Journal of Mechanical Engineering and Sciences
ISSN (Print): 2289-4659; e-ISSN: 2231-8380
Volume 13, Issue 2, pp. 5048-5073, June 2019

DOI: https://doi.org/10.15282/jmes.13.2.2019.20.0417

Fatigue life assessment of welded joints in a crane boom using different structural stress approaches

Brahami Riad[1*], **Hamri Okba**[2], **Sfarni Samir**[1]

[1]Laboratoire de Génie de la Construction et Architecture (LGCA), Faculté de Technologie, Université de Bejaia, 06000 Bejaia, Algeria
Phone : +213 664429440
[*]Email : Brahami.riad@gmail.com
[2]Laboratoire de Génie Mécanique et Développement (LGMD), Ecole Nationale Polytechnique, 16200 Alger, Algeria

ABSTRACT

This article presents a study of the fatigue strength of welded parts in a crane boom. A finite element analysis was carried out over the whole structure. Two critical welded zones were identified and a detailed analysis was carried out on them, in the form of sub-models. Three different approaches for estimating the structural stress in welded zones, were presented and applied to each sub-model (surface extrapolation, Xiao-Yamada, and Dong's method). Results were compared and discussed. The evaluation of fatigue resistance by the use of appropriate S-N curves for each method was also carried out and discussed. The use of these approaches on a complex industrial structure, and on tubular joints with hollow sections required to perform many adaptations and to solve several difficulties. The results obtained show a relatively good correlation between the surface extrapolation and Xiao-Yamada methods. However, the method of Dong shows more conservative results. Analysis and discussion of the results are presented in the conclusion.

Keywords: Fatigue assessment; welded joints; structural stress approach; finite element analysis; fatigue strength.

INTRODUCTION

The structural integrity of mechanical assemblies is an absolute necessity in the industrial world, so that each mechanical design ensures its functionality during the whole life of a project. Today, various welding processes, such as Metal Inert Gas (MIG), Metal Active Gas (MAG), Tungsten Inert Gas (TIG), etc., widely used in the mechanical industry, have made it possible to design and erect large structures as bridges, offshore floating platforms, huge storage tanks… In engineering, the assessment of welded fatigue assemblies is generally a work restricted by design guides, among which are: Eurocode 3 [1] for steel structures and assemblies, BS 7608 [2] and the IIW recommendation for tubular [3] and non-tubular [4] structures.

Several fatigue assessment approaches for welded parts, are proposed [5–7]. Each of these approaches has a stress or strain determination method, appropriate for its assessment.

The nominal stress approach, also known as the global approach, continues to be the primary method of fatigue assessment. Over the years, it has acquired extensive experience in fatigue analysis of welded parts and structures. The concept of the nominal stresses approach is still widely used. Its application requires to know the applicable allowable value in relation to a corresponding classified structural detail [8,9]. This approach is based on the analysis of the stress field, not far from the potential site of cracking. The fatigue strength S-N curve for each welded structural detail is abbreviated to FAT, which means "Fatigue design class". This abbreviation is combined with a number which indicates the permissible nominal stress range $\Delta\sigma_n$ at $N = 2 \times 10^6$ cycles, with a probability of survival Ps = 97.7 % [4]. but the development of the computer tool has pushed mechanical engineering to very high levels, however, the design requirements of manufacturers is increasingly complex, generally cause significant limitation of the nominal stress approach.

The structural stress approach, also called the geometric stress approach, is an approach among local approaches. According to Radaj [10,11] the application of the approach in the early times goes back to Haibach, where in a historically early contribution, a series of experimental tests using strain gauges was made by the latter. It was then established and extended on the fatigue assessment of tubular welded joints of offshore structures. However, a new term called Hot Spot Stress (HSS) for the evaluation of structural stress was introduced in this period. Taking advantage of the progress of FE calculation software and computers, the (HSS) expanded on non-tubular joints [12,13]. For this, and in order to assess the fatigue strength by using the structural hot spot stress approach, the analysis of the structural hot spot stress range in mechanical elements, is a necessary operation for estimating the service life of a welded mechanical assembly by using dedicated S-N curves. The HSS approach, describes always the macrostructural behaviour, without taking into account local notches. In addition, different measurement methods can be applied such as strain gauges, analytical calculations using engineering formulas and also finite element analysis which is widely used [11]. Generally premature cracking of welds develops initiating from the toe or the root of the welds, which are places of concentration of local stresses by excellence. Also, these weld joints are the cause of considerable uncertainties linked to geometric irregularities, heterogeneity of the materials and potential residual stresses. These irregularities are known to have an influence on the fatigue strength. As a result, HSS range determination takes into account the concentration of critical point stresses (hot spot) that are highly dependent in physical models on the local weld profile and, in numerical analysis on mesh refinement [14].

The determination and computation of structural hot spot stress is a know-how generally confined to experts who often deal with problems related to the fatigue of welded structures. However, several methods have been developed to ensure accuracy of structural stress, which plays an important role in determining the life of any welded structure. Many problems related to experimental manipulations can be overcome using FEA [12,15–17].

Several research works propose FE-based techniques to calculate and evaluate the structural stress. Hobbacher [4] and recently Niemi et al [18] have proposed a FEA technique, in order to determine the structural hot spot stress. These methods are approved by IIW, and are widely used for the fatigue assessment of welded joints in engineering offices. It is generally based on the linear or quadratic extrapolation of the stresses measured on the surface of parts welded at two or three reference points from the hot spot in the weld toe. Another technique proposed by Dong [19] has given a new definition on the distribution of equivalent structural stress in a way that conforms to the theory of elementary structural mechanics. The two components of membrane and bending stress adapted to this new definition are used in this technique to calculate the stress at the weld toe. Thus, Dong concludes his approach by analysing a large amount of fatigue test data found in literature and propose a single S-N curve called "Master S-N curve", which correlates several types of welded joints under different solicitations. More recently Xiao and Yamada [20] proposed to extract the estimated stress at 1 mm below the surface of the weld toe in order to calculate the structural stress. Later, several proposals, have been the subject of new methods of structural stress approach as an example Kim et al [21] and Shen et al [22], but their applications at the industrial scale always remain individual experiences. Few works have been found in the open access literature, concerning the application of the "Xiao - Yamada" and Dong's methods in the calculation of the structural stress of complex industrial structures. The absence of application of these methods on some typical welded joints (such as hollow section tubular joints) should also be noted.

In this work, an evaluation of the fatigue life in the most critical areas of a typical crane boom (GMR 2010) manufactured by the national company ENMTP is performed. This structure is modelled in 3D and analysed by the finite element method in order to obtain the structural hot spot stress and structural stress in the most solicited weld toes. The calculation of structural stresses at the weld toe is done by three methods. The application of two modes of analysis namely the surface analysis mode represented by the IIW method and through the thickness represented by Dong [19] and Xiao-Yamada [20] on parts of industrial scale structure, has the aim to express the reliability and robustness of each of these methods on curved or tubular type joints by comparing the lifetimes obtained through the S-N resistance curves appropriate to each method. The steps of evaluation and estimation of service life are presented in the following diagram (see **Error! Reference source not found.**):

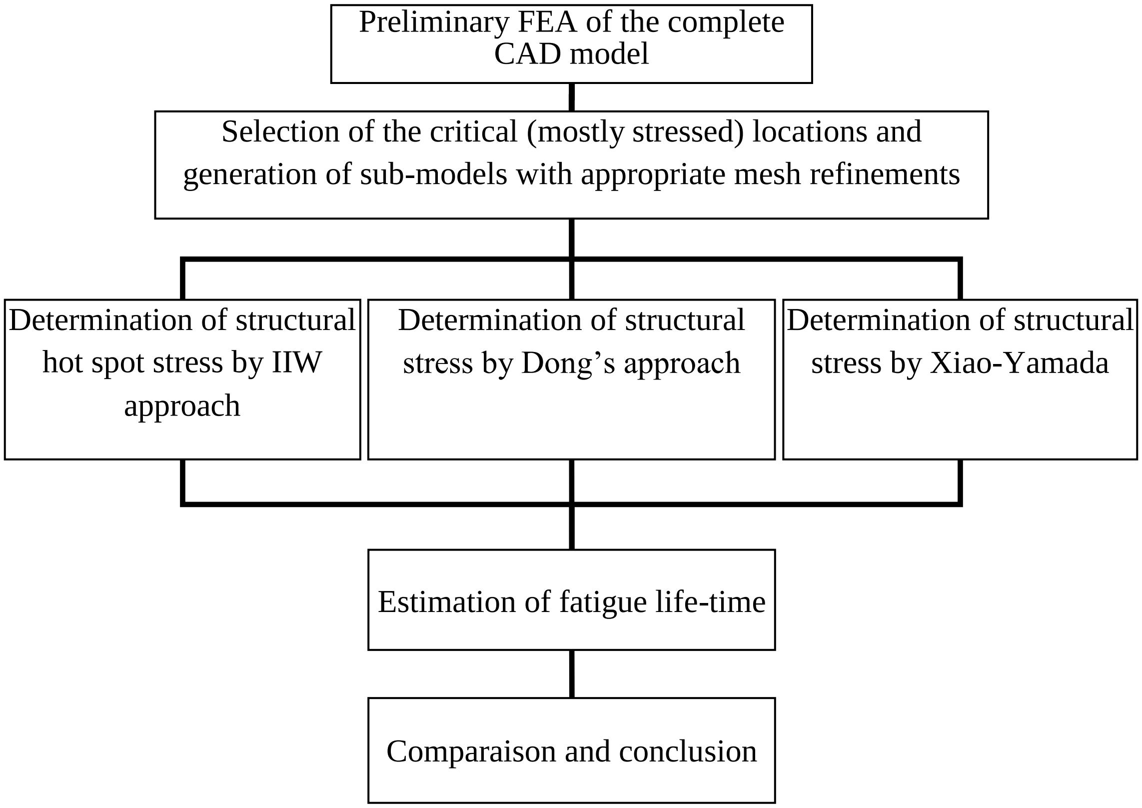

Figure 1. Diagram representing the main steps of the presented study.

TECHNIQUES FOR THE DETERMINATION OF STRUCTURAL STRESS

Determination of the structural stress at the weld toe using surface stress extrapolation

The determination of the structural hot spot stress, according to the recommendations of IIW [4,18] takes into account all the effects of increasing stress in a structural detail except that due to the local weld profile itself. while generally, a local notch causes a nonlinear maximum stress σ_{nl}, the related dimensions and load parameters in the vicinity of the joint give the structural stress. However, the determination of structural hot spot stress is usually made on the surface of the component designed to be evaluated. Generally, these stresses σ_{hs} are defined for the structures in plates, shells and tubes.

If structural discontinuities are not comparable to a structural detail class, this automatically makes the definition of the nominal stress approach unsuitable. For this reason, it is usually replaced by structural hot spot stress approaches. The determination of the stresses at the reference points form the hot spot under consideration, makes it possible to calculate the structural hot spot stress by the extrapolation method.

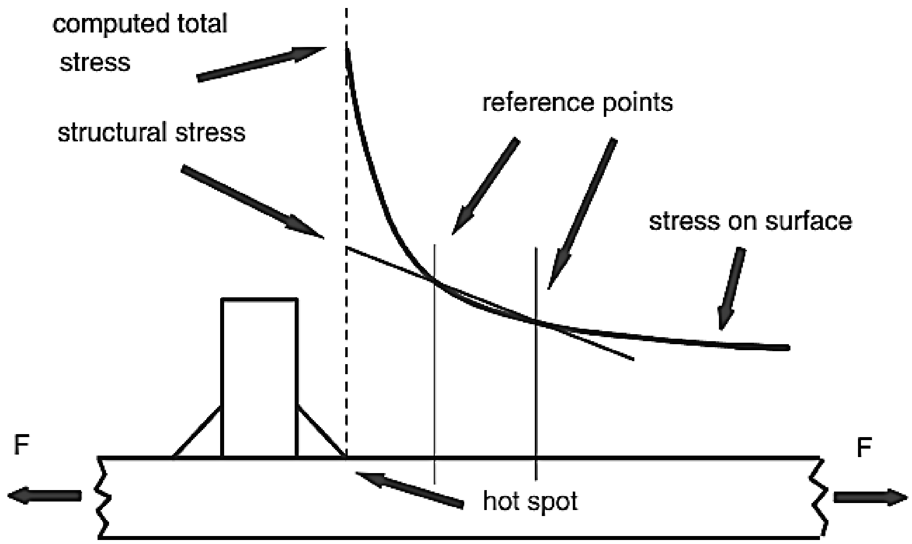

Figure 1. Definition of structural hot-spot stress [4].

The method as defined in [4,18] is limited to the evaluation of weld toe. However, other potential fatigue crack initiation sites, such as weld root, can be assessed using the structural hot spot stress approach, always by using σ_{hs} as reference for evaluating the stress in the zone of interest.

In addition to the definition of the structural hot spot stress, the hot spots that are located on weld toes are classified in two types, which are defined according to their orientation and their location on the plate. Table 1 and **Error! Reference source not found.** give a description and show the two types of hot spots.

Table 1. Type of hot spots.

Type	Description	Determination
a	Weld toe on plate surface	FEA or measurement and extrapolation
b	Weld toe at plate edge	FEA or measurement and extrapolation

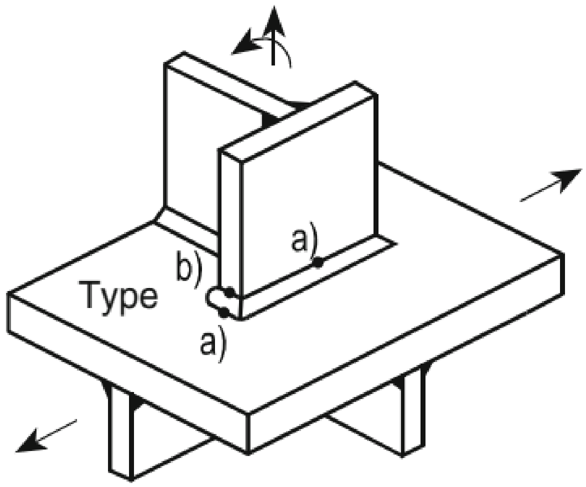

Figure 2. Types of hot spots [4].

Two methods are possible for the determination of structural hot spot stresses, which acts normal to the weld toe. It can either be determined by a special FEA procedure or by extrapolation from stress measurements. For these methods, it is first necessary to establish the reference points. Then, the structural hot spot is determined by extrapolating the stresses of these reference points to the weld toe. In order to avoid any influence related to the form of notches, the reference points are chosen at a specific distance from the weld toe (for example at a distance of 0.4t, or at 1.0t). Three methods are possible for the identification of hot spots. They will be presented hereafter. They can be used either by measuring (experimentally) the stress on several different reference points, or by post-treating results of a FEA. These results are usually compared to experimental results for fatigue life prediction.

Calculation of Structural Hot Spot Stress

The analytical methods do not allow to obtain the structural hot spot stress and the parametric formulas are rarely available. For this the analysis of the structural discontinuities and details is generally done by the FEA.

It is possible to use models either with solid or with shell elements. The stresses have to be evaluated at the specified reference points. The Finite Element model has to be built in a way to have a sufficient extent, in order to avoid influence of the Boundary Conditions on the zone of interest (welded zone).

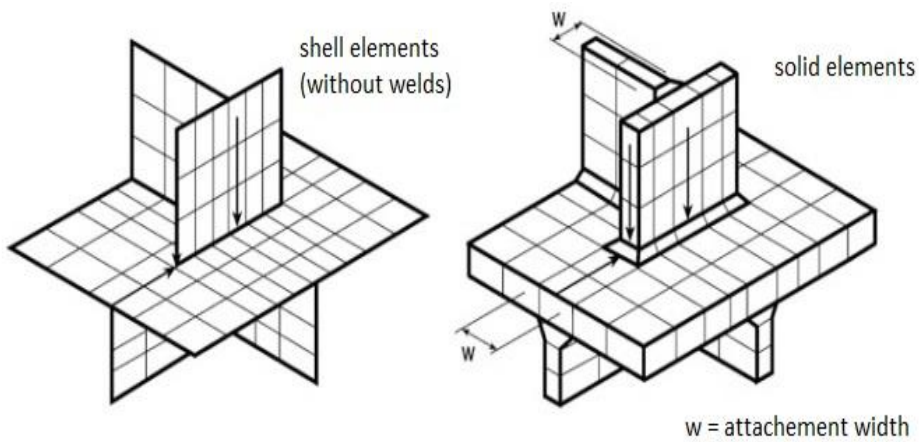

Figure 3. Typical meshes and stress evaluation paths for a welded detail [4].

Surface stress extrapolation methods

The principle of these methods is to define reference points for post-treating the stress results at a specific distance of the weld toe, in order to avoid local effects. Different methods are proposed, according to the hot spot type and mesh size.

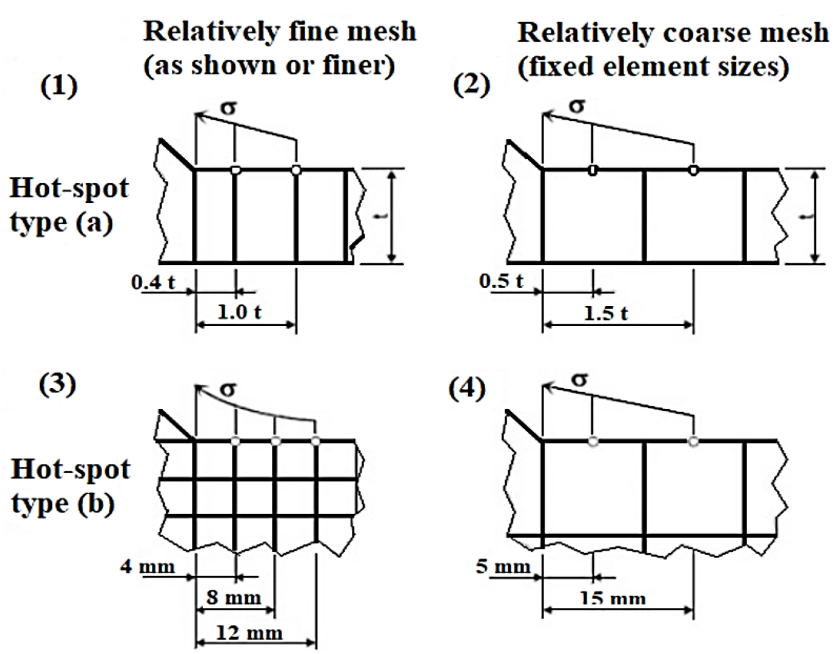

Figure 4. Reference points for different mesh sizes, type (a) (schemes 1 and 2), type (b) (schemes 3 and 4) [4].

The following extrapolation equations give the way to obtain the structural hot spot stress σ_{hs} from the stresses raised in the reference points. For hot spots of type (a), three equations of extrapolation are presented. Two equations are related to fine mesh models, and the last one is related to coarse mesh models. Eqn. (1) as shown below requires a fine mesh with an element length that does not exceed 0.4 t around the hot spot area. The hot spot stress calculation focuses on two reference points which are distant at 0.4t and 1.0t of the weld toe, respectively.

$$\sigma_{hs} = 1.67\ \sigma_{0.4t} - 0.67\sigma_{1.0t} \tag{1}$$

For the Eqn. (2) always with a fine mesh, it is rather based on three points of reference, which are respectively at a distance of 0.4t, 0.9t and 1.4t from the weld toe. The equation is written as follows:

$$\sigma_{hs} = 2.52\sigma_{0.4t} - 2.24\sigma_{0.9t} + 0.72\sigma_{1.4t} \tag{2}$$

On the other hand, Eqn. (3) is related to coarse mesh models. In this case, the length of the element is approximately equal to the thickness of the plate. Two reference points are needed at 0.5t and 1.5t from the hot spot, this equation is presented as follows:

$$\sigma_{hs} = 1.50\ \sigma_{0.5t} - 0.50\ \sigma_{1.5t} \tag{3}$$

Concerning hot spots of type (b), two equations are given according to the mesh. It has to be noted that, unlike for type (a), the reference points in type (b) hot spots, are given at absolute distances. In the case of a fine mesh Eqn. (4) below shows a quadratic extrapolation, where the reference points are taken at distances of 4, 8 and 12 mm from the hot spot.

$$\sigma_{hs} = 3\ \sigma_{4mm} - 3\ \sigma_{8mm} + \sigma_{12mm} \tag{4}$$

Eqn. (5) is used in the case of a coarse mesh with a length of element of approximately 10 mm. This equation is based on a linear extrapolation. The two reference points must be positioned at the mid-side nodes of the two first elements, i.e. at 5 and 15 mm of the hot spot, respectively.

$$\sigma_{hs} = 1.5\ \sigma_{5mm} - 0.5\ \sigma_{15mm} \tag{5}$$

Tubular Joints

The determination of the structural hot spot stress on the tubular joints, passes automatically by special recommendations published in [3]. These recommendations are approved by IIW. According to Hobbacher [4] it is allowed to use two reference points, and perform a linear extrapolation of the stresses calculated or measured. It is mentioned that the measure of simple uniaxial stress is convenient.

Parametric formulae have been established for the stress concentration factor k_{hs}, in many joints between rectangular and circular section tubes. Thus, the structural hot spot is given by:

$$\sigma_{hs} = k_{hs}\ \sigma_{nom} \tag{6}$$

where σ_{nom} is the nominal axial or bending stress in the braces, calculated by elementary stress analysis or uniaxial measurement.

Determination of the structural stress by the method of Dong

Following crack propagation considerations, Dong [19] proposed modifications related to the resulting stresses from coarse mesh finite element models. The propagation of cracks starting at weld toe and crossing the thickness of the plate, is controlled by the total stress, normal, to the path of crack. An analysis based on the finite element method (FEA) allows to evaluate the linearized structural stress from the weld toe in the direction of the thickness. The distribution of the linearized stresses on the thickness of the plate constitutes membrane stresses and bending stresses. It is used to predict fatigue life. The following figure shows the stress distribution across the thickness used by Dong for determination of bending and membrane stresses.

Determination of the structural stress by the method of Xiao-Yamada

An alternative method for the assessment of structural stress or geometrical stress of welded parts and welded structural details has been proposed by Xiao and Yamada [20]. Theoretically, this method is based on a generalized analysis of crack propagation. It concerns in particular the cracks that originate at weld toe. Xiao and Yamada [20] found that fatigue life is generally related to crack propagation. For this, it better can be expressed by the stress value at a point 1 mm below the surface on the expected cracking path (assumed to be normal to the surface of the plate at the level of the weld toe). This stress at 1 mm below the surface of the welded part has a relation with the lifetime of the propagation of crack or with the strength of the structural detail considered.

In order to be able to calculate the structural stress at 1 mm, the finite element method remains the method used with classical requirements on mesh design, knowing that the size of the element must not exceed 1 mm. However, it has to be mentioned that the method of Xiao-Yamada may not be suitable for thin plate (t ≤ 5 mm), as explained in [18].

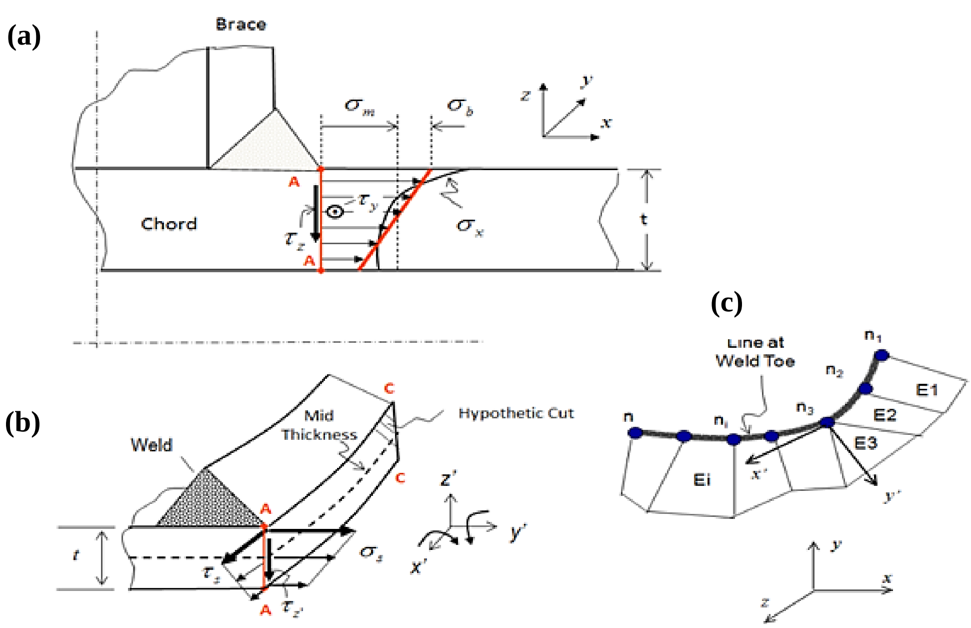

Figure 5. Definition of structural stress across the thickness according to the method of Dong [25].

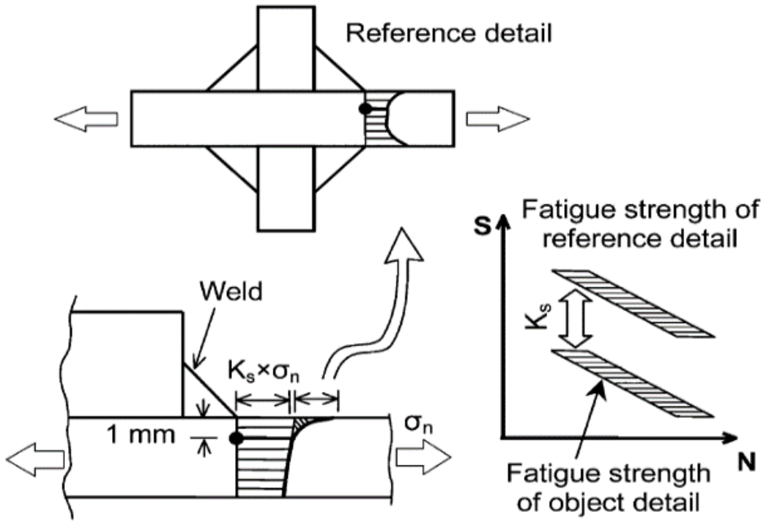

Figure 6. Determination of structural stress according to the method of Xiao-Yamada [20].

PRELIMINARY STUDY OF THE STRUCTURE

A 3D digital mock-up of the crane boom (GMR 2010) was performed using a CAD software. The dimensions are based on the drawings consulted in the design office of ENMTP company. The structure itself generally consists of metal profiles (UPN, tubes, L profiles…) of steel grade S235JR (according to European standards). Based on the CAD model, a finite element analysis of the structure was then performed in order to identify the stress concentration areas. Using a finite element computation software, a preliminary mesh of quadratic tetrahedron elements is applied on the whole structure.

This boom is articulated at the end by two cylindrical pieces, which allows the boom to have a rotational degree of freedom with respect to the mast (namely a rotation along the transversal axis, which allows to increase the lifting height to 30.5 m). A wire rope type (6 x 37 Fiber Core Wire Rope of 36 mm diameter) is attached between Jib Tie and the gantry of Boom. Figure 8 shows a complete schema on the crane.

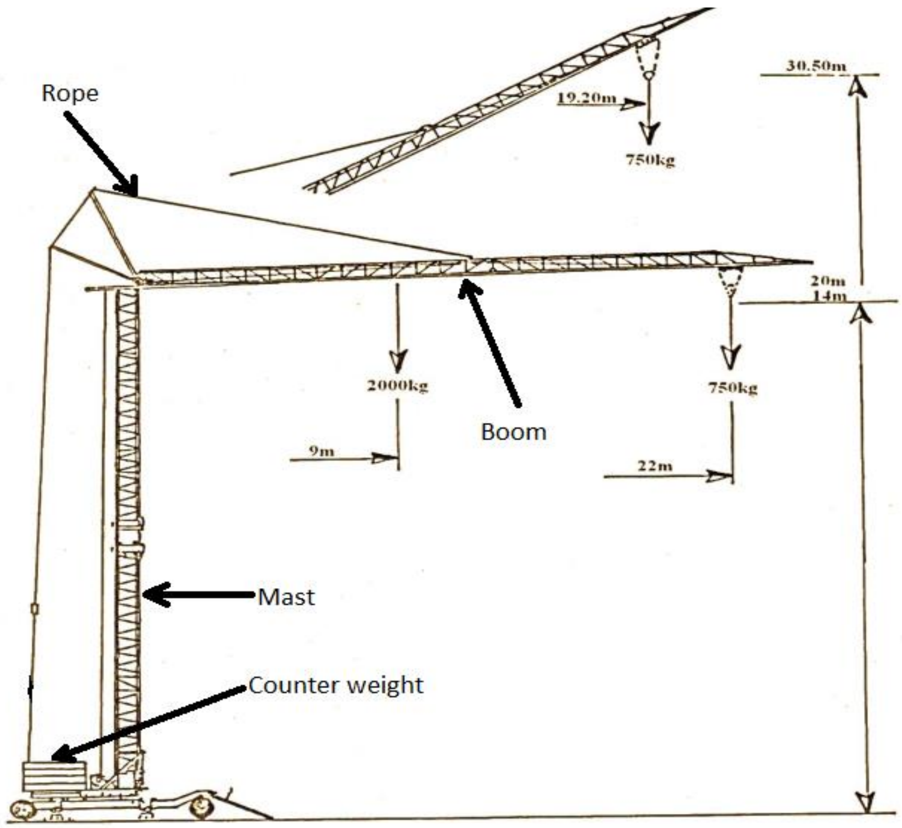

Figure 7. Complete schematic description of the crane ENMTP type GMR 2010.

The load application takes into account the maximum allowable loads given by the manufacturer for each lifting configuration. For example, if a load has to be carried at 9 m of distance from the mast, the allowable load is 2000 kg. At 22 m from the mast, the allowable load is 750 kg. Figure 9 shows a diagram representing the maximum allowable load with respect to the distance from the mast.

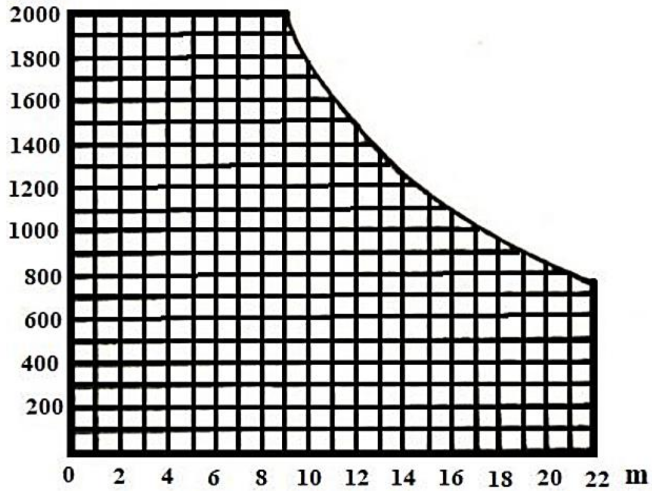

Figure 8. Load allowed to be lifted by the crane depending on the distance to the mast.

Two numerical studies were carried out with two loading configurations: 2000 kg and 750 kg applied respectively at 9 m and 22 m distance from the mast. The results have been carefully analysed in order to identify mostly stressed welded zones, with aim to perform the fatigue evaluation on them. Figure 10 shows the results obtained as well as the welded parts chosen for fatigue study and their positions in the structure. Two areas of interest were selected. The first zone concerns a welded assembly between the part of the Jib Tie (attachment part of the steel rope) and a hollow cylindrical tube of the structure (detail a). The second zone gives an assembly welded between two cylindrical hollow tubes of different diameter, type CHS T-joint (detail b). The dimensions of the welded parts are shown in Figure 11.

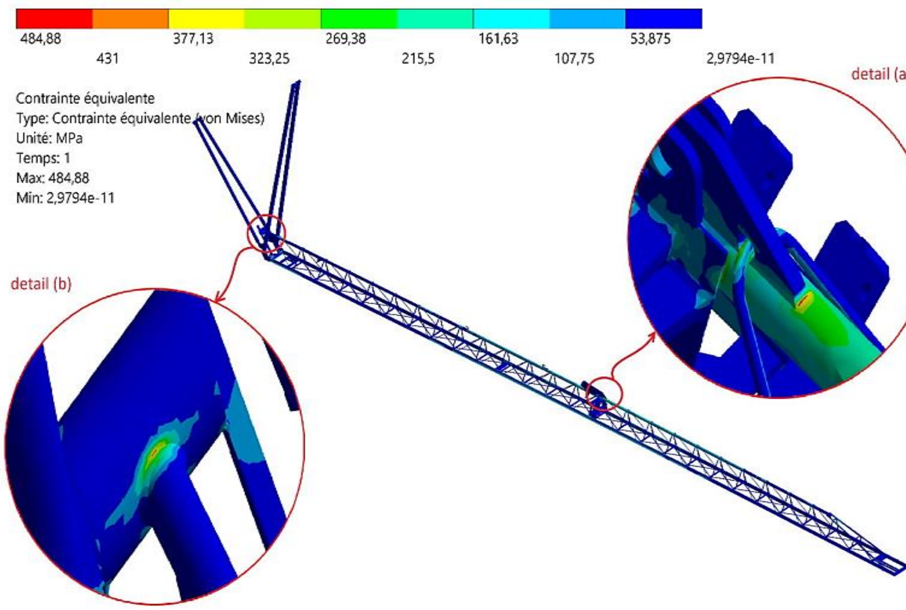

Figure 9. Results of the finite element analysis of the crane boom and location of areas of interest (750 kg load at 22 m distance).

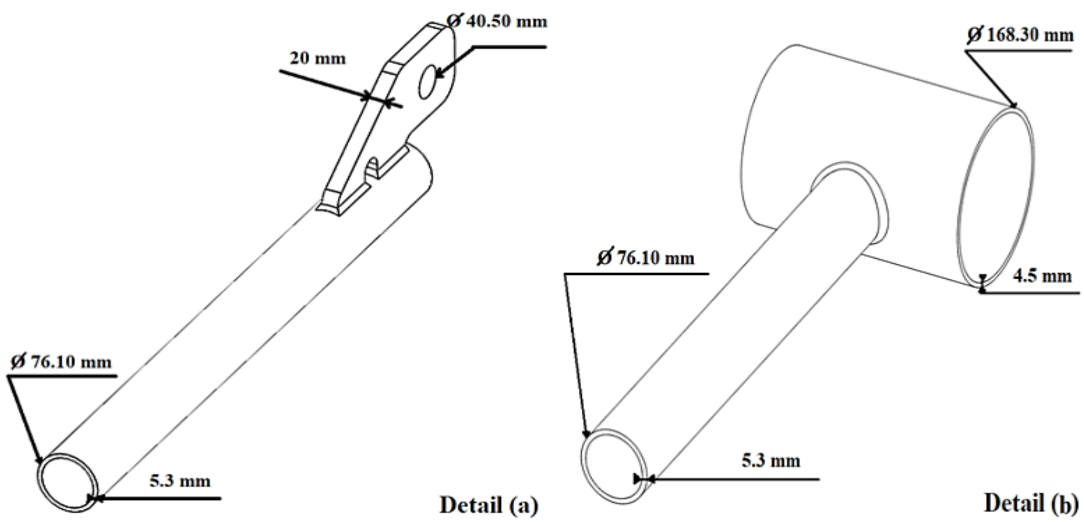

Figure 10. Main dimensions of the selected welded parts for fatigue assessment study.

DETERMINATION OF THE STRUCTURAL STRESS BY FEA

Often, the problems associated with the computer tool, and the time taken for a numerical resolution of large and complex structures, prevent the calculation engineers from using a fine mesh necessary to solve some complex problems which require an absolute precision of the results obtained. In order to overcome these problems, an important technique in the world of numerical simulation called sub-modelling was developed. This allows to define a refined mesh in the zone of interest, i.e. in the sub-model itself. In our case, this will allow us to mesh the welded zone more accurately. Attention was paid to define sufficiently large dimensions around the welded zone of interest, in order to avoid any interference with the boundary conditions. The determination of structural stress is carried out according to three methods.

Determination of HSS with surface stress extrapolation methods

Following the recommendations of Hobbacher [4] and in order to determine the hot spot stress (HSS) according to the surface extrapolation method, the element sizes are to be defined in order to match the reference points as given in Figure 5. For this to avoid an influence of the singularity stress, the stress closest to the hot spot stress is usually evaluated at the first nodal point. Therefore, the length of the hot spot element corresponds to its distance from the first reference point.

A mesh has been carefully designed in such a way that the extrapolation of stresses at the weld toe can be carried out along the most critical extrapolation path. And for that, two mesh size using solid elements to 20 nodes were performed for each sub-model, knowing that the two sub-models have a hot spots (HS) type “a” located on the surface of the tubular parts. Figure 12 and Figure 13 show the mesh of the two sub-models with different sizes.

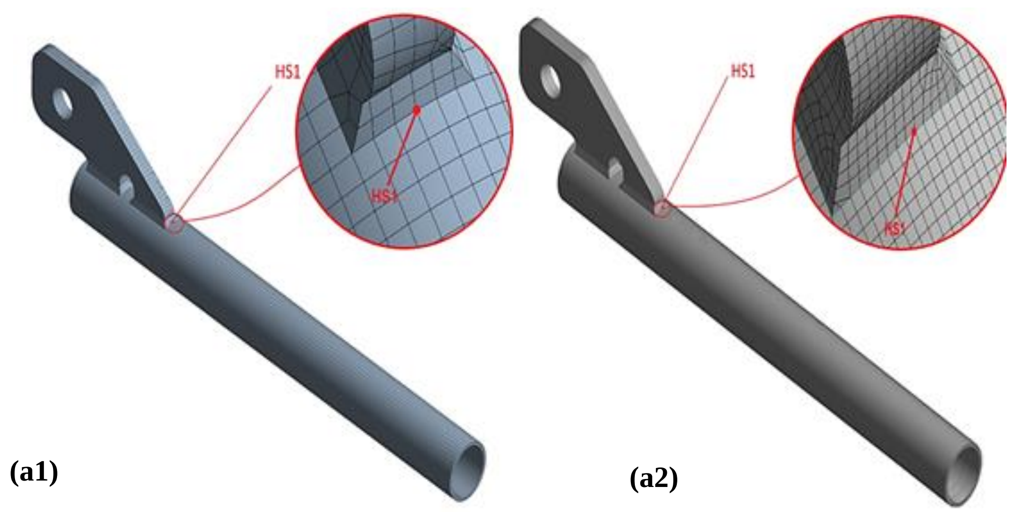

Figure 11. Mesh of sub-model (a) with two different element sizes.

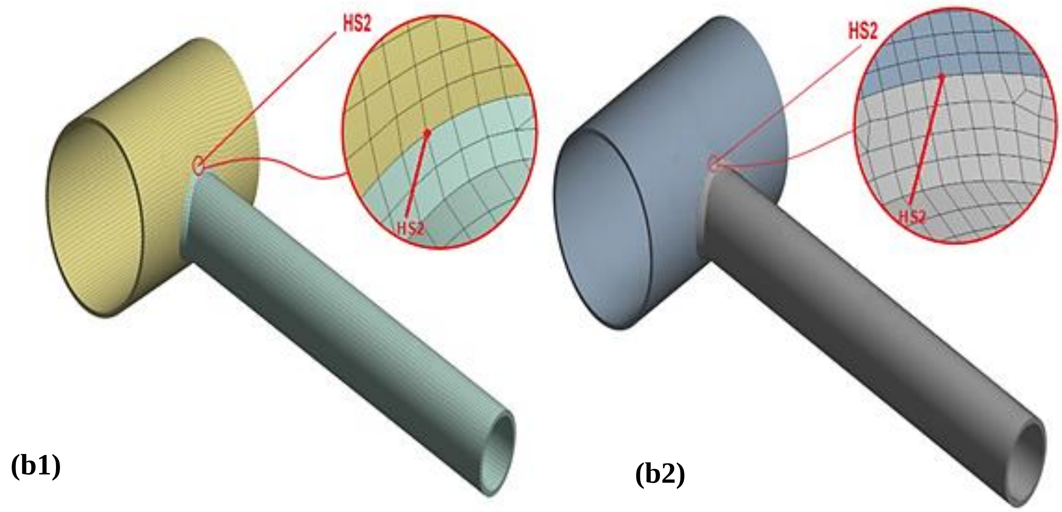

Figure 12. Mesh of sub-model (b) with two different element sizes.

From the results and as shown in Figure 12 and Figure 13, the hot spots: HS1 and HS2 for both sub-models are positioned at the weld toe, which is logical of geometric point of view (see Figure 2). The hot spot of sub-model (a) HS1 causes an eventual cracking which starts from the point indicated in red on Figure 12 and propagates through the thickness of tube. The extrapolation of HSS1 is done according to the recommendations of IIW [4], respecting the distances required for the extrapolation of the nominal stresses to the plane of cracking and which are oriented in the same direction of the loadings. These distances are 0.5 t and 1.5 t for sub-model (a1) in the case of a coarse mesh. They are set to 0.4 t and 1 t for (a2) in the case of a fine mesh. Figure 14 shows the results of the maximum principal stresses obtained from HS1 (maximum tension stresses). Figure 15 shows the normal stress distribution from the hot spot point along X axis as a function of linear distance.

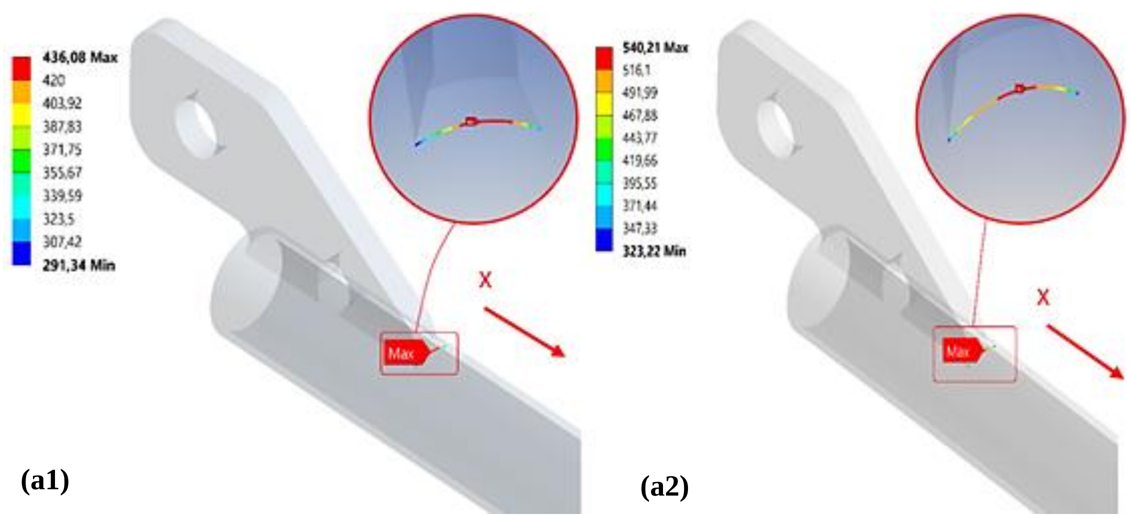

Figure 13. Results of the maximum principal stresses around the weld toe for sub-model (a).

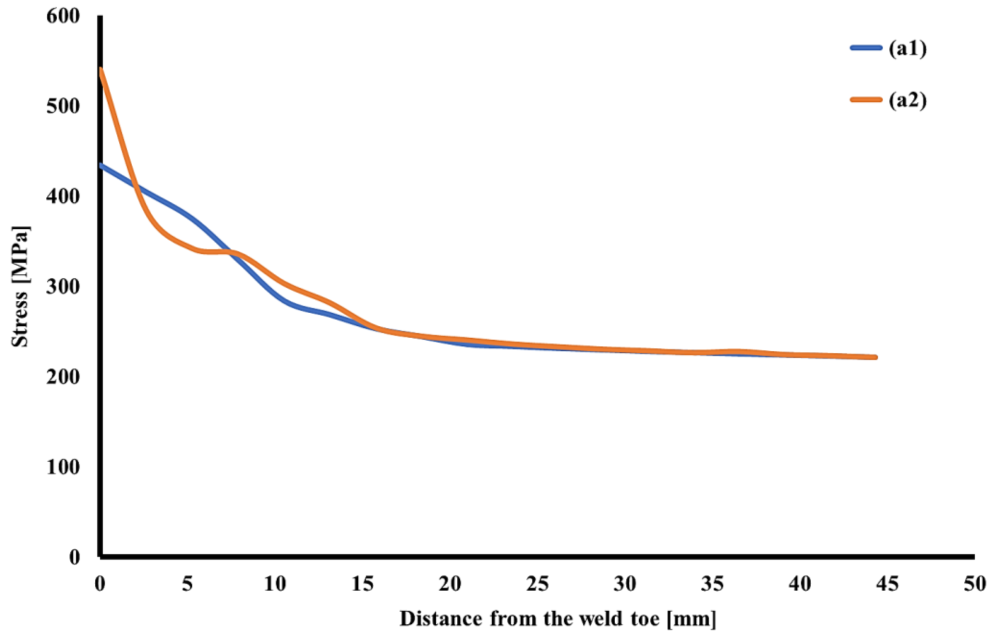

Figure 14. Calculated structural hot spot stress for sub-model (a).

Noted that the maximum principal stresses at the weld toe in this case of sub-model (a) have almost the same direction as the X axis. The second sub-model (b) or CHS T-joint gives more complex results. The HS2 is positioned on the Chord tube surface exactly at the saddle point level. In this case, some technical problems concerning the calculation of HSS2 must be exposed. These problems are related to the application of IIW's recommendations [4], where the rules generally apply to nontubular welded parts. However, the recommendations required for the assessment of the structural hot spot stress for tubular structures [3] do not have clearly defined FE methods. Besides that, the position of HS2 as found in sub-model (b) and which is positioned at saddle point, not at Crown point of Chord tube, complicates the determination of HSS2 numerically.

In order to overcome these problems a numerical resolution technique and a supposition are employed. The numerical technique is based on the determination of the direction of application of the maximal principal stress at the considered hot spot, in order to predict the possible cracking path on the HS2. On the other hand, the supposition stipulates that the surface of the part is approximated to a flat surface, thus taking advantage of the small thickness of tube Chord which is 4.5 mm.

The analysis of the maximum principal stresses orientation in the HS2 and the surrounding area (Figure 16) shows that the direction of these principal stresses are parallel to the surface of Chord tube. Thus, the crack path is expected in the direction of the Chord tube thickness. The intensity results of these principal stresses calculated numerically by FEA around the weld toe are represented in Figure 17.

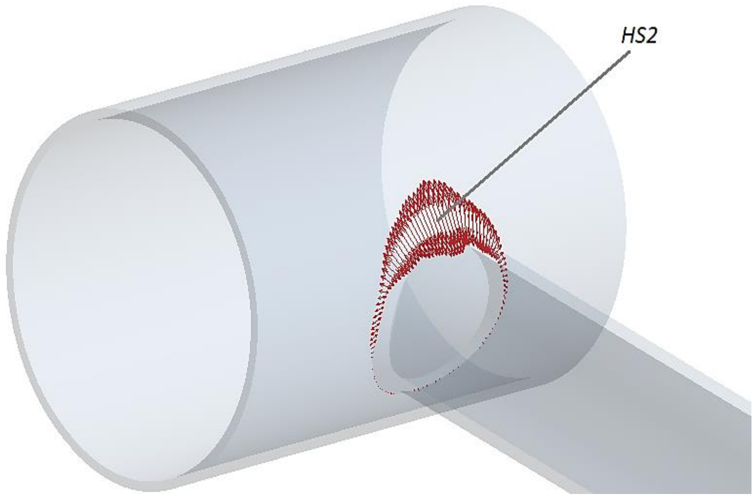

Figure 15. Distribution and orientation of the maximum principal stress around the weld toe of sub-model (b).

The difference in peaks stress recorded between (b1) and (b2) is related to the size of elements, from where it should be emphasized that the stress at weld toe is theoretically infinite for a negligible weld toe radius (ρ ≈ 0 mm). The determination of HSS2 is always done by the surface extrapolation method, for this the HSS2 is calculated by use of Eqn. (1) and Eqn. (3) in the same way as HSS1 thus taking advantage of the size of the elements and the thickness of tube Chord in a way to maintain the supposition already exposed. Figure 18 shows the distribution of normal stresses from weld toe and along the Y-axis that is perpendicular to the crack path as a function of the linear distance to HS2.

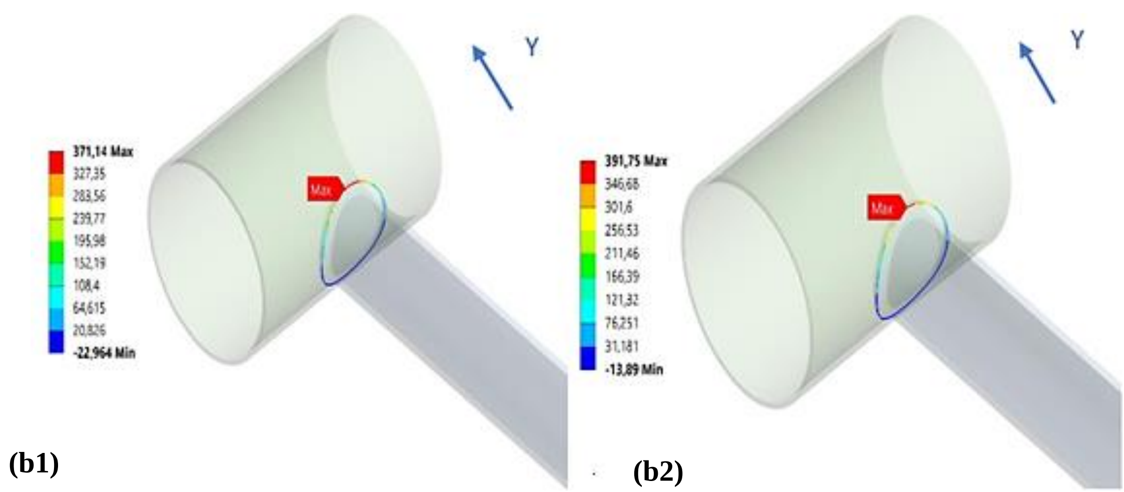

Figure 16. Results of maximum principal stresses around weld toe, for sub-model (b).

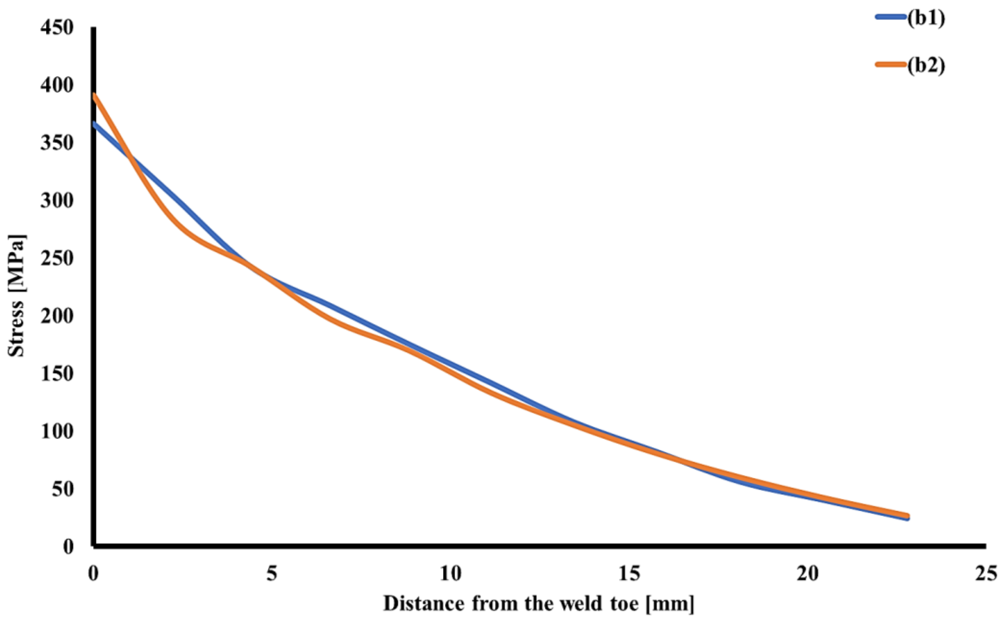

Figure 17. Calculated structural hot spot stresses for sub-model (b).

The results obtained from each case of HS1 and HS2 are shown in Table 2. A remark must be made before starting the analysis of comparisons. This remark concerns the values of HSS1 and HSS2 which are greater than the yield stress of the base metal used in the structure of the crane boom (235 MPa). Additional information is already exposed concerning this point by Lee et al [13]. The authors indicate that in order to assume an elastic behaviour of the material, the stress range of the hot spot found in elastic analysis should not exceed twice the yield strength of the weakest material. Indeed, the results found for HSS1 and HSS2 are not greater than twice the yield stress of the base metal.

Table 2. HSS values.

		HSS1 [MPa]	HSS2 [MPa]
Sub-model (a)	a1	443.48	-
	a2	437.49	-
Sub-model (b)	b1	-	329.55
	b2	-	339.11

As was found on the values of Table 2, a small difference is recorded between the values of the hot spots stresses of sub-models (a1) and (a2) and between sub-models (b1) and (b2). This slight difference between the values of the stresses at weld toes is mainly related to the sensitivity to the size of the mesh.

On the other hand, a remarkable difference of about 100 MPa is recorded between the sub-model (a) and the sub-model (b). This allows to confirm that the detail studied in the sub-model (a) is more critical for the structural integrity of the crane (GMR 2010) with regards to fatigue damage.

Determination of the structural stress according to Dong

The determination of the structural stress according to the method of Dong as described in [19,23–26], is mainly defined as the sum of the membrane stress (σ_m) and the bending stress (σ_b). The procedure itself is based on an equilibrium approach. Thus, membrane and bending stresses are calculated by the FEM at a distance δ of the weld toe. Then, by applying the principle of equilibrium of forces and moments on the considered element, membrane and bending stresses are recalculated exactly at the weld toe, allowing to get the structural stress at that point.

In our case, and in order to numerically proceed to the application of the method of Dong, on the two sub-models, the mesh applied on the two assemblies selected is a mesh with hexahedral quadratic elements. The size of the elements is mainly equivalent to the thickness of the welded parts under study, i.e., 5.3 mm for the sub model (a) and 4.5 mm for the sub-model (b). Figure 19 shows the mesh applied to the two sub-models. For 3D solid elements, the elements along the weld toe line, called welding elements, are used for the purpose of calculating structural stresses. The shape difference between the two sub-models, requires a proper procedure for each one of them.

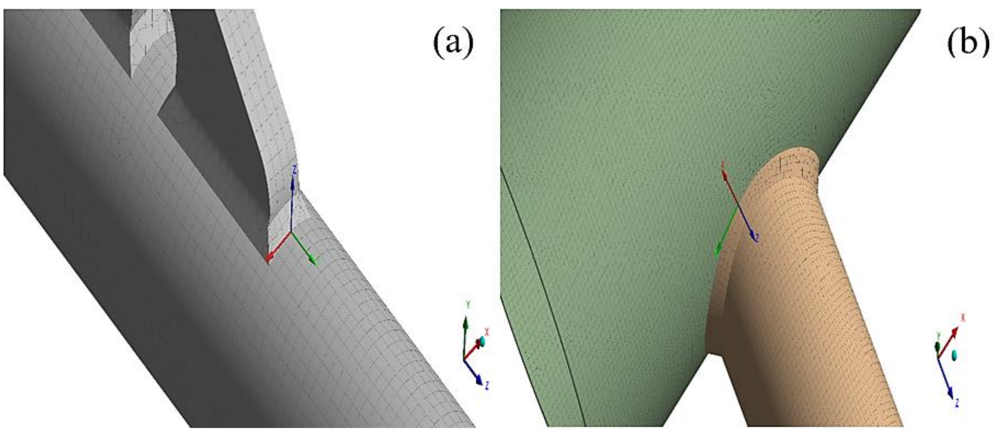

Figure 18. Model mesh and example of a coordinate system located on a weld toe for the Method of Dong.

For example, the sub-model (a) has a straight weld line, while the sub-model (b) has a circumferential weld shape. For this latter case, a system of local coordinates has been set at each node along the weld toe respecting the orientation of the axes along the line of the weld toe. Figure 19 shows two examples of the orientation and location of two local coordinate systems for each sub-model in the normal and tangential direction of the expected cracking plane. An analysis of different types of stress that have been recorded around the weld toe for each sub-model, is given in Figure 20 and Figure 20. This aims to locate the zones that have the maximum probability of crack initiation and growth.

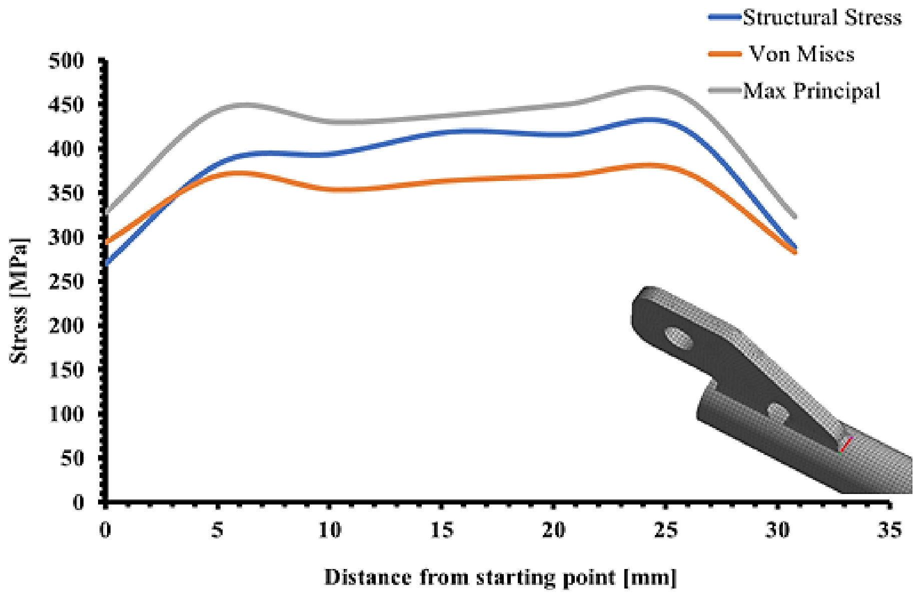

Figure 19. Stress distribution along the weld line for sub-model (a).

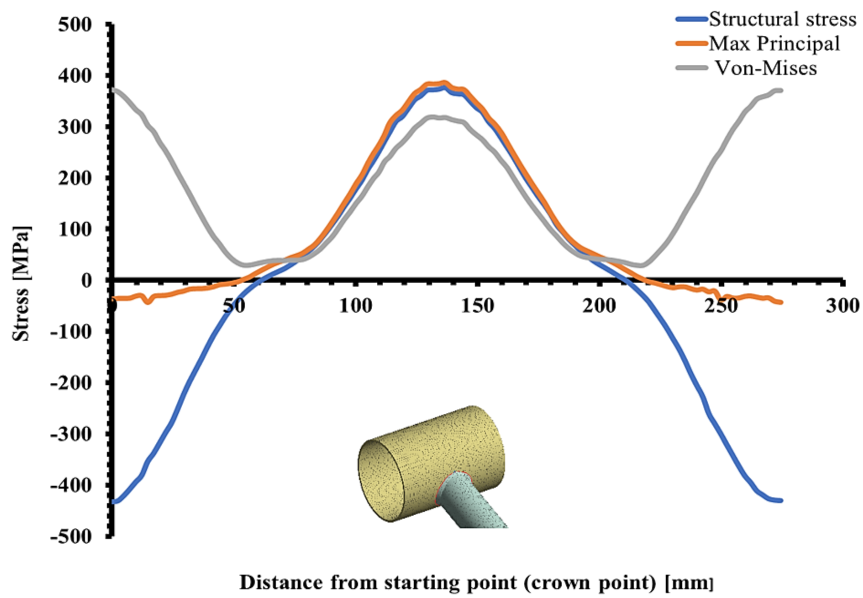

Figure 20. Stress distribution along the weld line for sub-model (b).

The results of maximum structural stress of each sub-model are presented in Table 3.

Table 3. Results of Dong's approach for the studied sub-models.

	Membrane stress (σ_m) [MPa]	Bending stress (σ_b) [MPa]	Structural stress (σ_s) [MPa]
Sub-model (a)	253.55	171.82	425.37
Sub-model (b)	75.70	299.92	375.62

The results obtained with Dong's method, show a similar tendency as those obtained by surface extrapolation methods. Indeed, sub model (a) is still the most critical as it shows higher stress results.

Determination of structural stress according to Xiao-Yamada.

The determination of the structural stress according to Xiao-Yamada in a welded construction is based on the stress value calculated at 1 mm below the surface in the direction corresponding to the expected crack path. For this, a fine mesh is required to determine the stress at 1 mm below the weld toe. The results of structural stress obtained are presented in Table 4.

Table 4. Results of structural stresses according to Xiao-Yamada.

	Sub-model (a)	Sub-model (b)
Structural stress [MPa]	322.71	194.49

As given in Table 4, the structural stress of sub-model (a) is always greater than the one obtained for sub-model (b). Thus, the Xiao-Yamada method, confirms the tendencies obtained with the two previous methods, showing that sub-model (a) represents the most critical location for potential crack initiation and propagation.

LIFE PREDICTION WITH STRUCTURAL STRESS APPROACHES

The fatigue lifetime estimation of boom crane GMR 2010 requires the estimation of stress concentration zones. In this purpose, the estimation of the lifetime of sub-models (a) and (b) is carried out. The lifetime estimation for the three presented methods (and two related sub-models) is done as follows:

For the surface extrapolation method, the use of fatigue resistance curves, S-N, described in the IIW recommendations for hollow welded tubular joints [3], is mainly related to the nature of the sub-models (a) and (b). The position of the structural hot spot stress is located on the singularity zone between the surface of tubes and weld toe. The lifetime estimation is calculated from the fatigue resistance equations given in [3].

In the structural stress approach proposed by Dong [19,23–26], Eqn. (7) is used to calculate the equilibrium equivalent structural stress [24,25]. On the other hand, fatigue life integral I $(r)^{1/m}$ was calculated using a polynomial function of the degree of bending (r), assuming load-control conditions (Eqn. (9), ref. [24,25]):

$$\Delta S = \sigma_s / t^{(2-m)/2m} \, I(r)^{1/m} \tag{7}$$

where

ΔS is equilibrium equivalent structural stress.

t is the plate thickness.

r is the degree of bending, such as:

$$r = \frac{|\sigma_b|}{|\sigma_m| + |\sigma_b|} \tag{8}$$

$$I(r)^{1/m} = 0.0011r^6 + 0.0767r^5 - 0.0988r^4 + 0.0946r^3 + 0.0221r^2 + 0.014\,r + 1.2223 \quad (9)$$

$I(r)^{1/m}$ fatigue life integral from crack propagation analysis considering the effects of the degree of bending, elliptical crack front type, and Load controlled conditions.
In order to estimate the number of allowable cycles (lifetime) of the structure (N), Eqn. (10) developed by Dong can be used.

$$\Delta S = C \times N^h \quad (10)$$

Where the parameters used in our case for C and h are [24,25]:

Statistical basis	C	h
Mean	19930.2	
+2σ	28626.5	
-2σ	13875.8	-0.32
+3σ	31796.1	
-3σ	12492.6	

The value of C = 13875.8 corresponding to -2σ is chosen for our calculation. This value is related to a probability of survival of *Ps* = 95%, which is suitable for comparison with the other methods (using FAT curves at 97.7% of survival probability).
In the approach proposed by Xiao Yamada [20], an S-N design curve corresponding to FAT 100 is proposed. This S-N curve allows to estimate the number of allowable cycles (N) corresponding to a given value of structural stress.

The fatigue lifetime results obtained for each sub-model are then shown in Figure 22 and Figure 23.

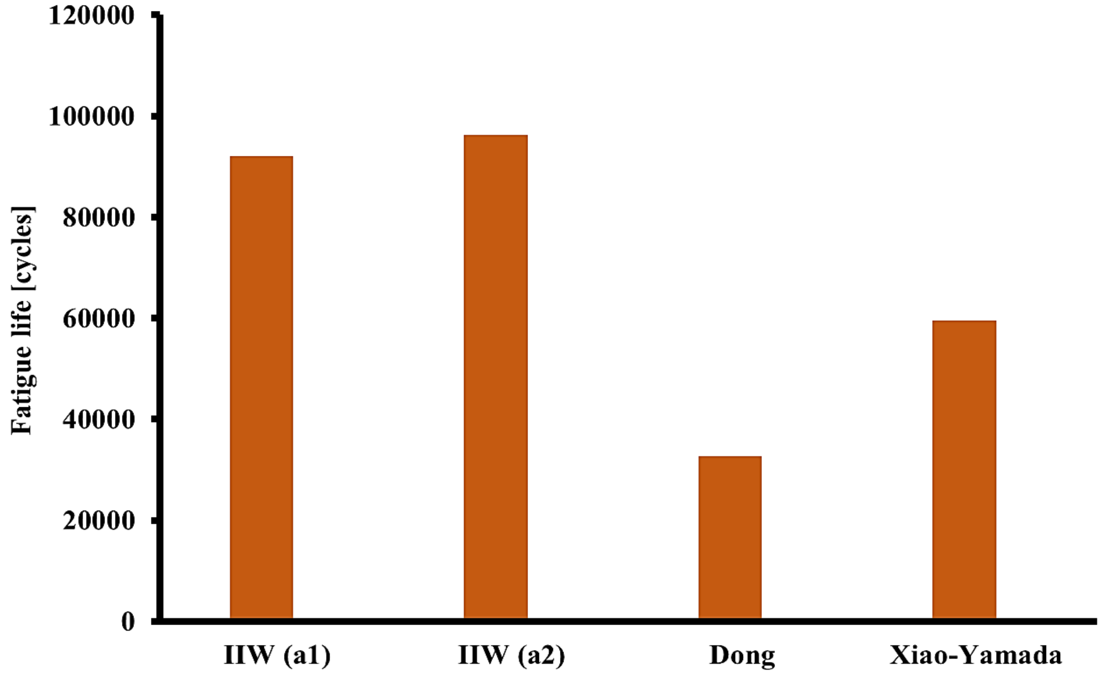

Figure 21. Lifetime results for sub-model (a).

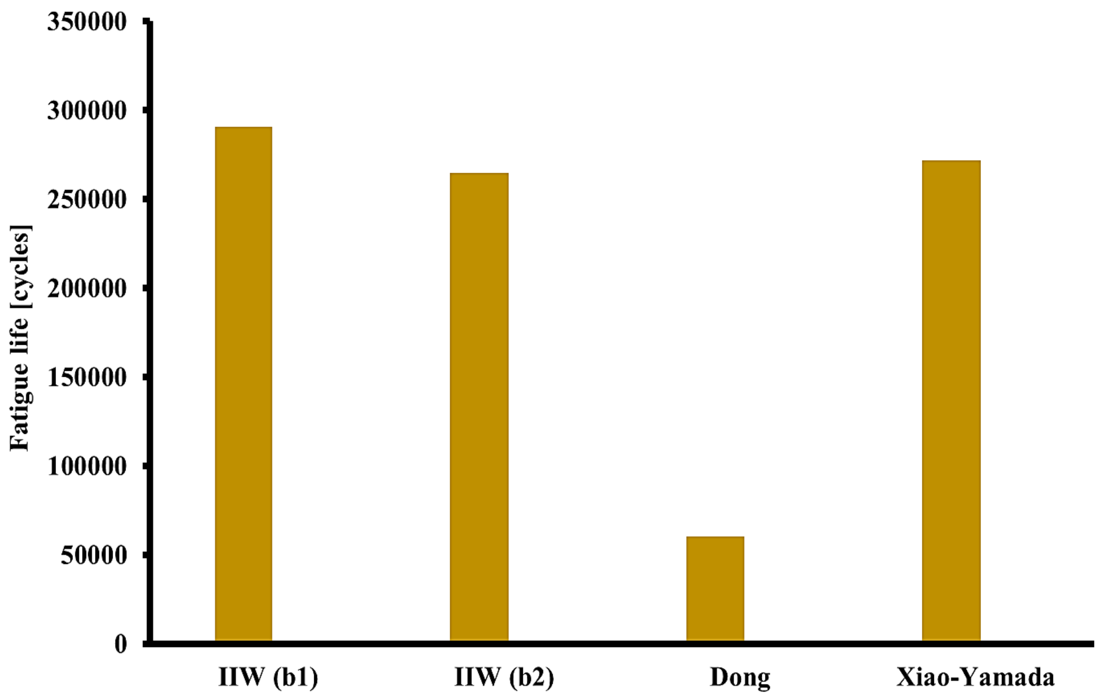

Figure 22. Lifetime results for sub-model (b).

The comparison of fatigue life times obtained, is done in a general way between the results of sub-model (a) and sub-model (b), where an appreciable difference is to be reported. Generally, for sub-model (a) the results are relatively close and ranging from 60 000 to 100 000 cycles, with the exception of the Dong method, where lower results (in the order of 30 000 cycles) have been recorded. The results of sub-model (b) show a considerable variability between the IIW and Xiao-Yamada results on the one hand and the results of Dong on the other hand, Indeed, a remarkable difference of more than 200 000 cycles has been recorded. This is probably related to the nature of the master curve used by Dong. Indeed, Dong et al [26] have built their master curves using parts of thickness varying between 5 mm and 100 mm. The thickness of the studied sub-model (b) is out of this range. Additional experimental tests aiming to build master curves for parts with lower thickness could confirm this assumption. These tests are actually being scheduled and may be reported in a separate article. Finally, the lifetime results confirm that the sub-model (a) is the most critical zone of the crane boom, with a lower number of allowable cycles recorded for all the methods used. Noted that as mentioned above, all life results are compared to S-N curves with similar survival probability P_s.

CONCLUSIONS

In order to estimate the fatigue lifetime of the ENMTP GMR 2010 crane boom, a study was conducted to estimate the service life of the welded zones. A 3D model of the structure was first created using a computer design software. A FE model was then generated including the geometrical and material input data provided by the company's design office. A preliminary analysis of the global structure was performed in order to identify mostly stressed areas. Two critical areas were thus selected and two corresponding sub-models were performed in order to estimate accurately the structural stress at each critical location.

- Three different approaches to calculate the structural stress were used, and compared. IIW approach was used on one hand (with two different mesh sizes), and Xiao-Yamada and Dong approaches were used on the other hand. It should be noted that these two methods are generally presented for very simple structures in the literature. Their use and adaptation to large-scale industrial structures constitutes an innovative work.
- Fatigue lifetimes have been predicted using the design S-N curves recommended by these approaches.
- The lifetime prediction results, were obtained for the two sub-models with three different approaches. They reveal an appreciable difference between the sub-models, confirming that detail (a) is the most critical.
- Relative differences in the results have been observed between the IIW and Xiao-Yamada approaches for the sub model (a). However, the method of Dong gives considerably more conservative results.
- The sub-model (b) shows similar results between the IIW and Xiao-Yamada approaches. On the other hand, the method of Dong shows again more conservative results. This is supposed to be caused by the applicability range of the master curves presented by Dong et al [28], which involve parts of different thicknesses, varying between 5 mm and 100 mm, while the thickness of the parts studied in the present

paper is out of this range. Experimental tests are being scheduled to validate this observation.

REFERENCES

[1] Standards E. Eurocode 3: Design of steel structures - Part 1-9: Fatigue. vol. 1. 2005.
[2] BSI. BSI Standards Publication Guide to fatigue design and assessment of steel products. BSI; 2014.
[3] Zhao XL, Packer JA. Fatigue design procedure for welded hollow section joints: Recommendations of IIW subcommission XV-E. Abington Publishing; 2000.
[4] Hobbacher A. Recommendations for fatigue design of welded joints and components. Second Edition. Springer International Publishing; 2016.
[5] Niemi E. Stress determination for fatigue analysis of welded Components. Cambridge CB 1 6AH, England: Abington Publishing; 1995.
[6] Fricke W. Fatigue analysis of welded joints: State of development. Marine Structures. 2003;16:185–200.
[7] Temarel P, Chen XB, Ergin A, Hermanski G, Hermundstad OA, Hirdaris SE. 17th International Ship and Offshore Structures Congress. ISSC, 2009;1:127–210.
[8] Aygül M, Al-Emrani M, Urushadze S. Modelling and fatigue life assessment of orthotropic bridge deck details using FEM. International Journal of Fatigue. 2012;40:129–142.
[9] Hobbacher AF. The new IIW recommendations for fatigue assessment of welded joints and components - A comprehensive code recently updated. International Journal of Fatigue. 2009;31:50–58.
[10] Radaj D. Design and analysis of fatigue resistant welded structures. Cambridge England: Woodhead Publishing Ltd; 1990.
[11] Radaj D, Sonsino C.M, Fricke W. Fatigue assessment of welded joints by local approaches. Second edition. Cambridge England: Woodhead Publishing Limited; 2006;136.
[12] Poutiainen I, Tanskanen P, Marquis G. Finite element methods for structural hot spot stress determination - A comparison of procedures. International Journal of Fatigue. 2004;26:1147–1157.
[13] Lee JM, Seo JK, Kim MH, Shin SB, Han MS, Park JS. Comparison of hot spot stress evaluation methods for welded structures. International Journal of Naval Architecture and Ocean Engineering. 2010;2:200–210.
[14] Radaj D. Review of fatigue strength assessment of nonwelded and welded structures based on local parameters. International Journal of Fatigue. 1996;18:153–170.
[15] Gorash Y, Comlekci T, Mackenzie D. Comparative study of FE-models and material data for fatigue life assessments of welded thin-walled cross-beam connections. Procedia Engineering. 2015;133:420–432.
[16] Fricke W, Kahl A. Comparison of different structural stress approaches for fatigue assessment of welded ship structures. Marine Structures. 2005;18:473–488.
[17] Kim MH, Kim SM, Kim YN, Kim SG, Lee KE, Kim GR. A comparative study for the fatigue assessment of a ship structure by use of hot spot stress and structural stress approaches. Ocean Engineering. 2009;36:1067–1072.
[18] Niemi E, Fricke W, Maddox SJ. Structural hot-spot stress approach to fatigue analysis

of welded components. Springer; 2018.

[19] Dong P. A structural stress definition and numerical implementation for fatigue analysis of welded joints. International Journal of Fatigue. 2001;23:865–876.

[20] Xiao ZG, Yamada K. A method of determining geometric stress for fatigue strength evaluation of steel welded joints. International Journal of Fatigue. 2004;26:1277–1293.

[21] Kim Y, Oh JS, Jeon SH. Novel hot spot stress calculations for welded joints using 3D solid finite elements. Marine Structures. 2015;44:1–18.

[22] Shen W, Yan R, Barltrop N, Liu E, Song L. A method of determining structural stress for fatigue strength evaluation of welded joints based on notch stress strength theory. International Journal of Fatigue. 2016;90:87–98.

[23] Kyuba H, Dong P. Equilibrium-equivalent structural stress approach to fatigue analysis of a rectangular hollow section joint. International Journal of Fatigue. 2005;27:85–94.

[24] Dong P, Hong JK, De Jesus AMP. Analysis of recent fatigue data using the structural stress procedure in ASME Div 2 Rewrite. Journal of Pressure Vessel Technology. 2007;129:355–362.

[25] Dong P, Pei X, Xing S. A structural strain method for fatigue evaluation of welded components. San Francisco, California, USA: ASME; 2014, p. V005T03A037.

[26] Dong P, Cao Z, Hong JK. Low cycle fatigue evaluation using the weld master S-N curve. ASME Pressure Vessels and Piping Division Conference 2006:1–10.