Optimized representation for overlapped welded components using shell FE along with the structural stress method

EngOpt 2016 - 5^{th} International Conference on Engineering Optimization
Iguassu Falls, Brazil, 19-23 June 2016.

Optimized representation for overlapped welded components using shell FE along with the structural stress method

Leonel Echer

Mechanical Engineering Department, Federal University of Rio Grande do Sul
Rua Sarmento Leite 425, Porto Alegre, RS, 90050-170, Brazil
leonel.echer@ufrgs.br

Rogério José Marczak

Mechanical Engineering Department, Federal University of Rio Grande do Sul
Rua Sarmento Leite 425, Porto Alegre, RS, 90050-170, Brazil
rato@mecanica.ufrgs.br

Abstract

The structural stress method is widely applied to fatigue analysis of welded components. The literature contains several modeling techniques capable of representing such structures. However, these techniques can only approximate the stiffness of the structure. The scope of the present work is to propose an optimization-based modeling technique to represent overlapped welded components (lap joints). This technique employs optimal parameters in order to reproduce the stiffness of the real structure without any significant errors. The design variable is defined as the thickness of the shell finite element representing the weld fillet. Linear programming is employed to solve the optimization problem. The objective function is defined as the residual error of the first natural frequencies obtained by a shell finite element model compared with the ones obtained by a solid model. This kind of modeling technique could be directly applied to large/complex problems, where global/local analysis are performed for structural integrity verification and fatigue life simulation. Once, this optimized modeling technique is used, global/local analysis are no longer needed and a single shell FE model can be applied for all the structural analysis. After proposing an optimization-based modeling technique, its result with respect to the structural stress are compared with the ones obtained with other methodologies presented in the literature and the standards. Results concerning errors when representing the structure total mass and first natural frequencies are presented.

Keywords: Structural stress, Shell FE, Lap joint.

1 Introduction

During the early '70s, the need of a tool capable of evaluating the fatigue life of welded components of ships resulted in the development of the, today well known, structural stress method [1, 2]. This method was developed only for moderate thick tubular joints. During the '80s the structural stress method was extended to non-tubular structures such as ship hull details [3]. However, as presented by [4], this method only gained high employability after the groundbreaking work of [5]. In which, the method and the structural stress, σ_{hs}, were formally presented employing a formulation derived from classical mechanical theory.

After the early '90s, the automotive industry started to employ the structural stress method alongside with the Finite Element Method, FEM, leading to the development of different methodologies for geometrical representation of weld fillets. [6] presented recommendations concerning different approaches for evaluating σ_{hs} and a methodology for modeling the weld fillet using oblique shell elements was designed as well. [7] proposed the use of rigid link elements in order to represent the weld fillet stiffness. Based on this work, [8] presented a technique using low order shell elements (Volvo method). [9] represented the stiffness of the weld fillet without geometrically modeling the fillet. It was 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. Based on the structural stress definition presented by [5], [10] proposed two different methodologies: one employing solid elements and the other plate/shell elements. The mesh insensitivity claimed by the author were disproved by [11]. Moreover, it was demonstrated by [12] that these methodologies fail when representing complex geometries where the transverse shear stress have a greater influence.

Due to the high amount of different existing modeling techniques, a combined effort of 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 [13]. This and other recent publications concerning fatigue analysis of welded components clarified several problems faced when creating finite element models of such structures. Nevertheless, there is still no consensus about the best methodology for stiffness representation of welded components. The present work aims to propose a methodology capable of representing the stiffness of this kind of structure in the most reliable way.

2 The Structural Stress Method

The classical formulation of the structural stress method, as presented by [5], consists of evaluating the stress acting on weld regions using extrapolation procedures (or linearization through thickness). The stress is evaluated at certain distances, $\delta_\bullet$ and δ_*, from the weld toe. The obtained stresses are then subjected to an extrapolation rule and the resulting stress is σ_{hs}.

2.1 Extrapolation of σ_{hs}

The extrapolation procedure employed in the structural stress method aims to exclude the non-linear stress peak, which is originated from the discontinuity region, i.e. weld toe. Therefore, the structural stress can be expressed as proposed by [5]:

$$\sigma_{hs} = \sigma_m + \sigma_b, \tag{1}$$

where σ_m correspond to the membrane stress and σ_b the bending stress, both evaluated at the plate surface. Figure 1 illustrates a typical extrapolation of σ_{hs}.

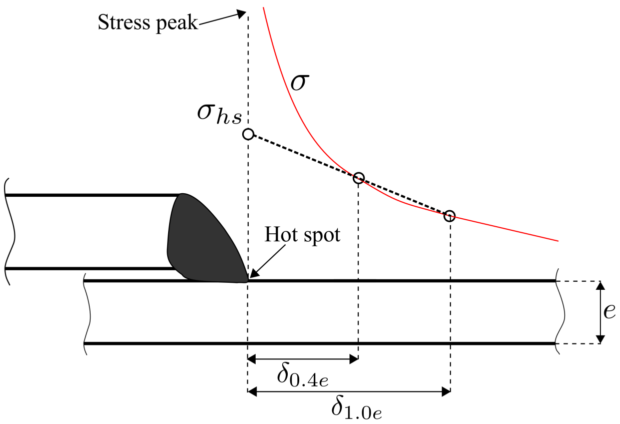


Figure 1: Structural stress extrapolation through δ distances.

In this illustration, the delta distances are considered as being equal to $0.4e$ and $1.0e$, where e is the plate thickness. Such distances are defined in accordance with the work of [14]. In which, different delta distances and extrapolation rules are proposed for different combinations of finite element mesh refinement and hot spot type.

2.1.1 Hot Spot Type

The hot spot type can be defined depending on the geometrical location of its occurrence. As presented by [15], the three most common types are: *(a)*, *(b)* or *(c)*. Types *(a)* and *(c)* are employed when the hot spot appears at the connection of the weld toe with the plate surface. Type *(b)* represents a situation where the hot spot occurs at plate edges. Figure 2 presents different hot spot types in a cruciform welded joint:

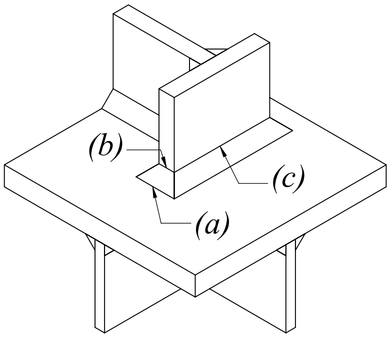


Figure 2: Hot spot types, according to [15].

2.1.2 Employed Extrapolation Rule

The present work employs FE analysis with refined mesh (element length equal to $0.4e$) and considers hot spot of type *(c)*. Such scenario implies in a linear extrapolation rule, presented as [14]:

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

where $\sigma_{1.0e}$ and $\sigma_{0.4e}$ are the stress evaluated at $\delta_{1.0e}$ and $\delta_{0.4e}$ respectively.

3 Problem Formulation

An overlapped welded joint is considered as the component of study. The present work aims to achieve an optimization-based methodology for representing this sort of structure using shell FE. This representation must reproduce the stiffness of the solid model without any significant error. In order to achieve this condition, an optimization problem is formulated and solved using Sequential Linear Programming, SLP.

3.1 Optimization Problem

The optimization problem considers the thickness of the shell element representing the weld fillet, t, as the single design variable. The objective function consists of the minimization of the error function:

$$Error(f_1(t), f_2(t)) = \sqrt{(f_1 - \bar{f}_1)^2 + (f_2 - \bar{f}_2)^2}, \tag{3}$$

where f_i and $\bar{f}_i$ correspond to the i-th frequency of the shell FE model and comparison model (solid), respectively. The optimization problem can be written as:

$$\begin{aligned} \underset{t}{\text{minimize}} \quad & Error(f_1(t), f_2(t)) \\ \text{subject to} \quad & t_{max} \leq 10 \cdot e, \\ & t_{min} \geq 0.001. \end{aligned} \tag{4}$$

where t_{max} is the upper constraint with respect to the design variable and t_{min} is the lower constraint. Geometries with different scales where tested in order to evaluate the influence of parameters as the structure depth, d, and the plate thickness e.

3.2 Geometry

Figure 3 presents the geometry of study alongside with its shell FE representation. The shell representation of the overlapped joint is based on the recommendations proposed by the International Institute of Welding [13]. High order shell FE are employed for modeling weld fillet and the structure (plates). Table 1 presents the geometry configuration of the 16 analyzed cases.

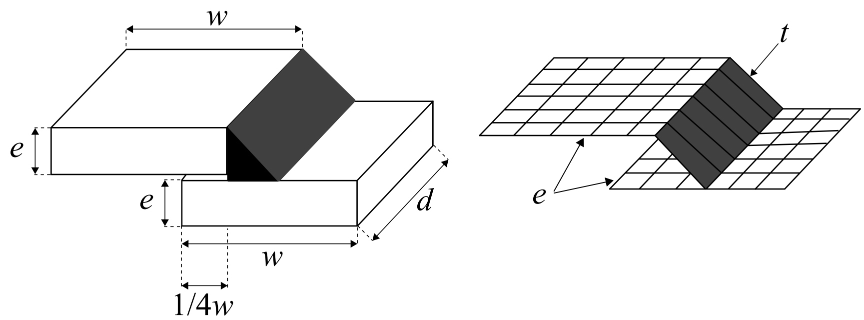


Figure 3: Overlapped joint: solid and shell FE representation.

Table 1: Geometrical parameters of the structure.

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

3.3 Parametric Optimization

The parametric optimization performed for each structure configuration consisted of the following steps:

1. Estimate initial value for parameter t;
2. Build and solve FE model;
3. Evaluate $Error$ and store t and f_i;
4. Check if the current solution satisfies the convergence criterion;
5. If the convergence criterion is satisfied, go to step 10;
6. Employ linear programming to solve the optimization problem and update the design variable t;
7. Use forward finite differences to evaluate the sensitivity;

8. Update the moving limits (step length) using past iterations results;
9. Return to step 2;
10. Store t, f_i and $Error$

The parametric optimization problem was solved employing a commercial mathematical software [16] and the FE problem was solved using a commercial FE software [17]. Initial estimation for t was randomly generated. A step length modifier, based on the derivative sign of past iterations, was applied in the actualization of the design variable for each new iteration. The employed stopping criterion can be written as:

$$Error \leq tol, \tag{5}$$

where tol is the tolerance, which was considered equal to $1 \cdot 10^{-4}$.

4 Results

The optimization results allowed the construction of curves for the optimal values of t considering different geometric configurations (e and d), which led to the design of a ruling curve for t. The shell FE model employing this ruling curve was then implemented and compared with other methodologies present in the literature and standards.

4.1 Optimal Thickness

The optimal t obtained for each geometric configuration is presented in Fig. 4:

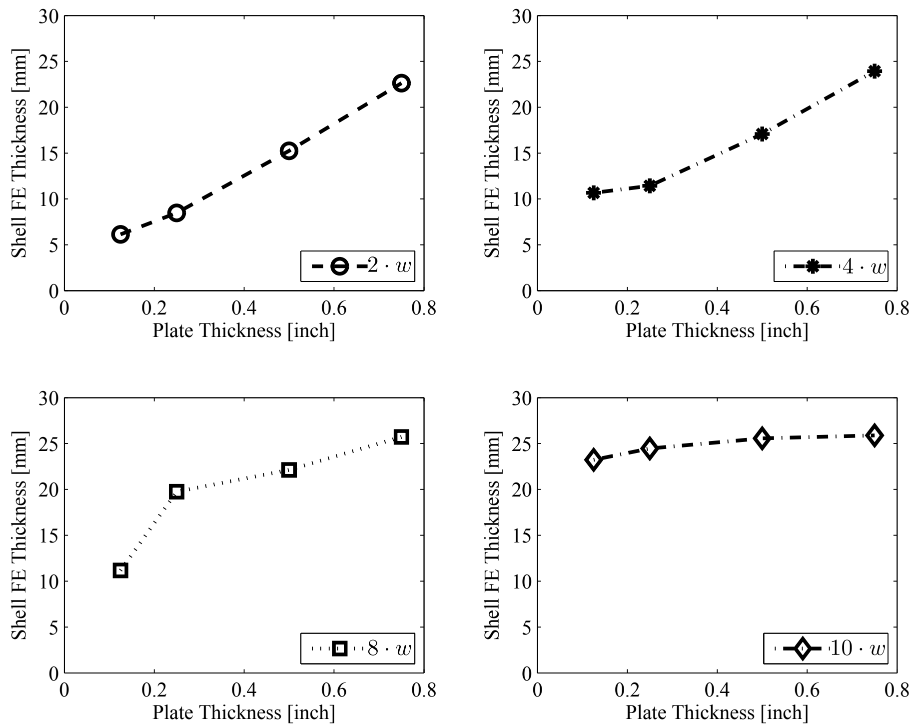


Figure 4: Results obtained for the parametric optimizations, abscissa correspond to e and ordinate to t.

Despite the small, but existing, correlation between results (Fig. 4) for different structure configuration, a least-squares fitting rule was applied in order to obtain a ruling equation for t. The least-squares fit and the overlapped results for optimal thickness t are presented on Fig. 5.

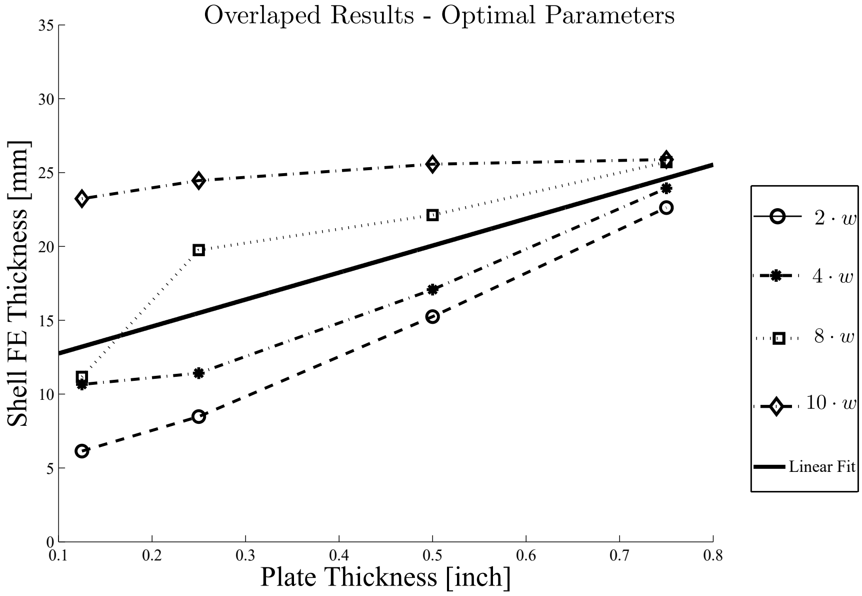


Figure 5: Fitting rule employed.

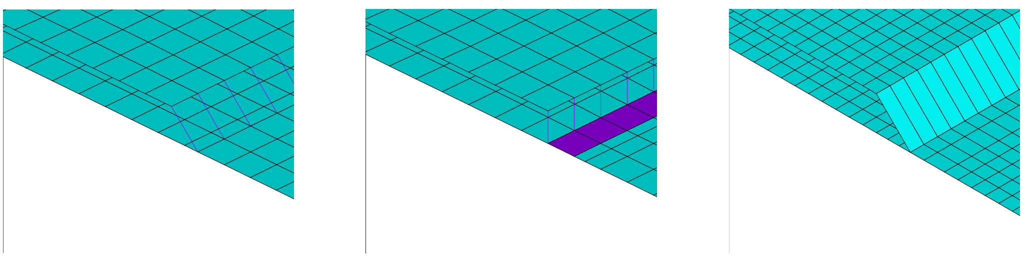
The linear fit resulted in the following linear rule:

$$t = 18.25 \cdot e - 10.93, \tag{6}$$

with a coefficient of determination, R^2, equal to 0.44. This value indicates a poor fitting quality. Notwithstanding, the possibility of determinate t using a simple linear equation (Eq. 6) is very interesting. By this point, this poor fitting quality will be accepted and after comparing results for f_i this discussion will be continued.

4.2 Comparison of Results

The linear equation for the determination of the optimal t for different values of e (Eq. 6) was tested for an overlapped welded component (Fig. 3) with dimensions: $508 \times 254 \times 6.35$ [mm] ($d \times w \times e$). The results were compared with the ones obtained by the methodologies of [7] and [9]. Finite element models employing this two modeling techniques and the one proposed in this work are presented in Fig. 6:



(a) FE model according to [7]. (b) FE model according to [9]. (c) Proposed methodology

Figure 6: Modeling techniques employed for modeling a lap joint.

4.2.1 Total Mass

The proposed methodology employs a virtual density corrector in order to reproduce the mass of the real structure. This implies a reliable representation of the structure mass when using the proposed method. Notwithstanding, the techniques proposed in the literature do not count with such feature, which leads to errors. However, only small percentile differences between results were observed, as presented in Table 2.

Table 2: Structure total mass.

Model	Mass [kg]	Difference to Proposed Methodology [%]
Methodology of [9]	12.89	0.18
Methodology of [7]	12.75	1.26
Proposed Methodology	12.91	-

4.2.2 First Natural Frequencies

Despite the minor difference between methodologies when comparing its results concerning structure total mass, the results for the first natural frequencies and *Error* present major disagreements. As can be seen in Table 3, the methodologies of [9] and [7] were not able to reproduce the structure stiffness in a reliable way.

Table 3: First natural frequencies error.

Model	Result [hz]		Difference to Proposed Methodology
Methodology of [9]	f_1	1.68	0.94
	f_2	3.22	0.95
	$Error$	63.53	132.65
Methodology of [7]	f_1	1.41	0.95
	f_2	2.74	0.96
	$Error$	64.09	133.81
Proposed Methodology	f_1	27.11	-
	f_2	61.9	-
	$Error$	0.48	-

These results indicate that, despite the very poor linear fitting ($R^2 = 0.44$) obtained, the proposed methodology was still capable of reproducing the structure stiffness.

4.2.3 Structural Stress

Regardless the huge differences found when reproducing the structure stiffness, the results for σ_{hs} showed a good agreement among all methodologies, as presented in Table 4:

Table 4: Structural stress evaluated employing different modeling techniques.

Model	σ_{hs} [MPa]	Difference to Proposed Methodology [%]
Methodology of [9]	30.94	0.03
Methodology of [7]	31.73	2.46
Proposed Methodology	30.95	-

This results denote the reliable representation of the stress state of overlapped welded joints by the proposed methodology. Furthermore, it indicates the superiority of the latter when compared to others presented in the literature. The reliable representation of the structure stiffness is highly desirable, specially when modeling large/complex geometries. Employing this optimization-based methodology, one can use the same FE model for both structural integrity and fatigue analysis.

5 Final Remarks

In addition to the reliable representation of weld fillets stiffness, the optimization-based methodology proposed in this work employs a straightforward linear rule for the determination of t. This simplicity, combined with the use of oblique shell FE for the representation of the weld fillet [13], makes the proposed methodology easy to implement. It also should be pointed out that the costly global/local approaches can be replaced by this kind of modeling techniques when representing complex structures composed only by overlapped joints. Since this technique can achieve a trustworthy representation of the structure stiffness, a shell FE model employing it can evaluate both structure integrity and σ_{hs} (fatigue life).

Despite the results obtained, more tests are needed to assess the robustness of this modeling technique. As a matter of fact, the effectiveness of the proposed technique is only suitable to lap joints, which is one of the several existing structural details for welded components.

Acknowledgments

The authors would like to acknowledge the financial support by the Brazilian research agencies, CAPES and CNPq (grant 309676/2013).

6 References

[1] ABS. The History of the American Bureau of Shipping. 150th Anniversary, *ABS - American Bureau of Shipping*, 2012.

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

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

[4] Radaj D, Sonsino C and Fricke W. Fatigue Assessment of Welded Joints by Local Approaches. *Woodhead Publishing Series in Welding and Other Joining Technologies, Elsevier Science.* 2006.

[5] Radaj D.Design and Analysis of Fatigue Resistant Welded Structures. *Elsevier.* 1990.

[6] Niemi E. Recommendations Concerning Stress Determination for Fatigue Analysis of Welded Components. *IIW - International Institute of Welding IIW-Doc.* 1994, University of Technology.

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

[**8**] Fermér M, Andrasson M and Frodin B. Fatigue Life Prediction of MAG-Welded Thin-Sheet Structures. *Proc. of SAE Conference SOC Automotive Engineers INC*, 1998, 4956.

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

[**10**] Dong P. A Structural Stress Definition and Numerical Implementation for Fatigue Analysis of Welded Joints. *IJF - International Journal of Fatigue*, 2001, 23(10), 865876.

[**11**] Fricke W. Fatigue Analysis of Welded Joint, 2003s: State of Development. *Marine Structures*, 2003, 16(3), 185200.

[**12**] Niemi E and Tonskanen P. Hot Spot Stress Determination for Welded Edge Gussets. *Welding in the World -London- IIS/IIW-1477-99.* 2000, 44(5), 3137.

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

[**14**] Doerk O, Fricke W and Weissenborn, C. Comparison of different calculation methods for structural stresses at welded joints. *IJF - International Journal of Fatigue*, 2003, 25(5), 359369.

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

[**16**] MATLAB. *The MathWorks Inc.*, version 7.12.0.635 (R2011a). 2011 Natick, Massachusetts.

[**17**] ANSYS R. ANSYS Mechanical APDL, Product Release 13.0 (2012). *Ansys Inc.*, 2012, Canonsburg, Pennsylvania.