CILAMCE2014
XXXV IBERIAN LATIN AMERICAN CONGRESS
ON COMPUTATIONAL METHODS IN ENGINEERING
FORTALEZA - CEARÁ - BRAZIL | NOVEMBER 23-26, 2014

A FORMULATION AND COMPARISON OF DIFFERENT SHELL FE MODELING TECHNIQUES FOR FATIGUE LIFE SIMULATION OF WELDED JOINTS

Leonel Echer

Rogério José Marczak

leonel.echer@ufrgs.br

rato@mecanica.ufrgs.br

Mechanical Engineering Department, Federal University of Rio Grande do Sul

Rua Sarmento Leite 425, 90050-170 Porto Alegre, RS, Brazil

Abstract. *A FE shell model capable of representing a welded structure without any significantly error on its stiffness could be widely applied to dynamic problems in which the structural stress method (hot spot approach) is employed for fatigue analysis. The scope of the present work is to formulate a modeling technique capable of doing so. In order to accomplish it, a parametric optimization for simulating welded structures using shell elements is made, the design variables in the proposed formulation are defined as the weld leg length and thickness of the shell element representing the weld fillet. The main goal of the optimization was to find a range of thickness/leg length which would not change significantly the first natural frequencies, and still deliver results similar to the ones obtained by a solid model. Sequential linear programming optimizations are performed in a T-shaped structure, with constant section and different plate thicknesses and depths. Once the optimal parameters are found, two different modeling techniques are presented and compared with three well established methodologies presented in standards and the literature. The differences in the results are compared for first natural frequencies, total mass, hot spot stress and fatigue life.*

Keywords: *Hot spot, Parametric Optimization, Weld fillet, Shell element.*

1 INTRODUCTION

After the great demand for ordnance and ships during the World War II, welding was consolidated as the major fabrication process employed at naval field. This resulted in an increasing interest on evaluating the fatigue life/strength of welded structures (Cary, 1979). Moreover, several normative entities and classification societies began to invest time and funds in order to accomplish a reliable method capable of doing so (ABS, 2012).

As presented by Radaj et al. (2006), the local stress field (which is responsible for crack initiation and propagation) on the surroundings of discontinuities as weld toes has been subject of several studies since the 1960's. The first attempts to evaluate fatigue strength using local strain data, measured a certain distance away from the weld toe (hot-spot), were performed by several researchers, as Erwin Eugen Haibach, Samuel Stanford Manson and Rudolph Earl Peterson.

The development of the structural stress method (hot-spot approach) occurred during the 1970's. It resulted from analysis performed on moderately-thick tubular joints. Those analysis showed that the region affected by the local notch, stress peak, extends from $0.3 \cdot e$ to $0.4 \cdot e$, where e corresponds to the plate thickness. (van Wingerde et al., 1995).

Matoba et. al. (1983) summarizes the attempts of employing the structural stress method to plate (non tubular) structures. These attempts took place in the early 1980's and were applied to ship hull details.

Radaj (1990) defined the structural stress (σ_{hs}) as the surface stress at the weld toe (hot-spot) composed by the summation of membrane (σ_m) and bending (σ_b) stresses. The structural stress were presented as a fictional value with no physical meaning. It was demonstrated that σ_{hs} can be evaluated either by extrapolation or by linearization through the wall thickness. Both procedures has shown to be capable of excluding the stress peak (highly nonlinear) caused by the geometric discontinuity.

After the early 1990's, the structural stress method gained high employability. Various methodologies for representing weld fillets were presented along the decade. Niemi (1994) presented recommendations concerning different approaches for evaluating σ_{hs}, a methodology for modeling the weld fillet using oblique shell elements were proposed as well.

A technique suggesting the use of rigid link elements in order to represent the weld fillet stiffness were presented by Fayard et. al. (1997). Based on this methodology, Fermr et. al. (1998) proposed a technique using shell elements (Volvo method). This technique was initially formulated for thin shell structures and later was extended for structures with moderate plate thickness by Fransson and Pettersson (2000).

A methodology for representing the stiffness of the weld fillet without geometrically modeling the fillet were presented by Eriksson and Lignell (2003). It can be accomplished by imposing an increased thickness for the elements in the intersection of different plates, where the weld fillet would exist in the real structure.

Dong (2001) proposed two different methodologies (solid and plate/shell elements) based on the structural stress definition presented by Radaj (1990). This techniques are known as the Battelle method. The author claims mesh insensitivity, although only simple study cases, most 2D, were presented (Fricke, 2003). It was demonstrated by Niemi and Tonskanen (2000) that these methodologies fail when representing complex geometries where the transverse shear stress have a greater influence. Measures aiming to circumvent this problem were suggested by Fricke (2003).

Combined effort of different normative entities along with the initiative of the International Institute of Welding, IIW, resulted in a series of recommendations concerning the use of the local approaches for fatigue design of welded components (Hobbacher et al., 2009). This and other recent publications concerning fatigue analysis of welded components, clarified several problems faced when creating finite element models of such structures. The correct selection of extrapolation procedure and δ distances can be easily chosen depending on the weld type. Nevertheless, there is still no consensus about the best methodology for stiffness representation of welded components. This work aims to propose a methodology capable of representing the stiffness of such structures in the most reliable way.

2 STRUCTURAL STRESS METHOD

One of the most employed approaches for evaluating the fatigue life/strength of welded components is the structural stress method. This method is most applicable to structures with complex geometries and/or subjected to complex loading scenarios. The structural detail of such structures seldom allow fatigue life analysis by other local approaches (Fransson and Pettersson, 2000).

2.1 Structural Stress

The structural stress method consists in the evaluation of the surface stress at certain distances from the discontinuity region. Subsequently, it is proceeded with a stress extrapolation to the weld toe. The structural stress, σ_{hs}, compute the stress increase caused by the structural configuration (macro-geometry). The nonlinear stress peak present on the surroundings of the weld is not enclosed by σ_{hs} (Fricke, 2003). Figure 1 illustrates the stress distribution at a discontinuity.

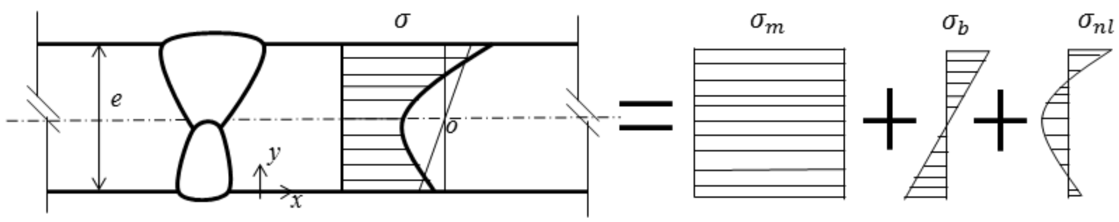

Figure 1: Stress components on the surroundings of a discontinuity [adapted from Hobbacher et al. (2009)].

Once the nonlinear stress, σ_{nl}, component is not taken into account, σ_{hs} can be written as Eq. (1) (Radaj, 1990):

$$\sigma_{hs} = \sigma_m + \sigma_b. \quad (1)$$

Figure 2 presents the surface stress in a welded component. The total stress σ shows the influence of the stress peak, while σ_{hs} ignores it. The structural stress is evaluated according to a linear extrapolation performed over the stresses obtained at two reference points. The spacing between the reference points and the weld toe is defined by δ distances. The lengths of δ are defined depending on the refinement of finite element mesh and hot spot type, as will be discussed later.

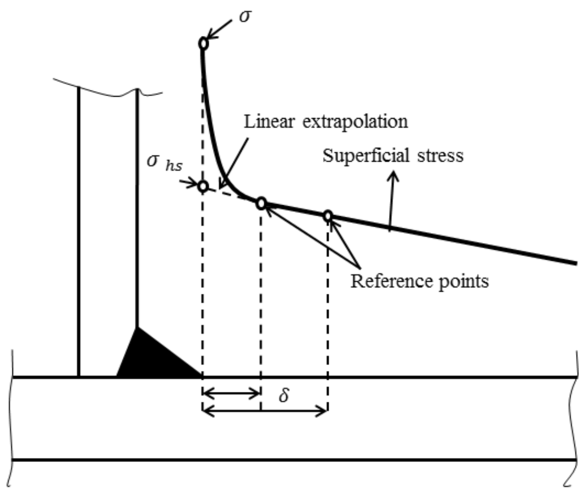

Figure 2: Structural stress extrapolated through δ distances from the weld toe.

2.2 Types of Hot Spots

According to Fricke et. al. (2001), depending on the geometrical location of occurrence of the hot spot, three different types of hot spots can be defined (Fig. 3).

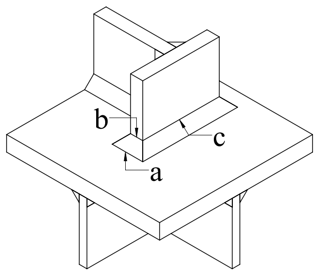

Figure 3: Different types of hot spots according to Fricke et. al. (2001).

If the hot spot occurs at plate edges it is classified as type b). Types a) and c) are employed when the hot spot appears at the connection of the weld toe with the plate surface. The type of hot spot is necessary for determining the position of the reference points for stress evaluation. Depending on the type of hot spot and the refinement of the finite element (FE) mesh, different extrapolation procedures are employed.

2.3 Stress Extrapolation

The extrapolation procedure is employed in the structural stress method aiming to exclude the nonlinear (stress peak) parcel of the total stress from the structural stress. While analyzing components with symmetric weld fillets, i.e., equal leg lengths, the first principal stress (σ_1) is the stress evaluated. However, if the component stress perpendicular to the weld face ($\sigma_\perp$) has an angle ϕ smaller than $60°$ with the plate surface, then $\sigma_\perp$ should be evaluated and employed as σ_{hs} (Hobbacher et al., 2009). These scenarios are illustrated by Fig. 4.

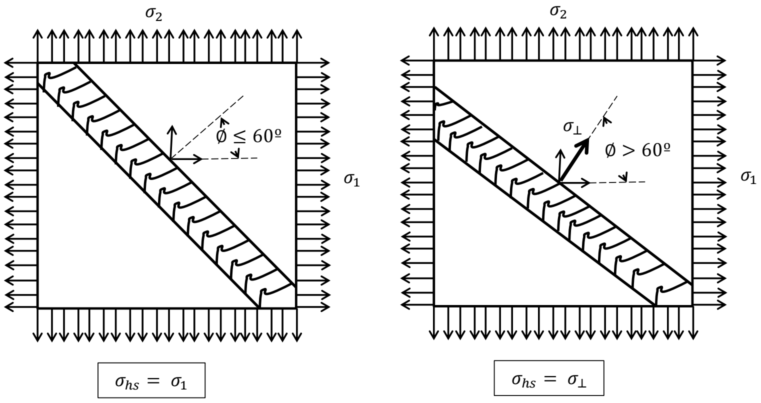

Figure 4: Structural stress dependency on the angle ϕ [adapted from Hobbacher et al. (2009)].

After evaluating σ_{hs} at the reference points, it is extrapolated to the weld toe. Usually, linear extrapolations are used, nevertheless, higher order extrapolation can be applied depending on the type of hot spot and the refinement of FE mesh. Figure 5 presents different types of extrapolations. The selection of distances δ for the reference points is presented as well.

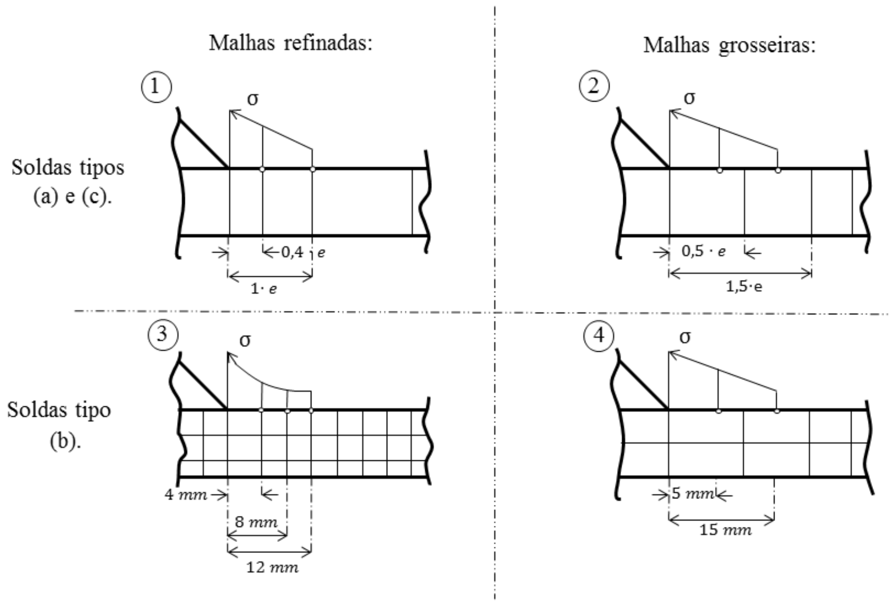

Figure 5: Different δ distances and extrapolation procedures [adapted from Hobbacher et al. (2009)].

Considering a scenario where a component with a symmetric weld fillet is modeled with refined mesh, the extrapolation equation can be written as following. For refined mesh, one can consider the relation of element size smaller than $0.4 \cdot e$.

$$\sigma_{hs} = 1.67 \cdot \sigma_{0.4 \cdot e} - 0.67 \cdot \sigma_{1.0 \cdot e}, \tag{2}$$

where $\sigma_{0.4 \cdot e}$ is the structural stress evaluated at a distance δ equal to forty percent of the plate thickness and $\sigma_{1 \cdot e}$ corresponds to the structural stress evaluated at δ equal to the plate thickness.

3 WELD MODELING TECHNIQUES

Due to the high computational time required for the simulation of solid models, plate/shell elements are preferred in structural stress analysis. A considerable number of methodologies for weld representation using plate/shell elements were proposed along the last decades by several authors. Nevertheless, there is still no consensus about which one produces the best results for most cases.

Some of the most widely widespread methodologies are discussed below. The methodologies presented were chosen mostly by the fact that its respective author have provided enough data to permit the reliable reproduction of these methodologies.

3.1 Weld Modeling With Shell Oblique Elements

Niemi (1995) proposed the representation of the weld fillet by the use of oblique shell elements. This is one of the first, and probably the most intuitive, methodologies ever presented. It consist in representing both plates and welds with 8 node shell elements. Figure 6 illustrates this methodology.

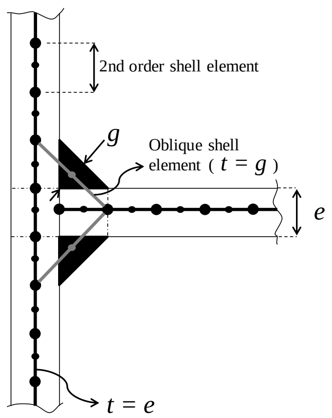

Figure 6: Weld fillet represented by shell oblique elements [adapted from Niemi (1995)].

As illustrated in Fig. 6, the shell element thickness, t, of the elements modeling the weld are equal to the weld throat, g. The rest of the structure is modeled with a shell element thickness equal to the plate thickness, e.

3.2 Weld Modeling With Rigid Link Elements

A methodology where rigid links are employed in order to represent the stiffness of the weld was proposed by Fayard (1997). This technique allows the direct evaluation of the structural stress. σ_{hs} is evaluated directly in the center of gravity, CG, of a shell element. Such element must be positioned to impose the location of its CG in a determined region. This region corresponds to the weld toe occurrence in the real structure. Figure 7 illustrates the methodology presented by Fayard (1997).

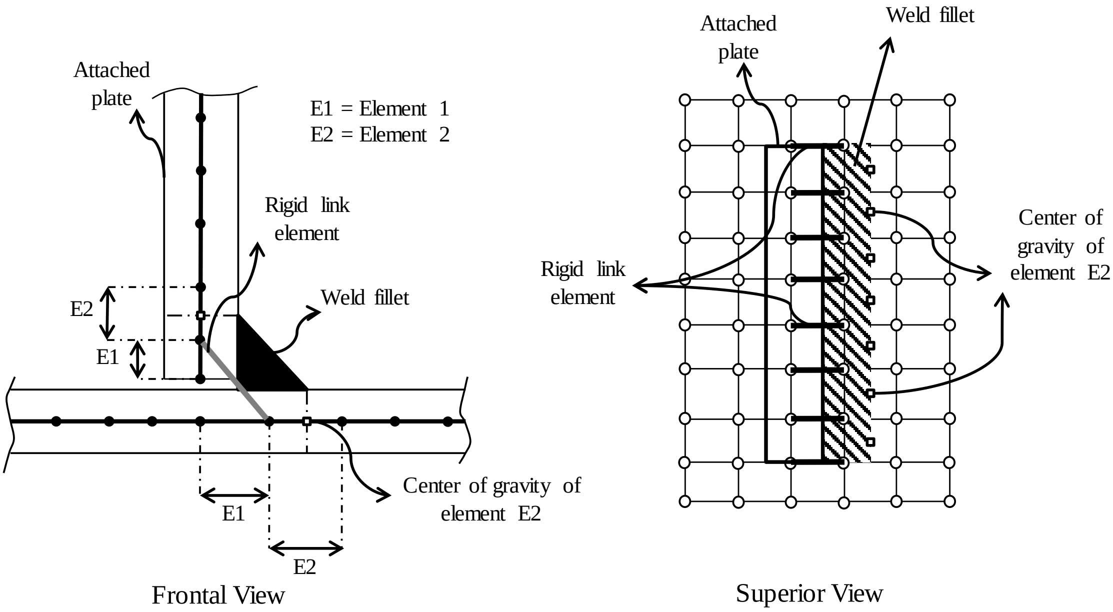

Figure 7: Weld fillet represented by rigid link elements [adapted from Fayard (1997)].

The plates forming the structure must be modeled using 4 node shell elements with the same thickness of the plates. Elements $E1$ and $E2$ must have specific sizes, allowing the stress evaluation directly at $E2$s CG. Such approach discards the use of any extrapolation procedure.

3.3 Weld Modeling Without Weld Geometrical Representation

Eriksson and Lignell (2003) proposed a modeling technique where the stiffness of the weld is represented without representing the weld fillet. It is afforded by the imposition of an incremented thickness. Such representation demands the physical connection of the plates. Figure 8 represents such methodology applied to a cruciform joint.

As illustrated, the elements representing the weld, i.e., actuating on the region of weld occurrence at the real structure, have its thickness increased according to the Eq. (3):

$$t_i = e_i + g, \tag{3}$$

where t_i represents the shell FE thickness and e_i corresponds to the plate thickness.

Such representation of the shell FE thickness is called increased thickness technique. All the plates of the joint should be extended to form a complete intersection. The weld fillet has no geometrical representation (Aygül, 2012).

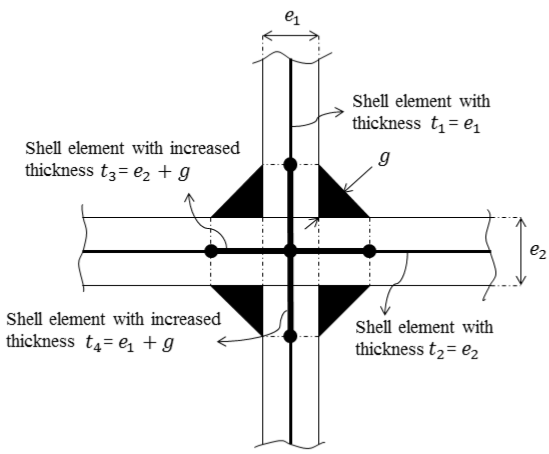

Figure 8: Modeling technique without weld geometrical representation [adapted from Eriksson and Lignell (2003)].

4 PROPOSED METHODOLOGY

The presented methodologies, and the majority of the others, share the same flaw. Those methodologies only are capable of approaching the weld stiffness. A better modeling technique should be able to reproduce its stiffness in a more reliable way.

Unfortunately, a perfect representation is impracticable. But one could perform a modeling technique where geometrical parameters are chosen in such a way that the error with respect to the weld stiffness could be minimized. A modeling technique that achieves this goal is formulated and presented below.

4.1 Geometry of Study

The formulated methodology employs the representation of the weld by oblique shell elements. The parameters chosen to be optimized are the thickness, t, of the FE representing the weld and a distance d. Such distance corresponds to the weld leg length. Figure 9 illustrates the shell FE representation and the parameters to be optimized.

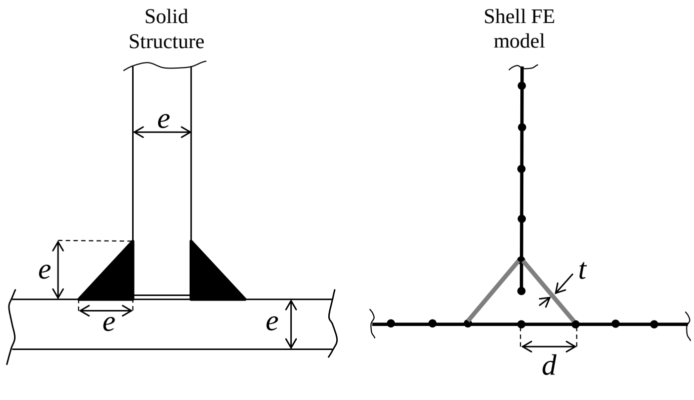

Figure 9: Shell FE representation and parameters chosen to the optimized.

As illustrated, the geometry of study consists on a T-shaped joint. The plates forming the joint have the same thickness. Weld fillets are symmetric and have leg lengths equal to the plates thickness. A generic illustration of the geometry is presented in Fig. 10.

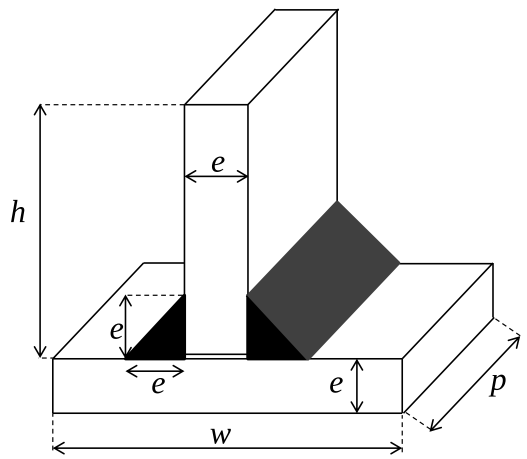

Figure 10: Geometry of study, solid representation.

The height, h, and width, w, are constant for all cases, while the plates thickness and depths, p, assume different values over the cases. 4 different plate thicknesses and 4 different structure depths are employed, totaling 16 cases. All the geometric parameters considered by the simulations are presented in Table 1:

Table 1: Geometrical parameters of the structure

Plate thickness:		Height:	Width:	Depth:
e[mm]	e [inch]	h [mm]	w [mm]	p [mm]
3.175	1/8	254	254	$2 \cdot w = 508$
6.35	1/4			$4 \cdot w = 1016$
12.7	1/2			$6 \cdot w = 1524$
19.05	3/4			$10 \cdot w = 2540$

4.2 Parametric Optimization Formulation

A representation of the structure modeled with solid elements, i.e., idealized T-joint welded component, was employed as reference for the evaluation of the error. The error was evaluated with respect to the structure stiffness. Nevertheless, evaluating the structural stiffness and its respective sensitivity while working with shell theory can be a laborious task. Aiming to avoid it, the error was evaluated in respect to the first natural frequencies of the structure. This is a perfectly reasonable strategy considering that the modeling technique was implemented in such a way to conserve the structural mass. In other words, the shell elements representing the weld employed an artificial density, reproducing the same mass of the solid structure.

The error function can be written as Eq. (4):

$$Err = \sqrt{(freq_1 - f_1)^2 + (freq_2 - f_2)^2}, \tag{4}$$

where $freq_i$ represents the i-th natural frequency of the comparison model, i.e., solid structure idealized representation, and f_i is the i-th natural frequency evaluated with respect to the shell model.

Once the error (objective) function is defined, the constrained optimization problem can be presented according to Eq. (5). The constraint functions with respect to the parameter d are defined considering the physical permissible limits, i.e., plate width w and plate thickness e. While the constraints on t consider as upper limit a shell FE thickness equal to 10 times the plate thickness e, and a lower limit equal to 0.01 times the plate thickness.

$$\begin{aligned}
\underset{t,d}{\text{minimize}} \quad & Err(f_1(t,d), f_2(t,d)) \\
\text{subject to} \quad & t_{max} \leq \frac{w}{2}, \\
& t_{min} \geq \frac{e}{2}, \\
& d_{max} \leq 10 \cdot e, \\
& d_{min} \geq 0.01 \cdot e.
\end{aligned} \tag{5}$$

The starting point for all optimizations was to solve the FE solid model of the structure, in order to retrieve the natural frequencies of the solid structure. Once the reference values of $freq_i$ are known, an initial random estimation for the project variables t and d is made. The FE shell model is then built and solved, natural frequencies (f_i) are stored and Err is evaluated. The sensitivity was evaluated through the classical Euler finite differences method. Finally, the optimization problem was solved with sequential linear programming (SLP). The flowchart illustrated by Fig. 11 presents an overview of the whole optimization procedure.

The stopping criteria employed was a residual error equal or minor then $1 \cdot 10^{-4}$. The increment needed by the finite difference method was defined as $1 \cdot 10^{-5}$. A step length modifier was applied to the actualization of the project variables t and d for each new iteration. Once the new set of parameters were defined by the SLP, the step length was modified according to the two sets of parameters previously defined in past iterations.

4.3 Parametric Optimization Results

The optimum parameters for different plate thicknesses and structure depths are presented in Fig. 12.

These results are overlapped in Fig. 13. The agreement between different results clearly evidences a considerable independence of the project variables with respect to the depths p. A minor disagreement in the behavior of the parameter d is visible when the plate thickness is equal to $\frac{1}{4}$ inch (6.35 mm).

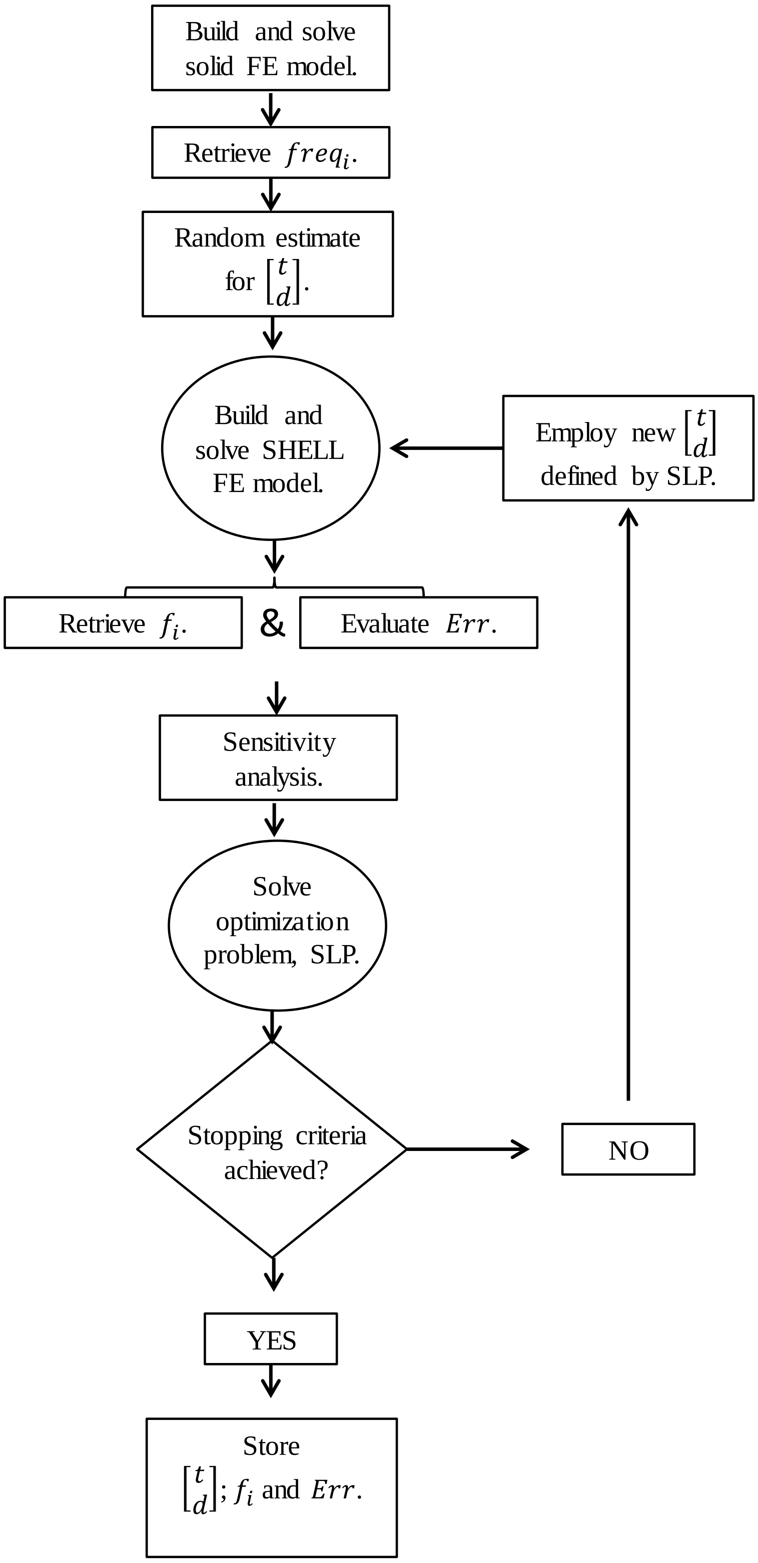

Figure 11: Flowchart for the optimization procedure.

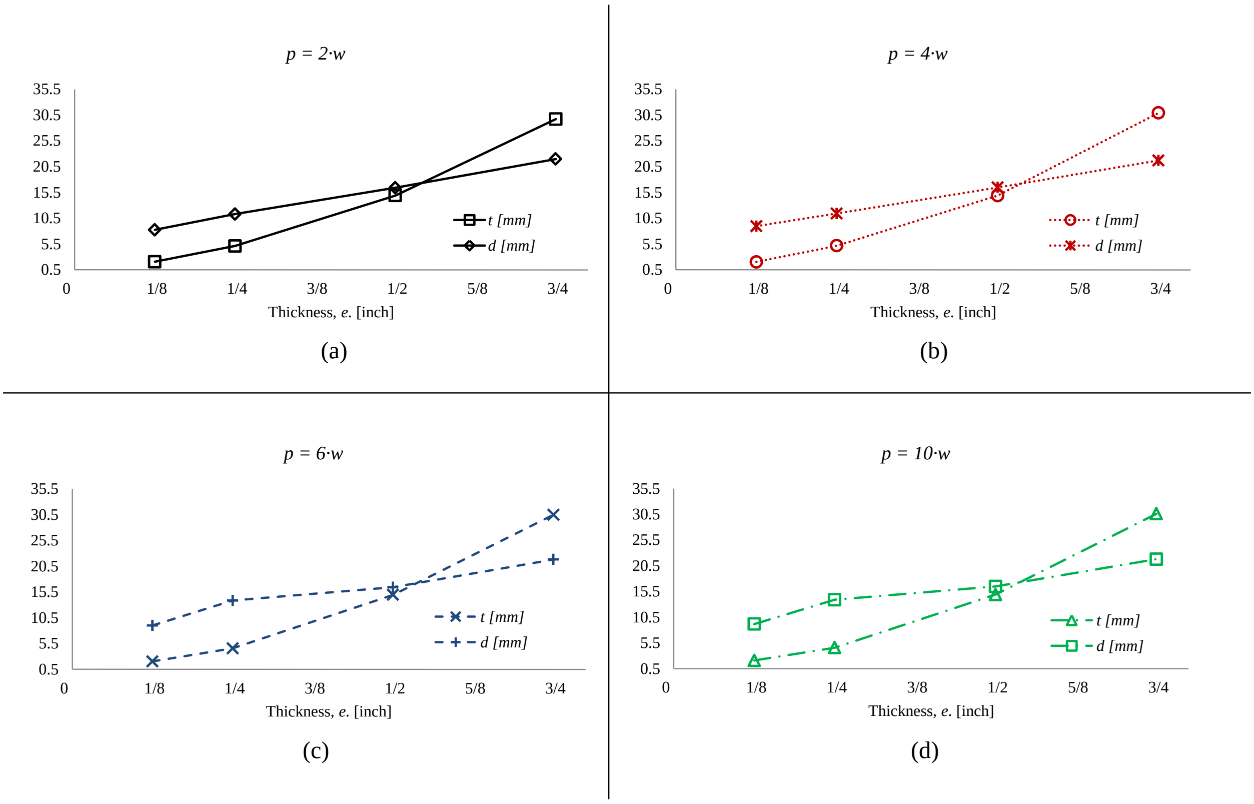

Figure 12: Optimum parameters for different e and p: (a) p equal to 508 mm; (b) p equal to 1016 mm; (c) p equal to 1524; (d) p equal to 2540 mm.

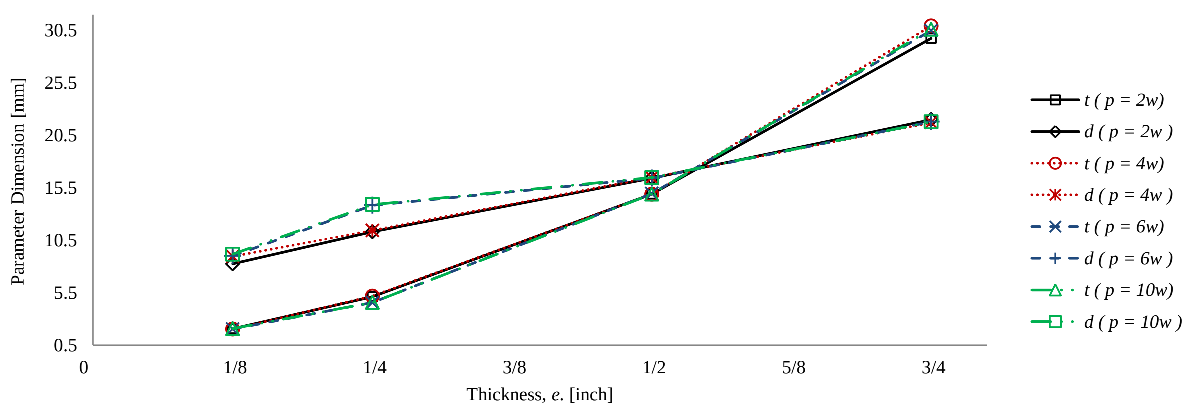

Figure 13: Overlapped optimum parameters.

Once the optimum parameters were evaluated, two different mean values approximation were performed in order to propose two different modeling techniques. The first, and more realistic, considers one mean curve for each parameter and trend lines are traced in order to equate the behavior of t and d with respect to different plate thicknesses e. Figure 14 illustrates proposal I.

Equations (6) and (7) represent the trend lines employed in Fig. 14. These equations rule the first methodology proposed.

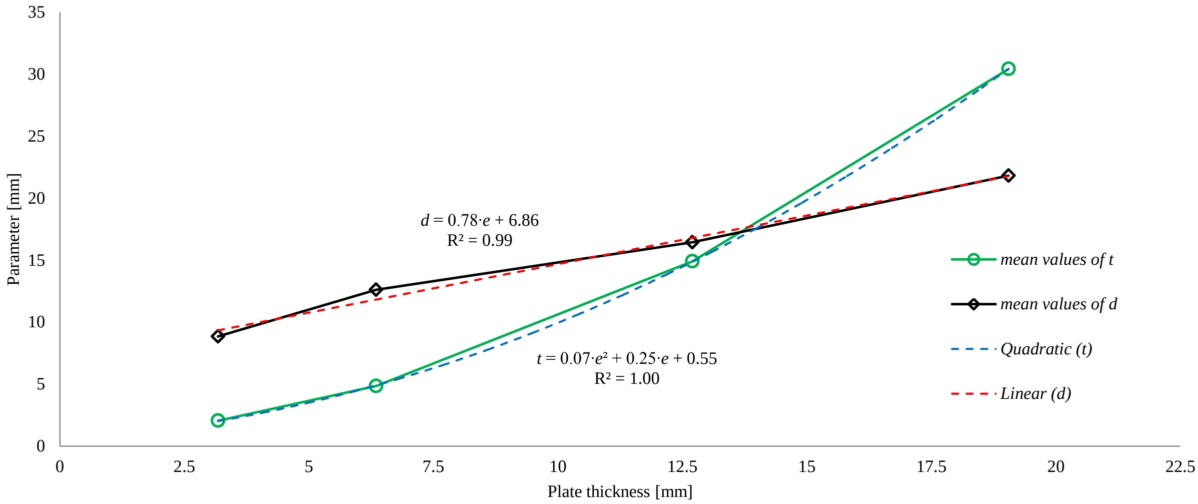

Figure 14: First proposed methodology.

$$t = 0.07 \cdot e^2 + 0.25 \cdot e + 0.55, \tag{6}$$

$$d = 0.78 \cdot e + 6.86. \tag{7}$$

The second proposed methodology considers a scenario where $t = d$, which results in a single curve of mean values. Such curve is then approximated by a trend line. Figure 14 illustrates the second proposed methodology, the trend line and its respective equation are presented as well. The equation of this trend line (Eq. (15)) rules the second proposed methodology.

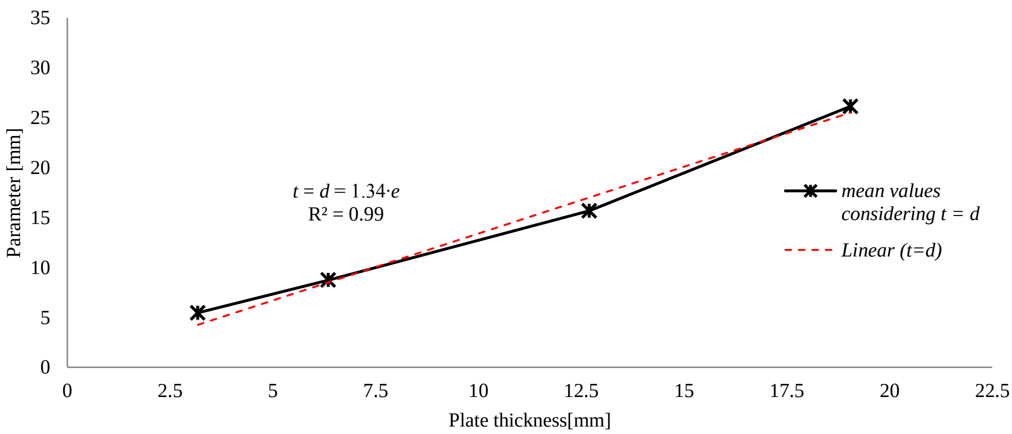

Figure 15: Second proposed methodology.

$$d = t = 1.34 \cdot e. \tag{8}$$

5 COMPARISON BETWEEN METHODOLOGIES

The proposed methodologies were employed along the structural stress method in order to evaluate the fatigue life of a T-joint welded structure. For comparison purposes, the same structure also had its fatigue life evaluated with well established methodologies earlier presented (Niemi (1995), Fayard (1997) and Eriksson and Lignell (2003)).

5.1 Geometry and Loading Scenarios

The geometry of study consists in a T-shaped welded structure with total dimensions equal to 254 mm X 260.35 mm X 508 mm (w X $h + e$ X p). A plate thickness equal to 6.35 mm was considered . Figure 16 illustrates the solid representation of the simulated structure.

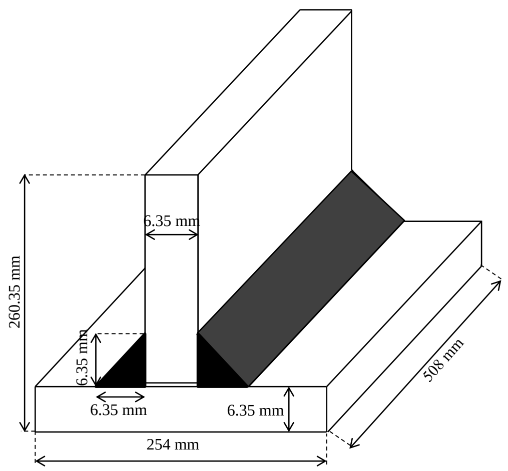

Figure 16: Representation of the structure employed in the simulations.

Two different loading scenarios were considered. Both extremities of the horizontal plate, i.e., sides edges, were clamped. The distributed loads were applied to the upper extremity of the vertical plate for both loading scenarios. The first was a tensile case with total force of $20KN$ uniformly distributed by the top of the vertical plate. The second consist in pure bending with a total force of $2KN$ uniformly distributed. Figure 17 illustrates both loading scenarios.

All five methodologies were implemented with shell FE. The method proposed by Fayard (1997) was modeled using 4 nodes elements, and a mesh density in agreement with the size of the element E2. All the other four methodologies employed 8 nodes shell elements with fine mesh, i.e., element size equal to $0.4 \cdot e$. The FE models, for comparison, were created according to the proposed by its respective authors. Figure 18 presents all the five created models.

The results obtained from the analyzed models were compared with respect to structure total mass, structural stress, and fatigue life.

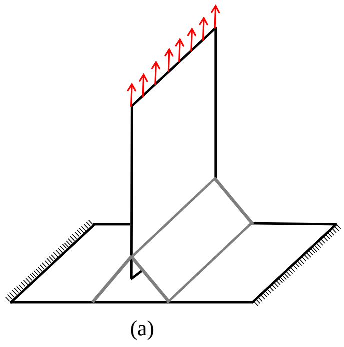

Figure 17: Two analyzed load scenarios: (a) Tensile; (b) Bending.

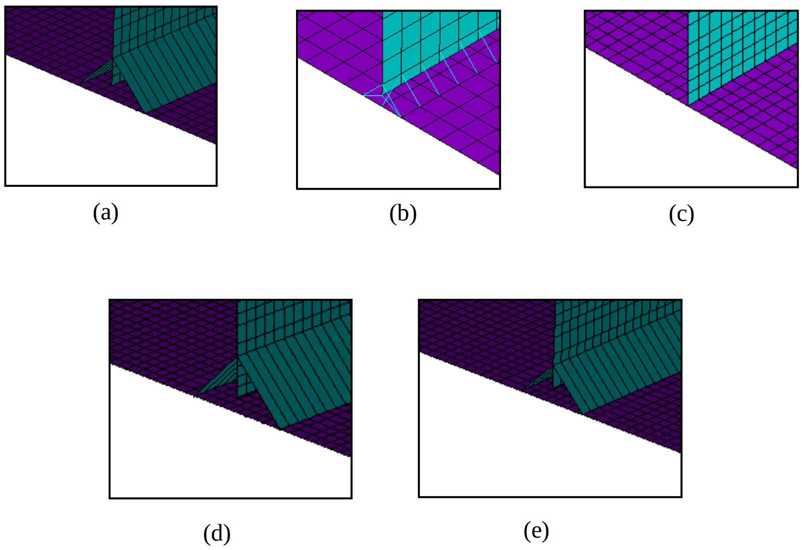

Figure 18: FE models analyzed: (a) Niemi (1995); (b) Fayard (1997); (c) Eriksson and Lignell (2003); (d) Proposal I; (e) Proposal II.

5.2 Total Mass

The error with respect to the total mass of the modeled structure and the mass of the real structure (solid) is presented in Fig. 19. Both proposed methodologies (Proposals I and II) force the conservation of mass. The other modeling techniques implemented does not count with such feature. Nevertheless, the method proposed by Fayard (1997) was implement, in this work, with a density corrector. Since such modeling technique depends directly on the cross sectional area of the rigid link elements, an arbitrary definition of this parameter could lead to vibration modes completely unrealistic. Considering these facts, mass conservation was applied to this model (Fayard, 1997) as well, although the author has not made any mention about such strategy.

5.3 First Natural Frequencies

The error with respect to the first natural frequencies, Eq. (4), is presented in Figure 20. Two different scales for the absolute error axis are presented, in order to provide a better visualization for all results.

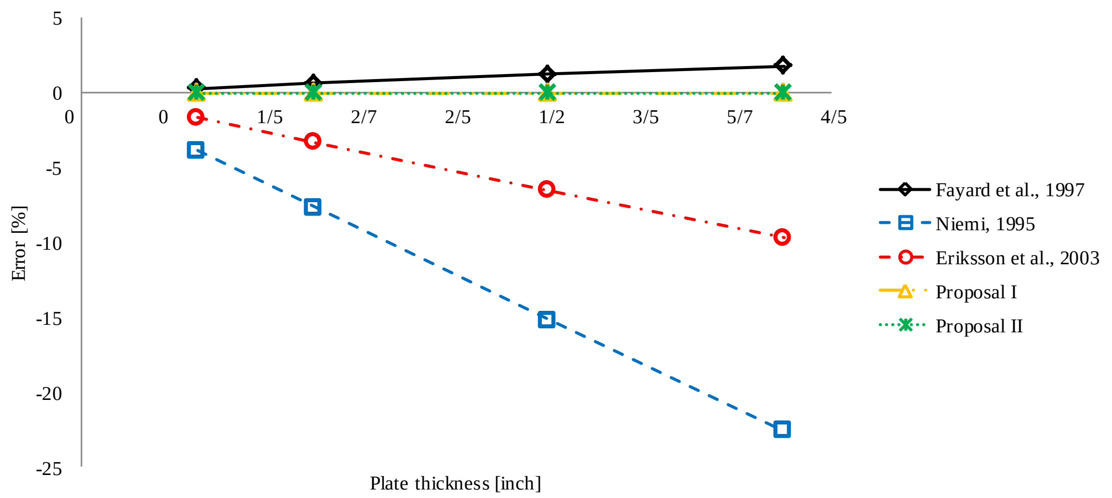

Figure 19: Percentile error with respect to the structure total mass.

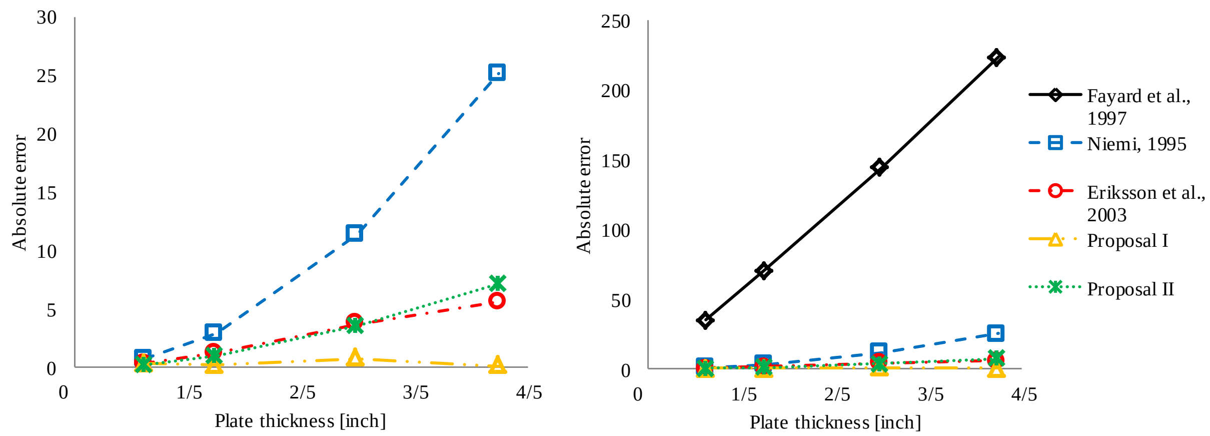

Figure 20: Absolute error, Err.

5.4 Structural Stress

The structural stress was evaluated directly at the elements CG for the methodology of Fayard (1997). All other models had its σ_{hs} evaluated through linear extrapolation as given by Eq. (2). Figure 21 presents a comparison between evaluated stresses.

The methodology proposed by Fayard (1997) was not subject to the pure bending loading scenario. Considering the plates connection performed with rigid link elements, it is not feasible for such structure support bending. This model (Fayard, 1997) was only subjected to the tensile loading scenario.

Table 2 presents the evaluated σ_{hs} for each model.

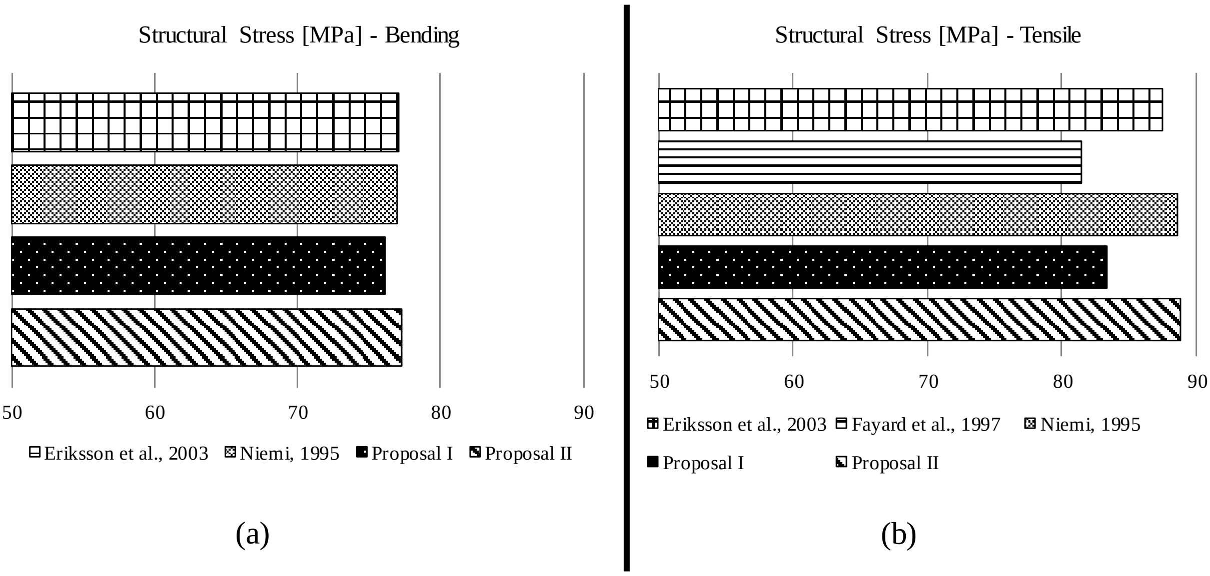

Figure 21: Structural stress comparison: (a) Bending; (b) Tensile.

Table 2: Structural stress comparison [MPa].

Modeling technique	Load case	
	Bending	Tensile
Niemi, 1995	76.99	88.58
Fayard et al., 1997	-	81.52
Eriksson et al., 2003	77.03	87.50
Proposal I	76.14	83.42
Proposal II	77.31	88.79

5.5 Fatigue Life

The component of study was modeled as a steel structure subjected to standard applications, i.e., a scenario where fatigue life correction factors with respect to corrosion, misalignment and others are not required. Whereas the structure consists of a T-joint welded component with load carrying fillet, its fatigue life can be directly determined by FAT 90 curve.

Considering the high agreement of results with respect to σ_{hs}, the fatigue life assessed is practically the same, less than 5% of max deviation, for all methodologies. Such condition is presented in both loading scenarios. The tensile case presents an approximated fatigue life of $4 \cdot 10^6$ cycles, while the bending case presents about $2 \cdot 10^6$ cycles.

6 FINAL REMARKS

The methodology I was capable of accurately reproduce a T-joint welded component stiffness using shell FE. Proposal II had a significantly higher error, max $Err = 5$, however this error is even smaller than those presented by the other 3 tested methodologies.

Both Proposals I and II successfully applied a virtual density aiming the mass conservation of the structure. Such feature can be highly desirable, specially in cases where the weld fillet is responsible for a significant amount of the structure total mass.

Despite the fact that only Proposal I has presented an insignificant error when representing the first natural frequencies of the structure, both methodologies presented structural stresses in accordance with the results obtained by the other well established methodologies.

Concordance of results concerning fatigue life ensure the consistency of the two proposals, nevertheless more tests with more complex geometries and loading scenarios are required in order to test how robust these proposals can be when reproducing the stiffness of welded components.

ACKNOWLEDGEMENTS

The author L. Echer would like to acknowledge the financial support by the National Council for the Improvement of Higher Education (CAPES). The valuable discussions with professor J. S. O. Fonseca must be acknowledge as well, he deserves all the credit and none of the blame.

REFERENCES

ABS, 2012. *The History of the American Bureau of Shipping. 150th Anniversary* American Bureau of Shipping.

Aygl, M., 2012. *Fatigue analysis of welded structures using the finite element method*, Chalmers University of Technology, Gothenburg, Sweden. , Sweden.

Cary, H. B, 1979. *Modern Welding Technology*. Prentice-Hall.

Eriksson, . and Lignell, A., 2003. *Weld Evaluation Using FEM: A Guide to Fatigue-loaded Structures*. Industrilitteratur.

Fayard, J.-L., Bignonnet, A., and Van, K. D., 1997. Fatigue Design of Welded Thin Sheet Structures. *European Structural Integrity Society*, vol. 22, pp. 145152.

Fermr, M., Andrasson, M., and Frodin, B., 1998. Fatigue Life Prediction of MAG-Welded Thin-Sheet Structures. *SAE Conference Proceedings*, pp. 4956. SOC Automotive Engineers INC.

Fransson, P. and Pettersson, G. 2000. *Fatigue Life Prediction Using Forces in Welded Plates of Moderate Thickness*. PhD thesis, University of Karlskrona, Sweden.

Fricke, W. et al., 2001. Recommended Hot Spot Analysis Procedure for Structural Details of FPSOs and Ships Based on Round-Robin FE Analyses. *11th International Offshore and Polar Engineering Conference*, vol. 4, pp. 8996.

Fricke, W., 2003. Fatigue Analysis of Welded Joints: State of Development. *Marine Structures*, vol. 16, n. 3, pp. 185200.

Hobbacher, A. et al., 2009. Recommendations for Fatigue Design of Welded Joints and Components. *Welding Research Council*, IIW.

Matoba, M., Kawasaki, T., Fuji, T., and Yamauchi, T., 1983. Evaluation of Fatigue Strength of Welded Steel Structures-hulls Members, Hollow Section Joints, Piping and Vessel Joints. *IIW-Doc. -1082-83*. International Institute of Welding.

Niemi, E., 1994. Recommendations Concerning Stress Determination for Fatigue Analysis of Welded Components. *IIW-Doc.* University of Technology.

Niemi, E.,1995 Stress Determination for Fatigue Analysis of Welded Components. *Woodhead Publishing*.

Niemi, E. and Tonskanen, P., 2000. IIS/IIW-1477-99 Hot Spot Stress Determination for Welded Edge Gussets. *Welding in the World -London-*, vol. 44, n. 5, pp. 3137.

Radaj, D., 1990. *Design and Analysis of Fatigue Resistant Welded Structures*. Elsevier.

van Wingerde, A. M., Packer, J. A., and Wardenier, J., 1995. Criteria for the Fatigue Assessment of Hollow Structural Section Connections, *Journal of Constructional Steel Research*, vol. 35, n. 1, pp. 71115.