<table>
<thead>
<tr>
<th>Title</th>
<th>Model and mesh generation of cracked tubular Y-joints</th>
</tr>
</thead>
<tbody>
<tr>
<td>Author(s)</td>
<td>Lie, Seng Tjhen; Lee, Chi King; Wong, S. M.</td>
</tr>
<tr>
<td>Date</td>
<td>2002</td>
</tr>
<tr>
<td>URL</td>
<td><a href="http://hdl.handle.net/10220/19240">http://hdl.handle.net/10220/19240</a></td>
</tr>
<tr>
<td>Rights</td>
<td>© 2002 Elsevier. This is the author created version of a work that has been peer reviewed and accepted for publication by Engineering Fracture Mechanics, 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: [Article DOI: <a href="http://dx.doi.org/10.1016/S0013-7944(02)00036-X">http://dx.doi.org/10.1016/S0013-7944(02)00036-X</a>].</td>
</tr>
</tbody>
</table>
Model and Mesh Generation of Cracked Tubular Y-Joints

S. T. Lie, *C. K. Lee and S. M. Wong

School of Civil and Structural Engineering
Nanyang Technological University
Nanyang Avenue
Singapore 639798

*E-mail: ccklee@ntu.edu.sg, Fax: 65-7921650

Pre-printed version of the paper appeared in the journal:
Engineering Fracture Mechanics, Vol.70, No. 2, pp161-184

Abstract
In this study, the methods for constructing accurate and consistent geometrical and finite element models for general cracked tubular Y-joints are described. Firstly, geometrical analysis of welded tubular joint is given and it is then extended to the modelling of general cracked Y-joints. The concept of crack surface and a simple mapping approach are suggested to model either through-thickness or surface cracks which can be of any length and located at any position along the brace-chord intersection. Secondly, the geometrical model developed will be used in the generation of consistent finite element meshes. The basic concepts used for the design and generation of 3D finite element meshes will be described. This will include the meshing procedures for discretization of tubular joints with through-thickness and surface cracks which are frequently regarded as one of the most difficult steps in the construction of tubular joint models. Finally, some mesh generation examples for uncracked and cracked Y-joints will be presented to demonstrate the use of the purposed geometrical model and mesh generation scheme developed.

KEYWORDS: Tubular Y-Joint, Geometrical Modelling, Welding Details, Through-Thickness Crack, Surface Crack, Finite Element Mesh Generation
1 Introduction

In offshore engineering, fixed steel/jacket structures formed by tubular members are widely used. For this type of structure, the tubular joints is one of the fundamental components since they impart the total structural system and the rigidity required. However, the configuration of tubular joint is complicated and high stress concentration presents the problem of cumulative fatigue failure due to large cyclic loading under various severe conditions. Consequently, much of the research effort on offshore engineering has been directed towards the estimation of stress concentration factors (SCF) of welded tubular joints and the stress intensity factor (SIF) of cracked tubular joints. Towards this end, the geometrical model used to describe the size of the welding and the crack details is one of the most important factors that will affect the accuracy of the analysis results. In fact, a consistent model is important when carrying out parametric studies of the behaviour of the joint using finite element (FE) analysis so that it will allow one to compare the results obtained from different researchers.

When carrying out FE analysis for tubular joints, in order to reduce the computational cost needed, shell elements that represent the mid-surfaces of the tubular member walls had been widely used to model joints without cracks [1-3]. However, Herion et al. [4] had made a comparison of models from different types of elements and showed that the best results were obtained from the models using 3D quadratic solid elements for all tubes and welds. For the cracked tubular joints, shell elements incorporated with line-spring elements [5] have been used to model the crack. Though this approach could model the global geometry of the tubular members correctly, the geometry of cracks could not be modelled accurately. In fact, to explore the fracture behaviour of the joint, 3D quadratic solid elements should be used throughout the whole FE model [6].

For the automatic preparation of the finite element model, many commercial software packages are available to generate 3D meshes for tubular joints such as PATRAN [7], FEMGEN [8], ANSYS [9], ABAQUS/PRE [10], I-DEAS [11] and PRETUBE[12]. However, Cao et al. [6] and Lee [13] found that these commercial software packages are not simple to use for mesh generation of tubular joints with cracks. Bowness and Lee [14] developed a method to generate dense meshes for surface cracks and to extract the crack tip opening displacement (CTOD) from the analysis results. However, it is found that the mesh quality outside the crack region is rather poor with unnecessary fine elements with high aspect ratio (>1:50). In order to alleviate this problem, Cao et al. [6,15] suggested a new approach to
generate meshes for cracked or uncracked tubular joints. Cao et al.’s method is well-defined in geometry and high quality meshes could be generated for regions away from the crack front. However, the meshes generated around the crack front are still badly distorted especially near the two ends of the crack.

The main objective of this paper is to introduce systematic geometrical modelling and mesh generation procedures for general welded and cracked tubular Y-joints. The geometrical analysis of tubular Y-joints will be outlined in Section 2 while the modelling of welded joint will be discussed in Section 3. In Section 4, the concept of crack surface will be introduced and the modelling procedures for through-thickness and surface cracks will be described. In Section 5, emphasis is given to the concept of mesh generation and the design scheme of high quality meshes for uncracked and cracked tubular Y-joints. In Section 6, mesh generation examples will be given to demonstrate the consistency of both the geometrical model used and the mesh generation scheme developed. Finally, conclusions for the current study will be given and some recommendations for future research works will be suggested.

2 Geometrical analysis of tubular Y-joint

In the past, only a rather limited amount of research effort had been directed to the detailed study of the geometry of tubular joints. Most of the researchers [14,16,17] modelled tubular joints using commercial FE packages such as ABAQUS [10], ANSYS [9] and PMBSHELL [18] (with weld modelling) without using a well defined and consistent underlying geometrical model. In addition, there are some limitations in these packages for the 3D modelling of welds and cracks especially for surface cracks. Cao et al. [15] carried out a detailed geometrical analysis for the intersection of a tubular joint and formulae for mapping a circle to the chord/brace intersection were developed. The details of the method and formulae will be briefly explained. Further study will be carried out to extend the model for the modelling of welded tubular joints.

2.1 Definition and parametric equations for intersecting curve without weld

In order to generate a mesh of a circular tube, the mesh can be first generated in a flat plane and then mapped onto the tube [15] as shown in Fig. 1. If the circular tube is cut by a longitudinal line and opened out along the line 1-1, a point a on the tube will be mapped to the point a’ on the plane. From the cross section X-X shown in Fig. 1, the arc length y’ is equal to
the distance $1-a'$ on the plane and this defines the mapping relationship. The relationship between the circular surface and the plane (i.e. the X-Y-Z and the X'-Y'-Z' coordinate systems) can be defined as

$$R_0 \omega = Y'$$  \hspace{1cm} (1)

where $\omega$ is the local polar angle defined on the chord circle (Fig. 1).

By using this mapping approach, a space curve on the circular surface will be transformed to a planar curve on the plane as shown in Fig. 2. The chord and brace can be defined in the two coordinate systems as

For the chord:  \hspace{2cm} $X^2 + Y^2 = R^2$  \hspace{1cm} (2)

For the brace: \hspace{1cm} $x^2 + y^2 = r^2$  \hspace{1cm} (3)

The intersection of the chord and the brace, which is a 3D curve, is defined by Eqn. (2) and (3) and can be expressed in the X-Y-Z coordinate system as

$$\left[ (X-R) \cos \theta - Z \sin \theta \right]^2 + Y^2 = r^2$$  \hspace{1cm} (4)

As shown in Figs. 1 and 2, the outer surface of the chord can then be transformed to a flat plane and is defined on the Y'-Z' plane by

$$R^2 \sin^2 \left( \frac{Y'}{R} \right) + \left[ Z' \sin \theta + R \left( 1 - \cos \frac{Y'}{R} \right) \cos \theta \right]^2 = r^2$$  \hspace{1cm} (5)

Though Eqn. (5) can be used to define the intersection curve directly, it is more convenient to define the intersection using a circle on a planar system [15]. Therefore, a local planar u-v coordinate system is used to define the circle and it will then be mapped to fit the intersecting curve in the Y'-Z' plane. The Y'-Z' plane will then be further mapped or wrapped to form a tube. As a result, a 3D intersecting curve is formed in the X-Y-Z coordinate system after two mappings as illustrated in Fig. 3 (plan view). In Fig. 3, $\alpha$ is defined as the polar angle in the u-v coordinate system. It is the main parameter in defining the intersecting curve and subsequently many other important geometrical parameters of the joints such as the dihedral angle, the weld thickness and the crack surface. The angle $\phi$ is the corresponding angle defined in the Y-Z plane. From Fig. 3, a circle of radius $r$ can be expressed as

$$u^2 + v^2 = r^2$$  \hspace{1cm} (6)
\[ u = r \sin \alpha, \quad v = r \cos \alpha \]  

If Eqn. (5) is made equivalent to Eqn. (6), then after some simple calculations one can show that the intersection curve in the X-Y-Z coordinate system can be expressed as

\[
X = R \cos \left( \sin^{-1} \frac{u}{R} \right)
\]

\[
Y = R \sin \left( \sin^{-1} \frac{u}{R} \right) = u
\]

\[
Z = Z' = \left[ v - R \left( 1 - \cos \frac{Y'}{R} \right) \cos \theta \right] \frac{1}{\sin \theta}
\]

As for the angle \( \phi \), it is the corresponding polar angle defined in the Y-Z plane and can be expressed as

\[
\phi = \tan^{-1} \left( \frac{Y}{Z} \right)
\]

2.2 Computation of dihedral angle, \( \gamma \)

For tubular joints, the dihedral angle, \( \gamma \), is an important parameter in the design of welding details. According to the American Welding Society (AWS) [19] and the American Petroleum Institute (API) [20] standards, the welding method and thickness of the weld are determined by the dihedral angle along the joint. Fig. 4 shows the geometry and the definition of the dihedral angle for a tubular joint. In Fig. 4, point A is lying on the intersecting curve while the planes \( S_c \) and \( S_b \) are the two tangential planes touching the surfaces of the chord and the brace respectively. \( \vec{n}_1 \) and \( \vec{n}_2 \) are the normal vectors of the tangential planes \( S_c \) and \( S_b \) respectively. \( \Psi \) is the angle between \( \vec{n}_1 \) and \( \vec{n}_2 \) and is the supplement of the dihedral angle, \( \gamma \). The dihedral angle at point A is defined as the angle between the tangential planes \( S_c \) and \( S_b \) at A [21] and such that

\[
\gamma + \Psi = \pi
\]

From the property of the surface gradients, the normal vectors \( \vec{n}_1 \) and \( \vec{n}_2 \) can be computed easily. From Eqn. (10), the dihedral angle, \( \gamma \), at point A can be computed as

\[
\gamma = \pi - \cos^{-1} \left[ \frac{\vec{n}_1 \cdot \vec{n}_2}{\| \vec{n}_1 \| \| \vec{n}_2 \|} \right]
\]
3 Modelling of welded tubular joint

In the past, some researchers [22,23] thought that detailed weld modelling is not necessary in some cases such as the analyses of T- and K- joints. However, recent research works show that weld modelling is necessary for the accurate prediction of joint strength and in SCF and SIF analyses [13]. In this section, the geometry of the welds will be studied in detail. Furthermore, a consistent weld model which complies with some commonly used codes of practice (API and AWS) will be suggested.

3.1 Inner and outer intersecting curves

As illustrated in Fig. 5, $R_1$ and $R_2$ are the outer and inner radius of the chord while $r_3$ and $r_4$ are the outer and inner radius of the brace. The thickness of the chord and brace are denoted as $t_c$ and $t_b$ respectively. Due to the thickness of the brace member, two intersecting curves will be formed at the joint as shown in Fig. 6. They are named as the inner and the outer intersecting curves respectively. For a given value of polar angle $\alpha$, the inner and outer intersection curves can be formed by replacing $r$ and $R$ (Fig. 2) by $r_4$, $R_2$ and $r_3$, $R_1$ respectively. The point $A_i (X_{Ai}, Y_{Ai}, Z_{Ai})$ and the point $A_o (X_{Ao}, Y_{Ao}, Z_{Ao})$ are corresponding points on the inner and the outer intersecting curves respectively. Their polar angles on the Y-Z plane are denoted as $\phi_i$ and $\phi_o$ respectively (Fig. 6). It was found that the difference between $\phi_i$ and $\phi_o$ is minimal and is negligible except when $r_3/R_1=1.0$ [24]. Similarly, $\gamma_i$ and $\gamma_o$ are, respectively, the inner and outer dihedral angles at points $A_i$ and $A_o$. In addition, $\beta_i$ and $\beta_o$ are respectively the angles between the normal of the inner and outer intersecting curves to the Z-axis on Y-Z plane (Fig. 7). Detailed computations of $\beta_i$ and $\beta_o$ are given in reference [24]. Note that for given values of $\alpha$ and $r_4$, the position of the point $A_i (X_{Ai}, Y_{Ai}, Z_{Ai})$ and the angles $\phi_i$, $\beta_i$ and $\gamma_i$ can be computed by referring to the geometrical model described in the last section. Similarly, the position of the point $A_o (X_{Ao}, Y_{Ao}, Z_{Ao})$ and the angles $\phi_o$, $\beta_o$ and $\gamma_o$ can also be obtained from $\alpha$ and $r_3$.


According to these specifications, the theoretical weld thickness, $T_W$ is dependent on the dihedral angle, $\gamma_o$. In general, the minimum weld thickness according to the API specification, $T_{API}$, and the AWS specification, $T_{AWS}$, can be expressed in the forms
In Eqns. (12) and (13), $k_{\text{API}}=f(\gamma)$ and $k_{\text{AWS}}=g(\gamma)$ are the minimum API and AWS weld thickness factors respectively. $f(\gamma)$ and $g(\gamma)$ are two continuous (but not necessarily smooth) functions of $\gamma$. The minimum API and AWS weld thickness factors are summarised in Table 1. In practice, the joint should be fabricated in such a way that $T_W \geq T_{\text{API}}$ or $T_W \geq T_{\text{AWS}}$.

3.3 Original contact thickness
Consider a non-welded joint as shown in Fig. 8, the original contact thickness, $T_1$ is defined as the effective surface contact thickness at a particular section normal to the intersection at the joint. The study of the original profile of the intersection (without any welding) in Figs. 6 and 8 shows that $T_1$ varies along the joint and is a function of $\gamma_i$, $\gamma_o$ and the change of curvature of the chord surface. Since in most practical cases $R_1 >> t_b$, one can assume that $\gamma = \gamma_i \approx \gamma_o$. Hence, $T_1$ can be approximated as

$$T_1 = k_1 \times t_b, \quad k_1 = \frac{1}{\sin \gamma}$$  \hspace{1cm} (14)

3.4 Basic assumptions for weld modelling
In order to model the welded joint properly, the following assumptions will be made to satisfy both the adequacy of weld thickness and the smoothness of the weld path.
1. The thickness of brace member, $t_b$, is small compare to the outer radius of the chord, $R_1$.
2. The smallest intersecting angle between the brace and chord members is greater than or equal to 30°.
3. The weld would be extended with extra thickness when $\gamma$ is greater than 135° though it is not necessary ($T_W$ need not exceed 1.75$t_b$ by the API and the AWS specifications).
4. It is assumed that the material properties of weld are same as the tubular members and hence, the gap or root as defined in AWS and API is not important in the modelling.

3.5 Modelling of weld toe and weld roots
To model the weld toe (point $W_o$ in Fig. 6), a distance of $T_2$ will shift out from the point $A_o$ to the point $W_o$ (fill outside) as shown in Fig. 7. The extent of the filling, $T_2$, will depend on $\gamma_o$ at
point A₀ and it is suggested that T₂ will vary from a finite value when \( \gamma_0 = \theta = 30^\circ \) (smallest intersecting angle assumed) to zero when \( \gamma_0 \) increases to \( 180^\circ \). The outer modification is expressed as

\[
T_2 = k_2 \times t_b
\]  
(15)

\[
k_2 = \text{Fos}_{\text{outer}} \left[ 1 - \left( \frac{\gamma_0 - \theta_s}{180^\circ - \theta_s} \right)^m \right]
\]  
(16)

where \( T_2 \) is the modified outer thickness, \( k_2 \) is the modification factor of the outer intersection curve, \( \text{Fos}_{\text{outer}} \) is a scale factor, \( m \) is a constant and \( \theta_s \) is the smallest intersecting angle (30°). Both \( \text{Fos}_{\text{outer}} \) and \( m \) will be determined later by the combination of \( k_1 \) and the inner modification factor, \( k_3 \). The modified equations for the outer intersecting curve (weld toe) can be written as

\[
\begin{align*}
Z_{W_o} &= Z_{A_o} + T_2 \cos \beta_o \\
Y_{W_o} &= Y_{A_o} + T_2 \sin \beta_o \\
X_{W_o} &= \sqrt{R_1^2 - Y_{W_o}^2}
\end{align*}
\]  
(17)

The point \( A_o \) \((X_{A_o}, Y_{A_o}, Z_{A_o})\) is defined by substituting values of \( \alpha, R_1 \) and \( r_3 \) into Eqns. (7) and (8).

For the weld root, when \( \gamma_i \) is between 30° to 90°, a distance of \( T_3 \) will shift in (cut inside) from point \( A_i \) to point \( W_i \) as shown in Fig. 7. For \( \gamma_i = 90^\circ \), no alteration is required (i.e. \( T_3 = 0 \)). When \( \gamma_i \) is greater than 90°, point \( A_i \) will be shifted out (fill inside) by \( T_3 \) to the point \( W_i \). The inner modification can be expressed as

\[
T_3 = k_3 \times t_b
\]  
(18)

\[
k_3 = \text{Fos}_{\text{inner}} \left[ 1 - \left( \frac{\gamma_i - \theta_s}{90^\circ - \theta_s} \right)^n \right]
\]  
(19)

where \( T_3 \) is the modified inner thickness, \( k_3 \) is the modification factor of the inner intersection curve, \( \text{Fos}_{\text{inner}} \) is a scale factor and \( n \) is a constant. Both \( \text{Fos}_{\text{inner}} \) and \( n \) would be determined by the combination of \( k_1 \) and the outer modification factor, \( k_2 \) (Eqn. 16) The modified equations for the inner intersecting curve (weld root) can be expressed as
\[ Z_{W_i} = Z_{A_i} + T_3 \cos \beta_i \]
\[ Y_{W_i} = Y_{A_i} + T_3 \sin \beta_i \]
\[ X_{W_i} = \sqrt{R_i^2 - Y_{W_i}^2} \]  

(20)

Again, the point \( A_i (X_{A_i}, Y_{A_i}, Z_{A_i}) \) is defined by substituting values of \( \alpha, R_1 \) and \( r_i \) into Eqns. (7) and (8).

3.6 Total weld thickness, \( T_W \)

The total weld thickness, \( T_W \), is the combination of the original contact thickness, \( T_1 \), the modified outer thickness, \( T_2 \), and the modified inner thickness, \( T_3 \). That is,

\[ T_W = T_1 + T_2 - T_3 = T_W = k_{Tw} \cdot t_b, \quad k_{Tw} = (k_1 + k_2 - k_3) \]  

(21)

As mentioned in Section 3.2, \( T_W \) must satisfy the minimum requirement of API and AWS specifications for weld thickness. Note that while the value of \( k_1 \) is only dependent on \( \gamma \), the values of \( k_2 \) and \( k_3 \) will depend on the values of the modelling parameters \( F_{os_{outer}}, F_{os_{inner}}, m \) and \( n \). In fact, one can obtain different models for the welding of the joint by adjusting the values of these parameters. By trial and error approach, the suggested values for \( F_{os_{outer}}, F_{os_{inner}}, m \) and \( n \) that will lead to a total weld thickness that satisfies the AWS and API specifications are found to be equal to

\[ F_{os_{outer}} = 0.3, \quad F_{os_{inner}} = 0.25, \quad m = 2.0, \quad n = 0.4 \]

4 Geometrical modelling of cracked surface and crack front

In many previous research works, various methods had been developed to model cracks such as the plane plate model [25], the plane strain T-butt approximation [26], the 3D T-butt model [27] and the line spring FE shell model [5]. However, it has been discovered that there are shortcomings in these models which can give significant underestimation. This is mainly due to the lack of the 3D weld toe notch effects such as the stiffening of the area adjacent to the crack plane and the difficulties in generating adequate meshes [14,28]. Bowness and Lee [14] developed another method to model a cracked tubular joint which is generally better than those discussed above but the method can only model a T-joint with a crack at the saddle or the crown. Cao et al. [15] carried out a detailed geometrical analysis for tubular intersection and developed a cracked tubular joint model [6]. Comparatively, Cao’s crack model is well
defined geometrically and flexible as it could model cracks at any position around the intersection. Nevertheless, the welding details used were not well defined and the meshes generated have high aspect ratio elements around the cracks tips. In the following sections, the geometrical model for the welded joint will be extended for the modelling of surface and through-thickness cracks.

4.1 Definition and determination of crack surface

Detailed measurements of crack growth revealed that cracks propagate under the weld toe (i.e. \( \text{W}_o \) in Fig. 9) as they grow through the chord wall [29]. Furthermore, if cracks have started from the surface of the chord they will propagate through the chord thickness such that they are perpendicular to the chord wall, as the energy required for such propagation is minimal. Further observation shows that the crack is actually propagating on a 3D surface formed inside the thickness of the chord. This surface is known as the crack surface, where the crack front lies on. As shown in Fig. 9, the crack surface is formed by joining a series of straight lines \( \text{W}_o\text{D} \) along the weld path. Furthermore, the lines \( \text{W}_o\text{D} \) will pass through the Z-axis and the thickness of the cracked surface is always equal to \( t_c \). In Fig. 9, the point \( \text{W}_o \) \((X_{\text{Wo}}, Y_{\text{Wo}}, Z_{\text{Wo}}) \) is defined by Eqn. (17) and the point D will be located according to the following assumptions:

1. Point D is on the inner horizontal cylinder.

2. \( |\text{W}_o\text{D}| = t_c = R_1 - R_2 \)

3. The line \( \text{W}_o\text{D} \) will pass through Z-axis.

The first assumption implies that

\[
X_D^2 + Y_D^2 = R_2^2 \tag{22}
\]

while the second assumption means that

\[
(R_1 - R_2)^2 = (X_{\text{Wo}} - X_D)^2 + (Y_{\text{Wo}} - Y_D)^2 + (Z_{\text{Wo}} - Z_D)^2 \tag{23}
\]

From the third assumption

\[
\frac{X_{\text{Wo}}}{Y_{\text{Wo}}} = \frac{X_D}{Y_D} \tag{24}
\]

From Eqns. (22) to (24), one can deduce that the coordinates of point D are given by
4.2 Mapping of normalised 2-D plane to the crack surface

After the crack surface is defined, it is then required to define the crack front which could exist at any location on the crack surface. In order to model the crack front, it is often more convenient to first define it on a normalised $u^\prime$-$v^\prime$ plane and then map it onto the crack surface as shown in Fig. 10. In Fig. 10, the $u^\prime$-axis relates to the crack length, $l_{cr}$, while the $v^\prime$ relates to the crack depth, $d$. Apparently, it is easier to define the $u^\prime$-axis by the polar angle $\alpha$ though it has no direct relationship with the physical length of the crack. In this mapping approach, a crack with any length and size can be modelled at any location. The two crack tips will be located and defined by the polar angles $\alpha_{Cr1}$ and $\alpha_{Cr2}$ in the $u$-$v$ system (Fig. 3). The coordinates $(u^\prime,v^\prime)$ are defined as

\[
\begin{align*}
\alpha_{Cr1} &= \frac{\alpha - \alpha_{Ch1} - \alpha_{Crang}}{\alpha_{Crang}} \\
\alpha_{Cr2} &= \frac{\alpha_{Cr2} - \alpha_{Ch1}}{2} \\
v^\prime &= \frac{d}{t_c}
\end{align*}
\]

where $\alpha$ = The polar angle corresponding to the point $(u^\prime,v^\prime)$
$\alpha_{Ch1}$ = The polar angle which defines the location of crack tip 1.
$\alpha_{Cr2}$ = The polar angle which defines the location of crack tip 2.
d = Depth of the crack.
t_c = Thickness of the chord member.

Note that $u^\prime \in [-1, 1]$, $v^\prime \in [0, 1)$ and $\alpha_{Ch1}, \alpha_{Cr2}, \alpha_o \in (360^\circ]$.

In practice, the crack tip positions are frequently described by defining (or measuring) the arc lengths, $l_{Ch1}$ and $l_{Cr2}$ on the global X-Y-Z coordinate system. The crack length, $l_{Ch1}$, will depend on the position of the crack tips and is defined as $l_{Cr} = l_{Cr2} - l_{Ch1}$. Since the weld is
defined by the polar angles \( \alpha_{Cr_1} \) and \( \alpha_{Cr_2} \) (Eqns. (26) and (27)), it is required to compute the values of \( \alpha_{Cr_1} \) and \( \alpha_{Cr_2} \) from \( I_{Cr_1} \) and \( I_{Cr_2} \) respectively. In this study, the value of \( \alpha_{Cr_1} \) and \( \alpha_{Cr_2} \) are computed from \( I_{Cr_1} \) and \( I_{Cr_2} \) by using a sample approximation procedure [24]. For example, in order to compute \( \alpha_{Cr_1} \) from \( I_{Cr_1} \), starting from the \( v \) axis, a sequent of points will be generated by increasing their polar angles gradually in small step equal to \( \Delta \alpha \). For each of this point, the corresponding arc length, \( I^* \), is computed until \( I^* \geq I_{Cr_1} \). The estimated value for \( \alpha_{Cr_1} \) is then defined as the one corresponding to the arc length which is closest to \( I_{Cr_1} \). In practice, it is found that a value of \( \Delta \alpha = 0.1^\circ \) will be accurate enough for virtually all applications and the computational cost needed is modest [24].

Suppose that the crack front curve is defined by the point \( Cr' \) in the \( u'-v' \) space as shown in Fig. 10. For any point \((u', v')\) on the curve, by using Eqns. (26) and (27), the corresponding value of \( \alpha \) can be obtained. Once \( \alpha \) is known, the coordinates of the point \( Wo \) \((X_{Wo}, Y_{Wo}, Z_{Wo})\) could be computed as outlined in Sections 2 and 3. In order to define the location of the crack front, the point \( Wo \) will be further modified. Assume the crack front is defined by the point \( Cr(X_{Cr}, Y_{Cr}, Z_{Cr}) \) with depth equal to \( d \), then by using a similar approach for the computation of point \( D \), it can be shown that the coordinates of the point \( Cr \) are given by

\[
Cr = \begin{vmatrix}
X_{Cr} \\
Y_{Cr} \\
Z_{Cr}
\end{vmatrix} = \begin{vmatrix}
1 - \frac{d}{R_1} & 0 & 0 \\
0 & 1 - \frac{d}{R_1} & 0 \\
0 & 0 & 1
\end{vmatrix} \begin{vmatrix}
X_{Wo} \\
Y_{Wo} \\
Z_{Wo}
\end{vmatrix}
\]

(28)

### 4.3 Modelling of through-thickness and surface cracks

In tubular joints, through-thickness crack and surface cracks are two most common types of cracks that appear in the joints. A through-thickness crack will propagate through the whole thickness of the chord, \( t_c \), and over a distance along the weld toe as shown in Fig. 10. It is defined as a straight horizontal line in the normalized space. Thus, to model the crack front of the through-thickness crack, one only need to define a horizontal line with thickness \( t_c \) in the normalized space. It will then be mapped onto the crack surface using Eqn. (28). In general, the equation for a through-thickness crack in the normalized space can be expressed as
\[ v' = \frac{d}{t_c} = 1 \quad \text{for } u' \in (-1, 1) \]  

(29)

A surface crack is defined as a curve that propagates on the cracked surface with varying depth along the weld toe. When the crack front is mapped to the normalized plane, it is similar to a semi-elliptical curve as shown in Fig. 10. In order to model the surface crack, a semi-elliptical curve will be first defined in the \( u' - v' \) space and then mapped onto the crack surface by Eqn. (28). In general, an elliptical curve defined in the \( u' - v' \) space can be written as

\[
\frac{u'^2}{a^2} + \frac{v'^2}{b^2} = 1
\]

(30)

where \( a \) and \( b \) are shape constants that define the ellipse.

Since when \( \alpha = \alpha_{c_h}, u' = 1 \) and \( v' = 0 \), one can deduce that \( a = 1 \). For the value of \( b \), it is in fact the ratio of the maximum crack depth, \( d_{\text{max}} \), to the thickness of chord, \( t_c \), and is defined as

\[
b = \frac{d_{\text{max}}}{t_c} \leq 1.0
\]

(31)

It should be noted that in addition to a semi-elliptical curve, one can also define any other crack pattern in the \( u' - v' \) space and then map the curve to the crack surface by using Eqn. (28).

5 Finite element mesh generation procedure

5.1 Finite elements used and the basic concepts employed for mesh generation

In both the cracked joint models developed by Bowness and Lee [14] and Cao et al. [6], only two types of solid quadratic elements are employed. They are the 3D prism singular elements and the 3D hexahedral elements, which are used to model the volume around the crack front and the other parts of the joints respectively. It is found that the limited choice of element types in their models resulted in the formation of distorted elements with high aspect ratio. In fact, further studies found that for a cracked joint model with complex geometry, it is difficult to obtain a mesh with good aspect ratio elements if only the hexahedral and the prism elements are used. Towards this end, five types of elements are employed in the current mesh generation procedure. The linear and quadratic forms of these five types of elements are listed in Tables 2 and 3 respectively. The linear order elements are used in the raw mesh generation and the mesh will be upgraded to quadratic elements for FE analysis.
In the present mesh generation scheme, 3D solid hexahedral elements are employed to model the tube members far from cracks (the far field region) while prism elements are employed to model the transition area between the refined region near the crack surface and the far field region. The quarter-point crack tip (QPCT) elements [30,31], of which the mid-side node are moved to the quarter point for the edge connected to the crack front, are employed to model the volume around the crack front. The QPCT element has been used by many researchers to model crack in plane plate and are found to be accurate and efficient to model the CTOD [32,33]. The tetrahedral elements are used for the connection between the QPCT elements and other types of elements surrounding the crack front. Apart from that, pyramid elements will be used to connect the prism elements with the tetrahedral elements around the crack front curve. Note that a pyramid element can be formed by collapsing one of the faces of the hexahedral element into a point as shown in Tables 2 and 3.

During the mesh generation procedure, the whole domain of the joint will first be divided into several zones as shown in Fig. 11. This approach will increase the efficiency and flexibility of the mesh generation scheme. It can simplify the problem by breaking down the whole joint into a number of simpler pieces that will be easier to handle than the whole joint. In addition, this also enables one to check and modify the mesh in each of the zones quickly and independently without affecting other zones and hence, increase the flexibility of the scheme when cracks are present in the joint.

After the mesh in each of the zones is generated, it will then be merged together to form the linear raw mesh for the whole joint. The raw mesh will then be processed and upgraded to a higher order mesh after the merging procedure is completed. The advantage of first generating the linear mesh is that it can largely reduce the complexity of the mesh generation procedure and thus highly distorted elements could be identified and eliminated. Besides, meshes in different zones could be merged and extracted conveniently since linear elements will only contain corner nodes.

After the raw mesh of the uncracked or cracked joint is formed, the elements in the raw mesh will be upgraded to quadratic elements by inserting mid-side nodes to the linear elements [34]. Next, the quadratic elements will be further processed in which the mid-side nodes will be moved to model the surface geometry more accurately. Finally, for the QPCT elements, the midside nodes will be shifted to the quarter points for the edges that are connected to the crack front.
5.2  Generation of raw mesh for tubular joint without cracks

As shown in Fig. 11, the Y-joint model will be divided into different zones. Most of the zones are concentrated near the joint with finer elements as it will be subjected to high stress gradient. The tubular joint is divided into three main zones, namely the fine mesh zone, the transition zone and the far field zone. The fine mesh zone CF is located at the joint intersection with three layers of elements in the thickness direction so that the stress concentration can be captured accurately. At the far field zone (zones A, E, H, ExtenCHL and ExtenCHR), only one layer of 3D solid elements will be employed since the stress distribution is nearly uniform in this region. Between the fine mesh zone and the far field zone, the transition zone (zones B, D and G1) is designed to increase the number of layers of elements in the thickness direction from one to three. Another transition zone, zone G2, will double the number of elements in the radial direction of the brace from sixteen to thirty two. In order to model the full length of the chord, additional meshes (ExtenCHL and ExtenCHR) are added to the two ends of the chord.

It should be pointed out that during the mesh generation process, the mesh generator will refer to the geometrical model for the formation of elements and nodes. In addition, for tubular joint without cracks, only the hexahedral elements are needed for the discretization of all the zones. After the raw mesh is formed, it will be further processed and upgraded to a quadratic mesh (Table 3) before the FE analysis is carried out.

5.3  Generation of raw mesh for tubular joint with cracks

Through-thickness cracks and surface cracks are the two most commonly found crack types appearing in tubular joints. Despite that the geometry of both crack types can defined on a same crack surface (Section 4.1), the mesh generation process for them is rather different. In fact, the mesh generation procedure for a joint model with surface crack is much more complicated than that for a through-thickness crack [6,14]. However, the same general approach used in the mesh generation for a joint without a crack is also applicable to the mesh generation process of model with cracks.

As shown in Fig. 12, the meshes for the cracked tubular joint are again generated in a zone by zone manner. However, compared to the mesh for uncracked joint, there are some changes in some of the zones and the corresponding sub-meshes. Parts of the sub-mesh in the zones CF and D will be extracted to form a new zone called CRBLOCK. It will then be further
modified in order to generate meshes for through-thickness or surface cracks. After the modification, this sub-mesh will be merged together with other sub-meshes to form the raw mesh of the cracked tubular joint. Actually, the zones that will be extracted are based on the types of crack to be modelled at the joint. For through thickness cracks, only part of the sub-mesh in zone CF is extracted (Fig. 13) whereas for surface cracks, part of the sub-mesh in zones CF and D (Fig. 14) will be extracted and modified. It is noted that the number and the location of elements extracted from the affected zones will depend on the length and the location of the crack. Therefore, before the mesh generation process is started, the location of the crack tips must be first defined through the parameters $\alpha_{C1}$ and $\alpha_{C2}$ as described in Section 4.2.

5.4 Generation of raw mesh for tubular joint with through-thickness crack

Based on the locations of the crack tips, a tube of elements (Fig. 15) will be generated through the whole thickness of the chord at two ends of the crack. This tube will consist of two rings of elements. Each ring of the tube will consist of eight elements and is generated in a local u-v coordinate system. The elements will first be rotated and aligned to the global position before it is inserted back to the joint. After the tube is aligned and positioned, elements of the tube will be generated through the chord thickness accordingly in each layer.

As shown in Fig. 16, twenty-seven QPCT elements are placed along the thickness direction in order to capture the CTOD accurately at the crack tip. The mesh density near the tip is then reduced to nine layers and subsequently three layers of elements as the distance away from the crack tip increases. The elements along the cracked surface will be separated and nodes along this cracked surface will be duplicated and connected to the corresponding separated elements in order to form the crack. Finally, the tube will be inserted back to zone CRBLOCK. As for the void between the zone CRBLOCK and the tube, it will be filled with six layers of prism and three layers of hexahedral elements.

5.5 Generation of raw mesh for tubular joint with surface crack

Even though a well defined and consistent geometrical model for the surface crack is available, generating a well graded mesh for surface crack is still a difficult task. Thus, before the mesh generation, some studies have been carried out in order to design an appropriate meshing scheme specifically for surface crack. It is found that, for a typical surface crack, the
crack front usually passes through at least one layer of the elements inside the chord thickness. Unlike the through-thickness crack, the surface crack front will not just passes through the thickness of the chord only. Instead, the crack front propagates from one end along the cracked surface to the other end while penetrating through the chord thickness. Obviously, the 3D mesh generation process around the crack front is complicated for such geometry. Thus, in order to reduce the complexity of the mesh generation procedure, the first layer of elements in zone CRBLOCK (Fig. 12) will be shifted down towards the bottom of the chord such that the crack front will be entirely located within this layer of elements. In order to model the surface crack, zone CRBLOCK for a surface crack is further sub-divided into four blocks, namely, SFBLOCK-A, SFBLOCK-B, DCUBE-A and DCUBE-B as shown in Fig. 17. Block SFBLOCK-A, block SFBLOCK-B and block DCUBE-A are extracted from zone CF while block DCUBE-B is extracted from Zone D. In addition, in order to maintain the aspect ratio of the elements around the crack front, the first layer elements will be adjusted such that the two crack ends will lie in the middle of the block SFBLOCK-A.

Similar to a through-thickness crack, a tube of elements will be generated from one crack end to another in block SFBLOCK-A as shown in Fig. 18. The void between the tube and the block SFBLOCK-A will be filled with pyramid, prism and tetrahedral elements as shown in Fig. 19. It is noted that the first ring of the tube will consist of QPCT elements as it is next to the crack front. A similar mesh generation procedure is also applied for block SFBLOCK-B. It should be noted that block SFBLOCK-A will be discretized with pyramid, prism and tetrahedral elements in order to connect it with the tube elements. Hence, the side faces of this block will be discretized into eight triangles as shown in Fig. 19. However, this will pose a problem (incompatibility of surface) when merging the sub-mesh of zone CRBLOCK with zones CF and D. In order to overcome this problem, blocks DCUBE-A and DCUBE-B are introduced. After these two blocks are extracted from zones CF and B respectively, they are modified by discretized them with tetrahedral and pyramid elements. The main purpose of blocks DCUBE-A and DCUBE-B is to link the side faces of block SFBLOCK-A with the sub-meshes in zones D and B.

After the meshes are generated in zone CRBLOCK (i.e. blocks SFBLOCK-A, SFBLOCK-B, DCUBE-A and DCUBE-B), it will then be merged with other zones to form the raw mesh of the cracked joint. Finally, it will be upgraded and processed to a higher order mesh before the analysis is carried out.
6 Mesh generation examples

In this section, three Y-joint models are employed to demonstrate the use of the mesh generator developed for the discretization of uncracked and cracked tubular joints. The geometrical parameters (as shown in Fig. 20) of the joint are listed below:

\[
\begin{align*}
\theta &= 60^\circ \\
t_c &= 25.4\text{mm} \\
t_b &= 25.4\text{mm} \\
R_1 &= 177.8\text{mm} \\
R_2 &= 152.4\text{mm} \\
r_3 &= 136.5\text{mm} \\
r_4 &= 111.1\text{mm} \\
L_{Ch} &= 4130.0\text{mm} \\
L_{Br} &= 2159.0\text{mm}
\end{align*}
\]

6.1 Mesh generation for uncracked Y-joint

Based on the geometrical parameters defined above, the mesh for the half model of the uncracked joint is generated and is shown in Fig. 21. The mesh generated consists of 5974 nodes and 958 elements. The zoom view of the mesh generated near the joint intersection are shown in Fig. 22. From Fig. 22, it can been seen that small elements with good shape are generated in this region. This will allow a more accurate computation of the SCFs along the intersection curve of the joint.

6.2 Mesh generation for Y-joint with through-thickness crack

The same geometrical parameters of the uncracked Y-joint model was reused in here during the generation of mesh with through-thickness crack. In this example, the through thickness crack are placed symmetrically at the crown heel. The propagation length of the crack, \( l_Cr \) (Section 4.2), the position of the first crack tip, \( l_{Cr} \) (Fig. 20), are equal to 112.13mm and 449.93mm respectively. Zoom view of the mesh near the region of the through-thickness cracked are shown in Fig. 23. Note that in Fig. 23, in order to display the elements around the cracked clearly, only the half model of the mesh was shown and a uniform tensile force was applied to the brace to "open up" the through-thickness crack. The half model of the mesh contains 7588 nodes and 1450 elements.

6.3 Mesh generation for Y-joint with surface crack

The same geometrical parameters of the through-thickness crack Y-joint models used in Section 6.2 are employed for mesh generation of joint with surface crack. The surface crack with the same length defined in Section 6.2 is symmetrically placed at the crown and the maximum depth of the surface crack is equal to 0.5\( t_c \). The finite element mesh generated (half
model) consists of 7529 nodes and 1524 elements. Again, a uniform tensile force was applied to open up the crack. The mesh generated near the surface crack are shown in Fig. 24.

7 Conclusions and future work
In this paper, an accurate and consistent geometrical model for general welded and cracked tubular Y-joints is presented. The geometry of the tubular Y-joint intersection is first analysed to facilitate the subsequent detailed modelling of welding and cracks. Based on the geometrical analysis, welding details compatible with the American Welding Society and American Petroleum Institute standards are modelled by modifying the inner and outer intersecting curves. Furthermore, a flexible mapping approach is suggested for the modelling of crack surface. Based on this mapping approach, through-thickness crack and surface crack of arbitrary length and located along the weld toe can be included in the geometrical model of the joint.

Based on the geometrical model, a special tailor-made automatic mesh generator has been implemented to generate structural meshes for finite element modelling. The main mesh generation concept used is to divide the problem domain into different zones whereby sub-meshes with different element density are generated in a zone-by-zone manner. Combination of the sub-meshes will then form the raw mesh, which is subsequently upgraded to quadratic mesh for finite element analysis. The computational cost needed for the mesh generation procedure is modest. In all the mesh generation examples presented, only a few seconds of CPU time are needed to generate the final quadratic mesh on a low-ended personal computer. One of the possible areas for future development is to validate and compare the present model with some actual fabricated Y-joints. This will allow the modeller to further fine tune the present model, particularly the parameters $F_{o}\text{outer}$, $F_{o}\text{inner}$, $m$ and $n$ (Section 3) and the constant $b$ (Section 4). Apart from that, the present geometrical model could be further developed to cover more types of tubular joints such as the K-, X- and multi-planar joints.

Another possible area which is now under series consideration is to carry out thorough parametric numerical studies on the SCFs and SIFs of uncracked and cracked tubular Y-joint and validate the solutions obtain with some published results [1,4-6,13,17]. Since the geometrical model used is easy to use, it will allow researchers to carry out fast parametric studies on the effect of different modelling parameters on the response of the joint under different loading and boundary conditions. More importantly, since the model developed in
this study is consistent and reproducible, it allows different researchers to compare their numerical results obtained.

References


Fig. 1: Mapping of a plane to a circular surface

Fig. 2: Coordinate systems for a general Y-Joint

Fig. 3: Double mappings of a circle to an intersecting curve (plan view)
Fig. 4: Geometry of the dihedral angle for a tubular joint

Fig. 5: Welded tubular Y-joint
Fig. 6: Inner and outer intersecting curves with the weld path at the joint – plan view

Fig. 7: Geometry and modelling of weld path – enlarged plan view

Fig. 8: Welded joint and non-welded joint
Fig. 9: Formation of cracked surface in the chord thickness

Fig. 10: Mapping of 2D normalised plane to a 3D crack surface
<table>
<thead>
<tr>
<th>Main Zone</th>
<th>Mesh of Zone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fine Mesh</td>
<td>CF</td>
</tr>
<tr>
<td>Transition</td>
<td>B, D, G1, G2</td>
</tr>
<tr>
<td>Far Field</td>
<td>A, E, H, ExtenCHL, ExtenCHR</td>
</tr>
</tbody>
</table>

Fig. 11: Mesh generation of tubular joints without cracks in different zones
Fig. 12: Mesh generation of tubular joints with cracks in different zones

Fig. 13: Zone CRBLOCK extracted from zone CF for the mesh generation of through-thickness crack
Fig. 14: Zone CRBLOCK extracted from zones CF and D for the mesh generation of surface crack

Fig. 15: A tube of elements generated at one of the tip for through-thickness crack
Fig. 16: Meshes around the through-thickness crack

Fig. 17: Division of zone CRBLOCK for surface crack
Fig. 18: A tube of elements generated along the surface crack front in block SFBLOCK-A

Fig. 19: Mesh around the surface crack front
Fig. 20: Tubular Y-joint model for mesh generation examples

Fig. 21: Mesh generated for un-cracked tubular Y-joint
Fig. 22: Zoom view for the uncracked Y-joint near the joint intersection

Fig. 23: Mesh around the region of the through-thickness crack

Fig. 24: Mesh around the region of the surface crack
### Table 1: Summary of $k_{API}$ and $k_{AWS}$

<table>
<thead>
<tr>
<th>Dihedral Angle, $\gamma$</th>
<th>Minimum $k_{API}$</th>
<th>Minimum $k_{AWS}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$50^\circ$-$135^\circ$</td>
<td>1.25</td>
<td>$\frac{1}{\sin \gamma}$</td>
</tr>
<tr>
<td>$35^\circ$-$50^\circ$</td>
<td>1.5</td>
<td>$\frac{1}{\sin \gamma}$</td>
</tr>
<tr>
<td>Below $35^\circ$</td>
<td>1.75</td>
<td>2.0 (for $\gamma &lt; 30^\circ$)</td>
</tr>
<tr>
<td>Over $135^\circ$</td>
<td>Build out to full thickness but need not exceed 1.75</td>
<td>Build out to full thickness but need not exceed 1.75</td>
</tr>
</tbody>
</table>

### Table 2: 3-D Solid linear elements

<table>
<thead>
<tr>
<th>Element Types</th>
<th>No. of Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Hexahedral/Cubic Element – (H8)</td>
<td>8</td>
</tr>
<tr>
<td>2. Prism/Wedge – (P6)</td>
<td>6</td>
</tr>
<tr>
<td>3. Quarter Point Crack Tip Element – (QP6)</td>
<td>6</td>
</tr>
<tr>
<td>4. Tetrahedron – (T4)</td>
<td>4</td>
</tr>
<tr>
<td>5. Pyramid – (PR8) (Collapsed Hexahedral)</td>
<td>8</td>
</tr>
</tbody>
</table>

### Table 3: 3-D Solid quadratic elements

<table>
<thead>
<tr>
<th>Element Types</th>
<th>No. of Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Hexahedral / Cubic Element – (H20)</td>
<td>20</td>
</tr>
<tr>
<td>2. Prism/Wedge – (P15)</td>
<td>15</td>
</tr>
<tr>
<td>3. Quarter Point Crack Tip Element – (QP15) (Collapsed Prism)</td>
<td>15</td>
</tr>
<tr>
<td>4. Tetrahedron – (T10)</td>
<td>10</td>
</tr>
<tr>
<td>5. Pyramid – (PR20) (Collapsed Hexahedral)</td>
<td>20</td>
</tr>
</tbody>
</table>