue test until full thickness crack was developed. The crack development process was monitored continuously during the fatigue test so that the SIF value at the deepest point of the crack could be computed from the advancement speed of the crack [27, 28].

HSS and SCF predictions

Fig. 33 shows the solid FE mesh (with welding details) used for the numerical analysis. Despite that from the geometrical and loading conditions shown in Fig. 32, one can expect that high stress concentration should occur at the heel of the intersection curve between the through brace and the chord. It is not known in advance whether the peak HSS should occur on the *brace size* or the

chord side of the welding. Fig. 34 shows the HSS distribution around the heel along the intersection curve between the through brace and the chord. From Fig. 34, it can be seen that the FE model predicted that the peak HSS should occur along the brace side of the welding. This fact was subsequently confirmed by actual strain measurements. Note that as the peak HSS location determines where the fatigue crack shall be formed, it is of much practical important that the FE model should correctly predict its exact location. Since for most tubular joints the chord is normally thicker than the brace, a fatigue crack locates on the brace side implies a large reduction of fatigue life when comparing with the case of a fatigue crack on the chord side.

Fig. 35 compares the SCF variations obtained from experimental studies with those obtained from numerical modeling on the through brace. It can be seen that the suggested geometrical models gave good predictions of the SCF values. Note that the most accurate SCF prediction was achieved by using the exact welding thickness which was constructed based on the *measured* weld thickness obtained from the actual joint. Since detailed welding profile is not available *before* the joint is constructed, for design purpose, the suggested models (SOLID_0W and SOLID_1W) should be used. Fig. 35 also confirms the importance of including the welding profile in SCF analysis. A model without welding profile tends to further overestimate the peak SCF and eventually gives a too conservative prediction of fatigue life. Note that in most design guide, a 10% increase in SCF or peak HSS shall imply a 30% reduction in the estimated fatigue life.

SIF prediction

Fig. 36 compares the values of SIF at the deepest point of the surface crack obtained from the experimental study and the numerical modeling. The different curves in the figure are corresponding to different values of the angle ω (Fig. 19) used in the construction of the crack surface geometry. It can be seen that the values of SIF are not sensitive to the values of ω used. In general, all the predictions based on different values of ω underestimated the values of SIF when the crack is shallow while conservative predictions are obtained during the later (more critical) stage of the crack development. Finally, from Fig. 36, one could conclude that a value of $\omega=0^\circ$ gave the most accurate SIF prediction and therefore this value should be used whenever no information of the actual value of ω is available.

7. *Conclusions and Future works*

In this paper, a systematic modeling procedure for the analysis of thin-walled tubular joints is presented. It is shown that two essential ingredients are needed to construct accurate numerical models for different levels of engineering analyses. The first ingredient is a set of hierarchical geometrical models for that gives consistent description on the geometrical details of the joint. The second ingredient is a family of associated mesh generation schemes which could discretize the geometrical models into appropriate types of FE meshes. A numerical example on the fatigue performance of a partially overlapped CHS K-joint showed that the modeling procedure developed could lead to reliable and conservative prediction on the actual responses and behaviour of the joint.

While the current approach could able to give good modeling results for most of the joint types and crack locations, it still has some limitations especially for the modeling of long and shallow crack. Furthermore, a large number of nodes and elements are required to capture the crack surface and crack mouth details in the model. Hence, a possible direction of future research is to combine the suggested crack surface and front geometrical model with the newly developed extended finite element method [36] to reduce the computational cost needed and to extend the range of application.

References

1. Lo, S. H., A new mesh generation scheme for arbitrary planar domains. *Int. J. Numer. Meth. Engrg.*, **21**, pp.1403-1426, 1985.
2. Lo, S. H., Automatic mesh generation and adaptation by using contours. *Int. J. Numer. Meth. Engrg.*, **31**, pp. 689-707, 1991.
3. Zhu, J. Z., Zienkiewicz, O. C., Hinton, E. and Wu, J., A new approach to the development of automatic quadrilateral mesh generation. *Int. J. Numer. Meth. Engrg.*, **32**, pp. 849-66, 1991.
4. Lee, C. K. and Hobbs, R. E., Automatic adaptive finite element mesh generation over arbitrary two-dimensional domain using advancing front technique. *Computers and Structures*, **71**, pp. 9-34, 1999.
5. Lohner, R., Regridding surface triangulations. *J. Comput. Physics*, **126**, pp. 1-10, 1996.
6. Lau, T. S., Lo, S. H. and Lee, C. K., Generation of quadrilateral mesh over analytical curved surfaces. *Fin. Ele. Anal. Design*, **27**, pp. 251-72, 1997.

7. Frey, P. J. and Borouchaki, H., Surface meshing using a geometric error estimate. *Int. J. Numer. Meth. Engrg.*, **58** pp. 227–245, 2003.
8. Lee, C. K., Automatic metric 3D surface mesh generation using subdivision surface geometrical model. Part 2: mesh generation algorithm and examples. *Int. J. Numer. Meth. Engrg.*, **56**, pp. 1615–1646, 2003.
9. Frey, P. J., Borouchaki, H. and George, P. L., 3D Delaunay mesh generation coupled with an advancing-front approach. *Comput. Methods Appl. Mech. Engrg.*, **157**, pp. 115-31, 1998.
10. Owen, S. J. and Saigal, S., H-Morph: an indirect approach to advancing front hex meshing. *Int. J. Numer. Meth. Engrg.*, **49**, pp. 289–312, 2000.
11. Yamakawa, S. and Shimada, K., Anisotropic tetrahedral meshing via bubble packing and advancing front. *Int. J. Numer. Meth. Engrg.*, **57**, pp. 1923–1942, 2003.
12. Ito, Y., Shum, P. C., Shih, A. M., Soni B. K. and Nakahashi, K., Robust generation of high-quality unstructured meshes on realistic biomedical geometry. *Int. J. Numer. Meth. Engrg.*, **65**, pp. 943–973, 2006.
13. Lee, C. K. and Hobbs, R. E., Automatic adaptive refinement for plate bending problems using Reissner-Mindlin plate bending elements. *Int. J. Numer. Methods Engrg.*, **41**, pp. 1–63, 1998.
14. Okstad, K. M. and Mathisen. K. M., Towards automatic adaptive geometrically non-linear shell analysis. Part I: Implementation of an h -adaptive mesh refinement procedure. *Int. J. Numer. Methods Engrg.*, **37**, pp. 2657-2678, 1994.
15. Lee, C. K. and Hobbs, R. E., Automatic adaptive refinement for shell analysis using nine-node assumed strain element. *Int. J. Numer. Methods Engrg.*, **40**, pp. 3601–3638, 1997.
16. Lohner, R. and Cebal J., Generation of non-isotropic unstructured grids via directional enrichment. *Int. J. Numer. Methods Engrg.*, **49**, pp. 219–232, 2000.
17. Larwood, B. G., Weatherill, N. P., Hassan, O. and Morgan K. Domain decomposition approach for parallel unstructured mesh generation. *Int. J. Numer. Methods Engrg.*, **58**, pp. 177–188, 2003.
18. Lee, C. K. and Lo, S. H., Automatic adaptive 3-d finite element refinement using different-order tetrahedral elements. *Int. J. Numer. Methods Engrg.*, **40**, pp. 2195–2226, 1997.
19. Lee, C. K. and Lo, S. H., A full finite element analysis using adaptive refinement and PCG solver with back interpolation. *Comput. Methods Appl. Mech. Engrg.*, **70**, pp 39-64, 1999
20. Zhao, X.L., Herion, S., Packer, J.A., Puthli, R., Sedlacek, G., Wardenier, J., Weynand, K., van Wingerde, A. and Yeomans, N., 2000, Design Guide for Circular and Rectangular hollow Section Joints under Fatigue Loading. CIDECT, TUV Germany.

21. Lee, C. K., Automatic metric 3D surface mesh generation using subdivision surface geometrical model. Part 1: Construction of underlying geometrical model. *Int. J. Numer. Meth. Engrg.*, **56**, pp 1593 – 1614, 2003.
22. Chiew, S.P., Lie, S.T., Lee, C.K., and Huang, Z.W., 2004, Fatigue performance of cracked tubular T joints under combined loads. I: Experimental. *Journal of Structural Engineering*, **130**, pp. 562-571, 2004.
23. Nguyen T. B. N., 2008, Model and Mesh Generation of Partially Overlapped Circular Hollow Section K-joints for Fatigue Studies. PhD Thesis, School of Civil and Environmental Engineering, Nanyang Technological University.
24. Chiew S. P., Lee C. K., Lie S. T. and Ji H. L., Fatigue behaviors of Square-to-Square Hollow Section T-joint with corner crack. I: Experimental studies. *Engineering Fracture Mechanics*, **74**, pp. 703–720, 2007.
25. Cao, J.J., Yang, G.J., and Packer, J.A., and Burdekin, F.M., Crack modelling in FE analysis of circular tubular joints. *Engineering Fracture Mechanics*, **61**, pp. 537-553, 1998.
26. Lie, S. T., Lee, C. K. and Wong, S. M., Modelling and mesh generation of weld profile in tubular Y-joint. *Journal of Constructional Steel Research*, **57**, pp. 547-567, 2001.
27. Lie, S.T., Chiew, S.P., Lee, C.K. and Huang, Z.W., Fatigue performance of cracked tubular T joints under combined loads. II: Numerical. *Journal of Structural Engineering*, **130**, pp. 572-581, 2004.
28. Lee C. K., Chiew S. P., Lie S. T. and Ji H. L., Fatigue behaviors of Square-to-Square Hollow Section T-joint with corner crack. II: Numerical Modelling. *Engineering Fracture Mechanics*, **74**, pp. 721–738, 2007.
29. American Petroleum Institute, 2000, Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms, API RP2A.
30. American Welding Society, 1996, ANSI/AWS D1.1-96 Structural Welding Code-Steel, Miami, USA.
31. Lee C. K., Lie S. T., Chiew S. P. and Shao Yongbo., Numerical models verification of cracked tubular T, Y and K-joints under combined loads. *Engineering Fracture Mechanics*, **72**, pp. 983-1009, 2005.
32. Lee C. K. and Xu QiangXun, A new automatic adaptive 3D solid mesh generation scheme for thin-walled structures. *Int. J. Numer. Meth. Engrg.*, **62**, pp. 1519-1558, 2005.
33. Lee C. K., Lie, S. T. and Chiew S. P. Finite element modeling of tubular joint structures for fatigue assessment. *Proceedings of the international symposium on fatigue and fracture of steel structures*, 4 December 2007, Singapore, pp. 97-117, 2008.

34. Lee C. K. and Lo S. H. (1992) Solving crack problems by an adaptive refinement procedure. *Engineering Fracture Mechanics*, **43**,147-163.
35. Barsoum R. S. On the use of isoparametric finite elements in linear fracture mechanics. *Int. J. Numer. Meth. Engrg.*, **10**, pp. 25-37, 1976.
36. Areias P. M. A and Belytschko T. Analysis of three-dimensional crack initiation and propagation using the extended finite element method. *Int. J. Numer. Meth. Engrg.*, **63**, pp. 760-788, 2005.

Figure 1. A CHS tubular joint and stress concentration at the connection

Figure 2. Some commonly used configurations of CHS tubular joints

Figure 3. Configuration and notations for a partially overlapped CHS K-joint

Figure 4. The overall model and mesh generation sequence

Figure 5. Relationships among the hierarchical geometrical models

(c) Mapping of elliptical curve to circle

Figure 6. Double mapping procedure for solving the tube-to-tube intersection problem

Figure 7. Coordinate systems used in the construction of geometrical model for a partially overlapped CHS K-joint

Figure 8. The three intersection curves and one junction point generated in a partially overlapped CHS K-joint

Figure 9. Fabrication sequence of a partially overlapped CHS K joint.

Figure 10. The welding process and geometrical parameters

Figure 11. Definition of the dihedral angle γ along the joint intersection

Figure 12. Intersection curves of a brace on the chord and the weld paths

Figure 13. Cross section of the weld model at Section 1 – 1 of Fig. 12 for $30^\circ \leq \gamma_i < 90^\circ$

Figure 16. Intersection details of the welding profile near the junction point

Figure 17. Surface cracks along the weld toe of the welding path on a tubular joint

Figure 18. A typical through thickness crack generated after fatigue failure of the CHS

(a) Plan view of a surface crack

(b) Cross-sectional view of a surface crack

Figure 19. A surface crack developed on a CHS tubular joint

(a) Parametric space used for the mapping

(b) 3D crack front

Figure 20. Mapping for the elliptical crack front

(a) Overall view of the surface mesh

(b) Zoom in view near the joint intersection

Figure 21. A typical mesh for the surface model SURF_0W
(NN=2372, NT=3824)

NN = Total numbers of nodes	NTet = Total numbers of tetrahedrons
NE = Total numbers of elements	NPyr = Total numbers of pyramids
NT = Total numbers of triangles	NPrism = Total numbers of prisms
NQ = Total numbers of quadrilaterals	NHex = Total numbers of hexahedrons

Legends for Figures 21, 23-31

Figure 22 Formation of solid mesh for thin-walled structures by extrusion and refinement

(a) Overall view of the inside part

(b) Zoom in view of the inside part

Figure 23. A typical mesh for the solid model SOILD_0W
(Full mesh: NN=14762, NE= 17394, NTet=304, NPyr=360, NPrism=12024, NHex=4706)

Figure 24. A typical solid mesh generated for the welding part and the junction point
(Welding part: NN=1020, NPrism=882, Junction part: NN=80, Ntet=108)

(a) View of the inside part

(b) Zoom in view of the inside part

Figure 25. A typical mesh for the solid model SOILD_1W
 (Full mesh: NN=15074, NE=19984, Ntet=476, NPyr=140, NPrism=14524, NHex=4844)

(a) Zoom in view of the outside part near the joint intersection

(b) Zoom in view of the inside part near the joint intersection

Figure 26. A typical mesh for the hybrid model HYDR_1W
(Full mesh: NN=3754, NE=5712, NT=3824, NPrism=1728, NTet=160)

Figure 27. A typical mesh for the model SOLID_CR with the crack block extracted.
(NN=15019, NE=19731, NTet=476, NPyr=140, NPrism=14368, NHex=4748)

(a) Zoom in view of the extracted part

(b) Connection block to the crack surface block
(NN=473, NE=326, NTet=128, NPrism=80, NHex=118)

Spider-web refined mesh
for the crack mouth block
(Zoomed in view given in
Fig. 29)

(c) Semi-transparent view of the crack surface and
the crack front blocks (NN=2210, NE=2768,
NTet=112, NPyr=176, NPrism=1456, NHex=1024)

Figure 28. The connection and crack surface blocks

Figure 29. The crack mouth block and element arrangement near the crack tip
(NN=1820, NE=1536, NPrism=512, NHex=1024)

Figure 30. Mesh generation Example 1: Surface mesh for a partially overlapped N joint (NN=2974 NE=5040, NT=114, NQ=4926)

Figure 31. Mesh generation example 2: Solid mesh with welding details for a partially overlapped N joint
 NN=13778, NE=18120, NTet=492, NPyr=162, NPrism=13176, NHex=4290

Figure 32. Dimensions of the full scale joint tested
(All dimensions in mm, modulus of elasticity: Chord=207.5GPa, Braces = 201.9GPa)

Figure 33. FE mesh used in the analysis

Figure 34. Hot spot stress around the through brace (100kN AX + 45KN IPB)

Figure 35: Comparison of SCF on the through brace under combined loading (100kN AX + 45KN IPB) obtained from different models against the measured results

Figure 36: Comparison of SIF at deepest point obtained from different models against the measured results.