FINITE ELEMENT MODELING TECHNIQUES OF 3D WELDED JOINTS THE STRUCTURAL HOT SPOT APPROACH

Keurrie Cipriano Goes, keurrie.goes@arvinmeritor.com
ArvinMeritor do Brasil Sistemas Automotivos – Commercial Vehicle Systems – Av. João Batista, 825, Osasco, SP, Brasil
Universidade de São Paulo, Escola Politécnica – Av. Mello Moraes, 2231, São Paulo, SP, Brasil

Arnaldo Freitas Camarão, arnaldo.camarao@arvinmeritor.com
ArvinMeritor do Brasil Sistemas Automotivos – Commercial Vehicle Systems – Av. João Batista, 825, Osasco, SP, Brasil

Andre Lopes dos Santos, andre.santos3@arvinmeritor.com
ArvinMeritor do Brasil Sistemas Automotivos – Commercial Vehicle Systems – Av. João Batista, 825, Osasco, SP, Brasil
Universidade de São Paulo, Escola Politécnica – Av. Mello Moraes, 2231, São Paulo, SP, Brasil

Gilmar Ferreira Batalha, gilmar.batalha@poli.usp.br
Universidade de São Paulo, Escola Politécnica – Av. Mello Moraes, 2231, São Paulo, SP, Brasil

Abstract. *The stress distribution in welded joints primarily depends on geometry, stress concentration, loading case and material elastic or plastic behavior. Classical analysis of fillet weld joints is not generally feasible when complex 3D geometry and load cases are involved. The Finite Element Modeling, when properly integrated with the weld parameters can be of great help for calculating these stresses. In this work a Finite Element modeling technique was developed to better adequate the representation of weld stiffness and stress distribution, considering the structural hot spot stress approach and mesh refinement sensitivity study to address the singularities at sharp corners normally seen at the toe of a weld. A T welded joint problem was studied to evaluate the practical use and benefits of this methodology. The welded joint was modeled with both 3D shell and 3D solid elements subjected to bending loading. The FE results were than compared with analytical studies and previous work. The stress results were imported to a Fatigue Analysis Program to predict the Fatigue Life according to the appropriate welded joint configuration, using Stress x Life (SxN) curves from standard weld codes (such as BS7608), considering material thickness, stress relief, environment and fatigue strength improvements. The practical use and benefits of this methodology is discussed and compared to other approaches. Major challenges associated with this modeling and improvements proposals are finally presented.*

Keywords*: Welded Joint; Finite Element; Modeling; Hot Spot Stress; 3D Solid Element*

1. INTRODUCTION

A welded joint submitted to a cyclic load can crack due to fatigue. Branco et al (1986), summarizes several aspects related to the fatigue strength of a weld joint as following:

- geometry and structural discontinuities
- stress distribution
- material and microstructure
- cyclic stress
- mean and residual stresses

As pointed out by Kou (2003) *apud* Sanders and Day, in developing any fatigue behavior criteria for welding, the severity of joint geometry is probably the most critical factor. This can be explained because the geometry of the seam weld results in high stress concentration that contributes to reduce the Fatigue Life strength (Fricke, 2003).

The classical method for calculating stresses in fillet welds requires the determination of the load transmitted through the weld per unit length. This load is usually composed by normal, shear and bending components (Shigley, 1989), but this method does not account Stress Concentration Factors (SCF) or geometric discontinuities at weld location, only nominal stresses are considered.

This work is based on Alexandre et al (2001), considering the finite element mesh refinement influence and integrating FE results to the Fatigue Analysis software.

The Fatigue Analysis software (FE-Fatigue) calculates the Fatigue Life distribution on the components. This software uses the stress results from the FE program (Ansys) and the Stress x Life curves from the British standard BS7608 (1993) to calculate the life distribution on the welded component.

2. HOT SPOT APPROACH

Hot-spot is a term which is used to refer to the critical point in a structure where fatigue cracking can be expected to occur due to discontinuity or notch. Usually the hot-spot is located at the weld toe. The hot-spot stresses account only the overall geometry of the joint and exclude local stress concentration effects due to weld geometry and discontinuities at the weld toe (Bäckström, 2003). Hot-spot stresses used in combination with modified Wöhler curves can be successful in predicting Fatigue Life time of welded details subjected to multiaxial fatigue loading (Susmel and Tovo, 2006). The figure 1 shows the approximate location of nominal and hot spot stress.

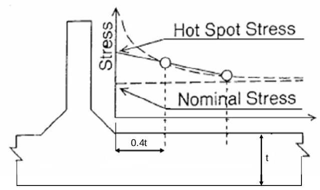


Figure 1. Approximate location of nominal and hot spot stress (Miki, 2003).

In this approach, the fatigue strength, expressed as an S-N curve, is generally based on strains measured in the specimen near the point of crack initiation (Niemi, 1995).

For design, there are three possible methods for such an analysis (Niemi, 1995):

1) The calculated nominal stress is multiplied by the stress concentration factor, K_s, for the appropriate structural discontinuity;
2) Strain ranges are measured during prototype or model tests at the hot spot as described in figure 2;
3) Stresses and strains are analysed by FEA using shell or solid elements.

This work considered the 3rd analysis, where the results include the biaxiality effects. According to the figure 1, all values of hot spot stress in FEA simulations were taken in a distance of *0.4*t* from weld toe, being *t* the throat thickness of the fillet welds (further dimensional information in section 4.)

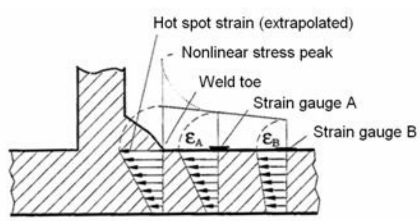


Figure 2. Measurement of the hot spot strain range using linear extrapolation method (Niemi, 1995).

3. FATIGUE ANALYSIS

The Stress results from the Finite Element Analysis were imported on the Fatigue Analysis software FE-Fatigue to calculate the Fatigue Life through BS7608 Stress x Life (S x N) curves.

The geometric discontinuities usually present in welded structures act as stress raisers. Such stress raisers can produce global or local effects and they frequently interact resulting in very high local stresses.

There are four basic approaches for Fatigue Life prediction of weld components (Iida, 1984), i.e. nominal, structural hot spot stress, local notch stress and fracture mechanics approach.

In this work the nominal and the hot spot stress, at critical location, were used. Although the hot spot is located at a local notch, the hot spot stress does not include the nonlinear stress peak caused by the local notch.

The T-joint configuration was classified as “F2” class with 2,3% of failure probability according to BS7608 (1993). A general curve from BS7608 is presented in figure 3 and the calculation equations to Fatigue Life are as following:

$$C_d = NS_r^{\ m} \tag{1}$$

$$Log(C_d) = Log(C_o) - d * \sigma \tag{2}$$

Where:
N = Predicted number of cycles to failure
S_r = Stress range per cycle
C_0 = Constant related with the mean SxN curve (50% of failure probability)
C_d = Predicted mechanical solicitation to failure
σ = The standard deviation of log (N)

d = Number of standards deviation below the mean S x N curve
m = The inverse slope of log (S_r) versus log (N) curve.

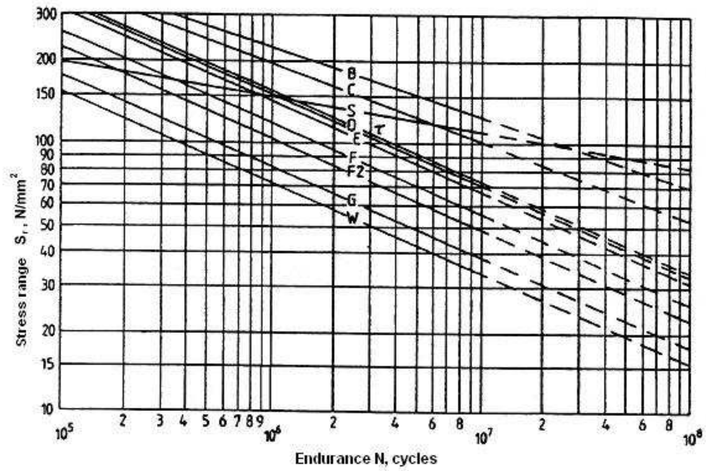


Figure 3. Fatigue life curve to each welded joint class (BS7608, 1993).

4. MATERIAL AND METHOD

Two materials were used to model the T-joint, as indicated in table 1.

Table 1. Monotonic Material Properties.

Monotonic Properties	Material	
	Class F2 welded joint	SAE1020
S_y (Yield Strength)	355 MPa	260 MPa
S_{ut} (Ultimate Tensile Strength)	500 MPa	441 MPa
E (Young Modulus)	207 GPa	207 GPa
ν (Poisson Ratio)	0.3	0.3

The SAE1020 was used for the base material and class F2 for the heat affect zone (HAZ) location. The stress-life (SxN) curves from both materials are showing in figures 4 and 5.

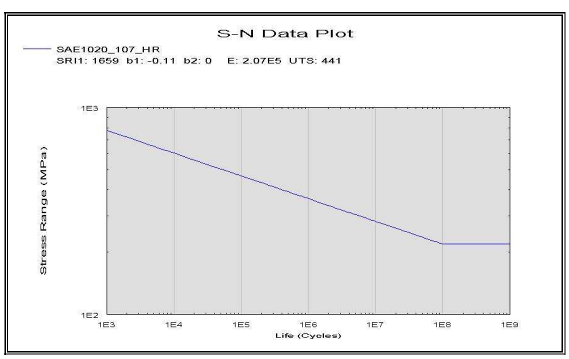


Figure 4. Stress-Life curve (logS x logN) to SAE1020.

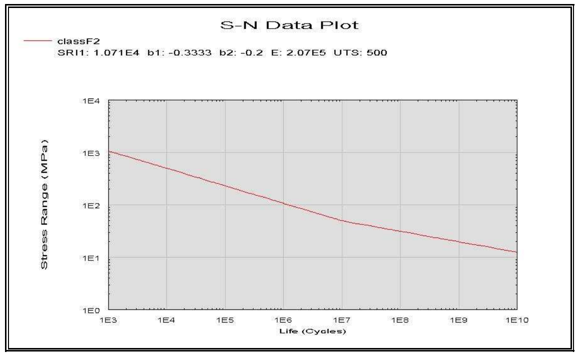


Figure 5. Stress-Life curve (logS x logN) to class F2.

The T-Joint configuration with fillet welds at right and left corners is shown in the figure 6.

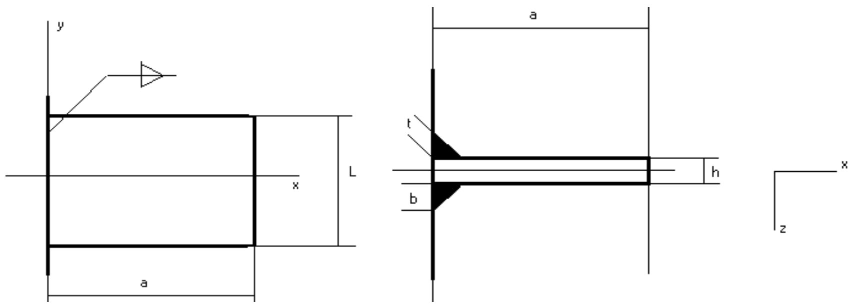


Figure 6. T-Joint configuration (Alexandre et al, 2001).

The section properties for double sided fillet weld:

- Throat Area

$$A = 2 \cdot t \cdot L = 2 \cdot b \cdot \cos 45° \cdot L \tag{3}$$

- Moment of Inertia about x axis

$$I_{xx} = \frac{t \cdot L^3}{6} = \frac{b \cdot \cos 45° \cdot L^3}{6} \tag{4}$$

The dimensions of the plates are:
a = 200 mm; L = 200 mm; h = 5 mm
The weld fillet size (b) is 3 mm and the throat thickness of the fillet welds (t) is 2.12 mm.

4.1. Element type

In order to show the influence of the finite element type and weld geometry on the numerical model, four different FE models for the T-joint were created. Two 3D shell models (Ansys Shell 63), based on the work done by Alexandre et al (2001) and two 3D solid models with 20 node quadratic hexahedron elements (Ansys Solid 95). The contact element (Ansys contact 174) was used to model the gap between the two plates at the no weld penetration location. The four FE models are listed bellow.

FE-MODELS – Has been considered four kinds of models following:
1 Shell 1 – with common T-Joint (fillet weld not considered; shell elements simply connected).
2 Shell 2 – with increased throat thickness in the weld zone;
3 Solid 1 – 3D solid elements without contact at surface 3-4 (figure 7)
4 Solid 2 – 3D solid elements with friction contact at surface 3-4 (figure 7)

4.2. Loading

A bending Fy load was applied on the 1-2 surface and clamping the lateral surfaces 5-6 and 7-8 by constraining all degrees of freedom (see figure 7).

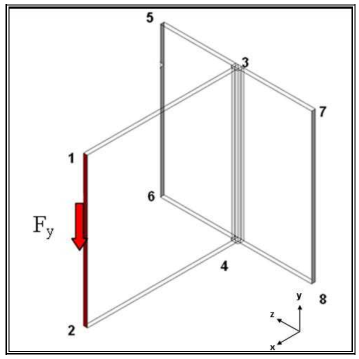
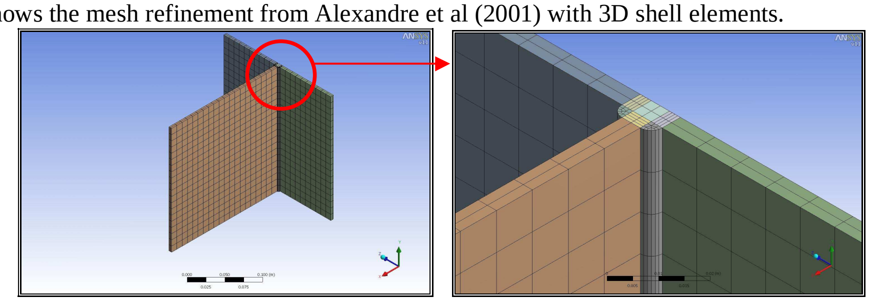


Figure 7. Loading and boundary conditions.

The load was out-of-plane bending moment F_y = 1,000 N.

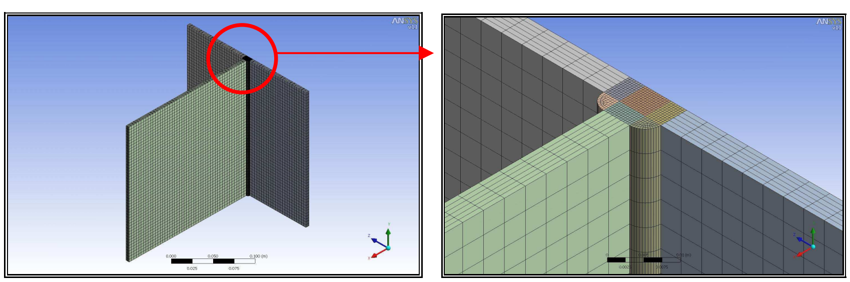
4.3. FE Model

The first two 3D solid models were initially modeled with a coarse mesh, as indicated on figure 8, to conduct the mesh refinement sensitivity study. The mesh refinement that led to minimum stress error in this analysis (maximum of 10% in stress results) is the one shown in figure 9. Both 3D Solids models 1 and 2 have the same FE mesh refinement. The figure 10 shows the mesh refinement from Alexandre et al (2001) with 3D shell elements.



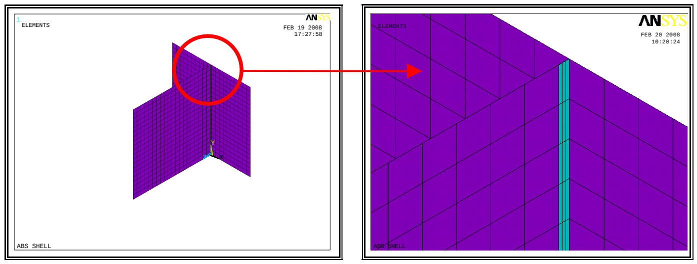
(a) (b)

Figure 8. Coarse mesh: Solid 1 and 2 mesh (a) and detail of the weld zone (b).



(a) (b)

Figure 9. Refined mesh: Solid 1 and 2 mesh (a) and detail of the weld zone (b).



(a) (b)

Figure 10. Shell 1 and 2 mesh (a) and detail of the throat thickness in the weld zone (b) (Alexandre et al, 2001).

The stresses from Finite Element Analysis results were compared to classical as per the equation

$$\sigma_z = \frac{(F_y \cdot a)}{I_{xx}} \cdot \frac{L}{2} \qquad (5)$$

5. RESULTS AND COMMENTS

5.1. Stresses and displacements

The 3D numerical models were analyzed applying the F_y on the surface 1-2 (figure 7).

Shell models 1 and 2 displacements from Alexandre et al (2001) are shown in figure 11. The displacement distribution on solids 1 and 2 are showed in figure 12, for the refined mesh only. The 3D shell analysis was conducted in Ansys classical code and the 3D solid analysis in the Ansys Workbench.

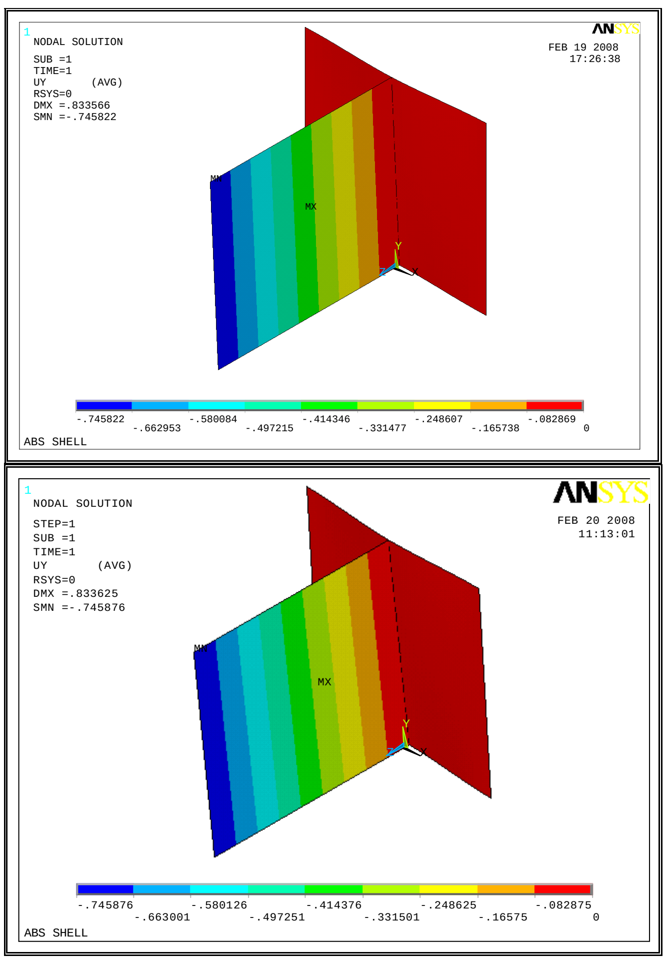


Figure 11. Displacement UY (mm) Shell 1 (top) and Shell 2 (bottom) (Alexandre et al, 2001).

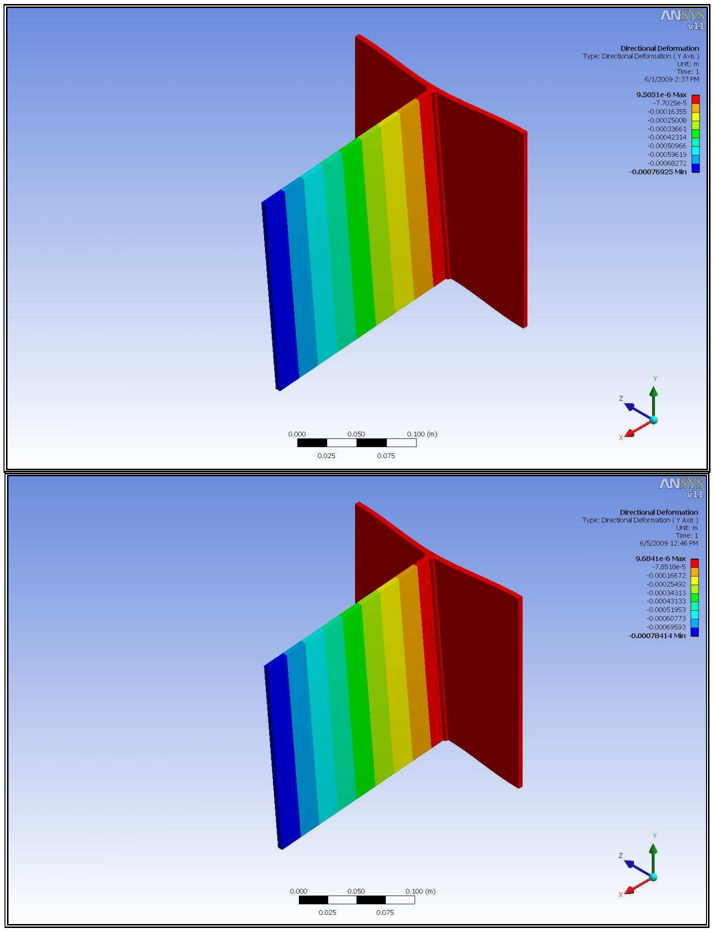
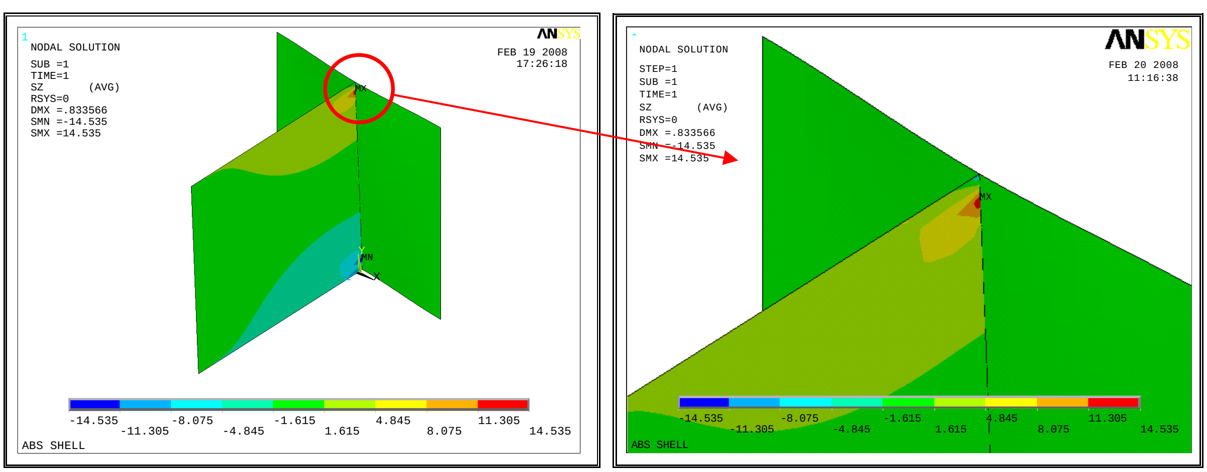


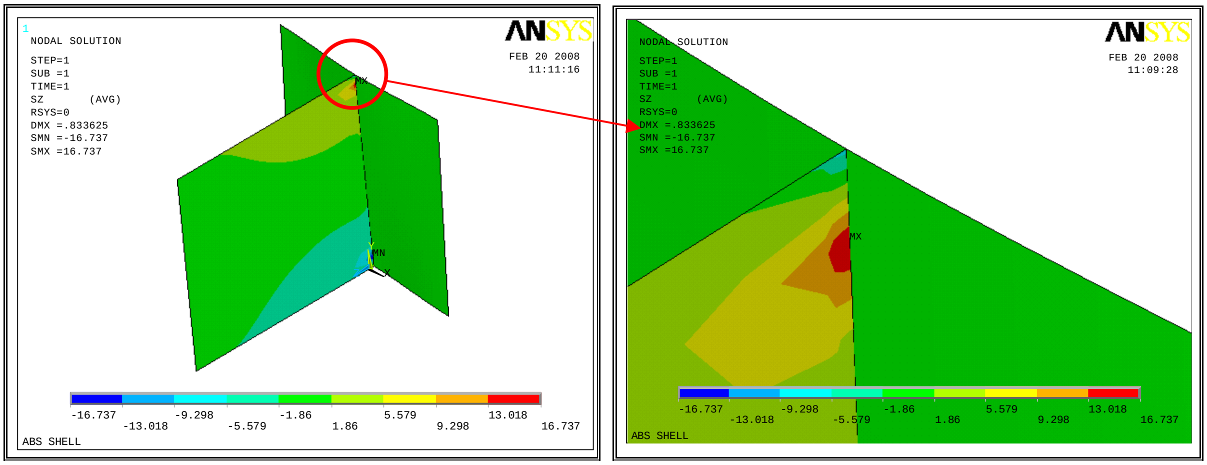
Figure 12. Displacement UY (m) Solid 1 (top) and Solid 2 (bottom).

The Normal stress σ_x (direction 1-3; figure 7) was used to compare with previous works like Alexandre et al (2001). The figures 13 and 14 show the normal stress results (σ_z) from Alexandre et al (2001) for the shell analysis and the solid models stress distribution in this work are showed in figures 15 and 16. Note that the “x” axis on the solid model built in this work corresponds to the “z” axis on (Alexandre et al, 2001) shell model.



(a) (b)

Figure 13. Normal Stress (MPa) σ_z to shell 1 model (a) and detail of weld location (b) (Alexandre et al, 2001).



(a) (b)

Figure 14. Normal Stress (MPa) σ_z to shell 2 model (a) and detail of weld location (b) (Alexandre et al, 2001).

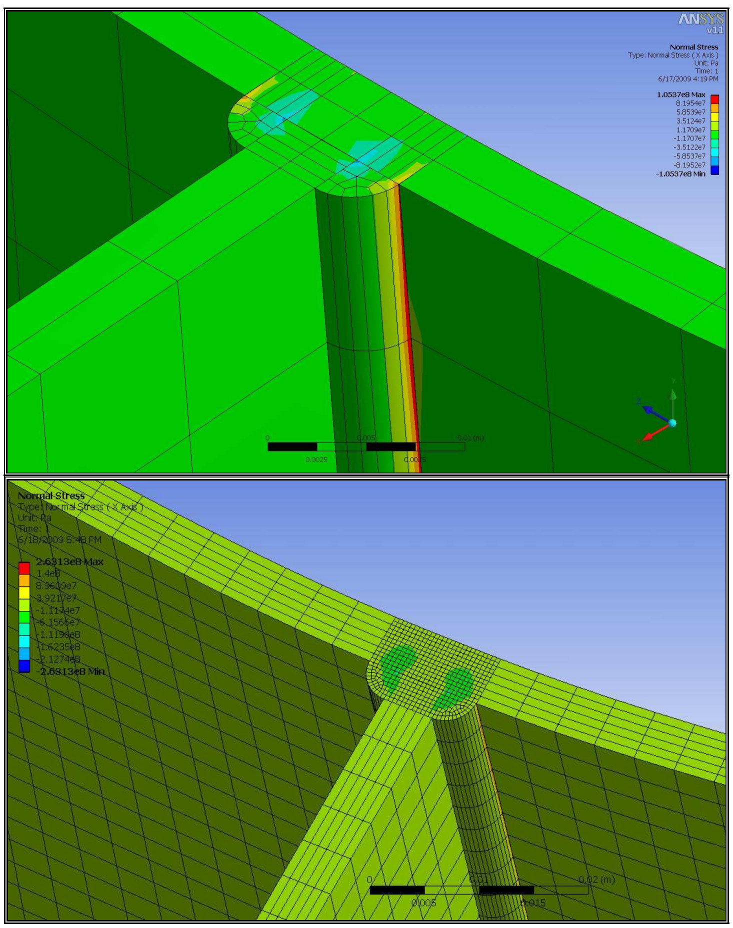


Figure 15. Solid 1 Normal Stress (Pa) σ_x (without contact) – detail of welded location. Coarse mesh (top) and refined mesh (bottom).

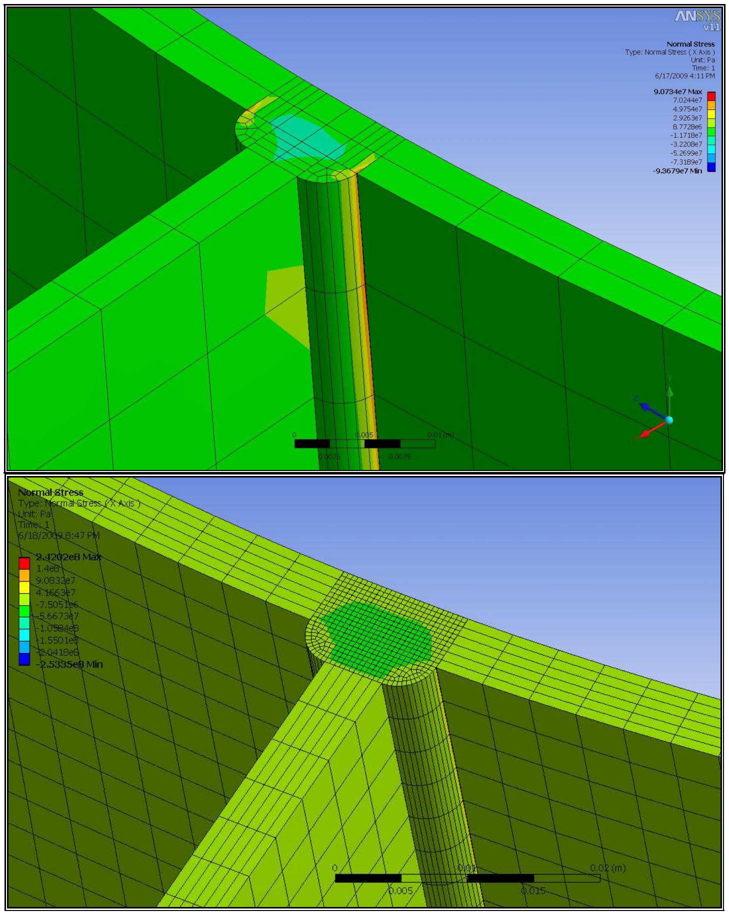


Figure 16. Solid 2 Normal Stress (Pa) σ_x (with frictional contact) – detail of welded location. Coarse mesh (top) and refined mesh (bottom).

Table 2 summarizes the normal stress σ (direction 1-3; figure 7) for each model and maximum displacement at edge 1-2. Nominal and hot spot stress are presented for each model in order to evaluate the stress concentration (SCF) and mesh refinement effects for the solid model.

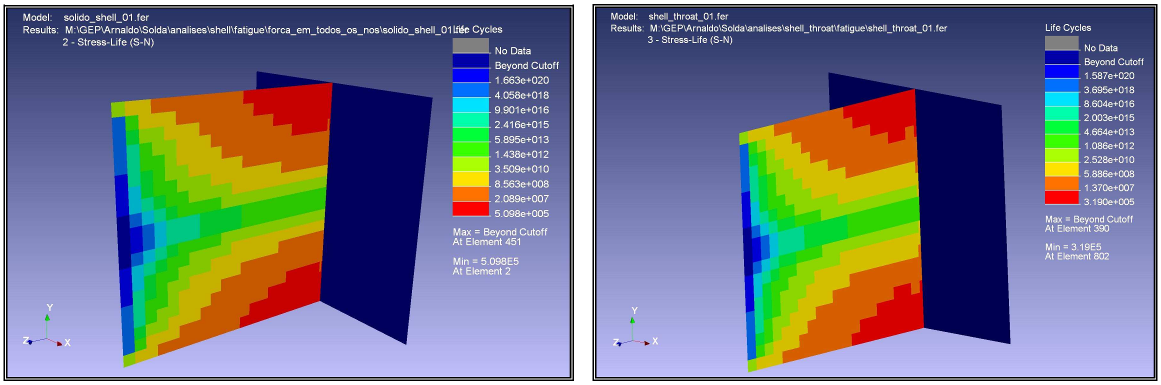
Table 2. Comparison of the stress and displacement for all models.

Models		**Normal Stress σ at weld location [MPa] (1)**		**Maximum displacement at edge 1-2**
		Nominal	**Hot spot**	**UY (mm)**
Classical		7	-	0.004
1 – Shell 1 (simply connected)		8	15	0.746
2 – Shell 2 (double sided throat thick.)		9	17	0.746
3 – Solid 1 (no contact)	a – coarse mesh	30	102	0.764
	b – refined mesh	30	136	0.769
4 – Solid 2 (friction contact)	a – coarse mesh	30	75	0.774
	b – refined mesh	30	89	0.784

(1) "x" axis on the solid model = "z" axis on (Alexandre et al, 2001) **shell model**

5.2. Fatigue

The stress results from each Finite Element Analysis were imported to the Fatigue Analysis software (FE-Fatigue) in order to calculate the Fatigue Life prediction. The class F2 stress life (SxN) curve was selected for the welded region and SAE1020 was assigned for the base metal. The Fatigue Life distribution for the shell models were obtained from the previous work conducted by Alexandre et al (2001) as shown in figure 17. The fatigue life distribution for the solid models in this work are shown in figures 18 and 19.



(a) (b)

Figure 17. Fatigue Life (cycles) to Shell 1 – simply connected at the joint location – (a) and Shell 2 – double sided throat thickness – (b) (Alexandre et al, 2001).

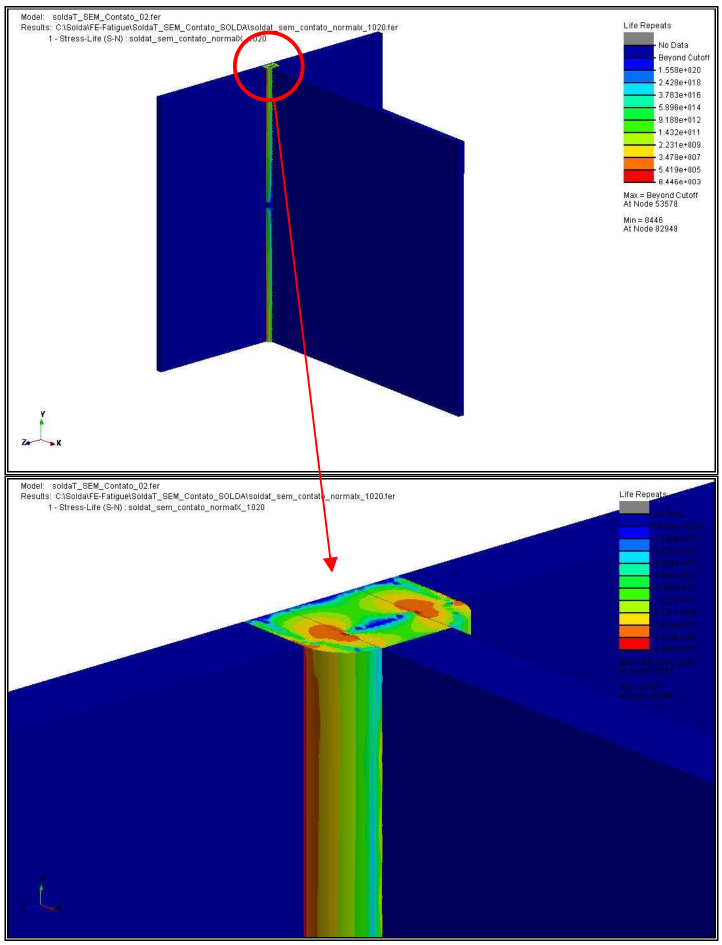


Figure 18. Fatigue Life (cycles) – Solid 01 (no contact) - refined mesh (top) and detail of weld location (bottom).

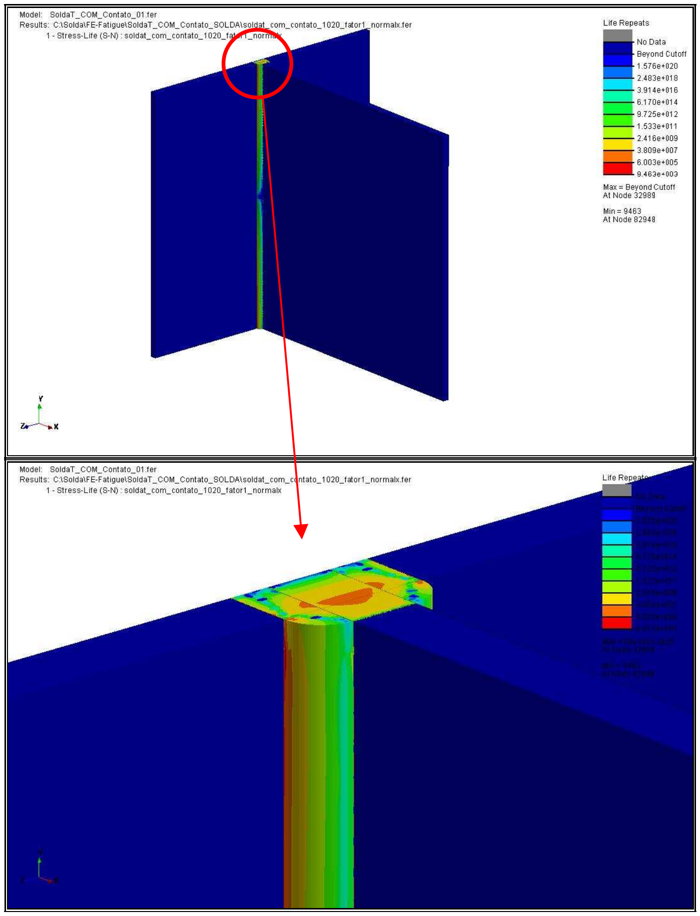


Figure 19. Fatigue Life (cycles) – Solid 02 (friction contact) - refined mesh (top) and detail of weld location (bottom).

As seen in table 2, the 3D solid models (model 3 and 4) presented the highest stresses in comparison to the other models. These models were able to better simulate the stress concentration at the weld toe, resulting in higher stress. The displacement and stress deviation from the classical result is due the fact that the finite element model considers the bending deflection of the clamped plate on the edges 5-6 and 7-8, resulting in higher stress and total displacement.

Fatigue Life prediction based on the obtained stress for each solid model is compared against classical results. This information is shown in table 3. The adopted bending stress considers a single reversed load F_y.

Table 3. Fatigue Life prediction for fully reversal σ_x.

Models		**FE-Fatigue (software) (Cycles)**		**BS7608 SxN curve (Analytical) (Cycles)**	
		Nominal Life	**Hot Spot Life**	**Nominal Life**	**Hot Spot Life**
3 – Solid 1 (no contact)	a – coarse mesh	6.30E6	1.32E5	4.60E5	1.40E5
	b – refined mesh	5.42E5	8.45E3	7.80E4	8.00E4
4 – Solid 2 (friction contact)	a – coarse mesh	8.70E6	1.87E5	5.50E5	3.40E5
	b – refined mesh	6.00E5	9.46E3	5.50E5	1.80E5

Table 4. Reference values for Fatigue Life predictions (Alexandre et al, 2001).

Models	**Life Prediction (N)**
Theoretical	$1,79 \times 10^6$
Shell elements	$2,35 \times 10^6$
Shell with increased throat thickness in weld zone	$1,46 \times 10^6$
Inclined shell elements for modeling weld zone	$2,74 \times 10^5$
3D – solid elements	$1,84 \times 10^5$

6. CONCLUSION

Integrating FE results with fatigue analysis software resulted in a more effective and reliable method to perform Fatigue Analysis of welded components.

The Finite Element determination of hot spot stress in welded components at the weld location does not substitute the experimental SCF inherent in S x N curves.

The 3-D solid modeling was able to better simulate the stress concentration and the stiffness of the welded plates.

The hot spot stress results on this study are significant higher than the previous work. This is caused by the finite element mesh refinement. So, it is strongly recommended to conduct finite element mesh refinement study to get precise results. The friction nonlinear contact analysis produced similar results when compared to the non contact analysis to simulate the gap of weld penetration at the T-joint location. Considering that contact analysis is extremely time consuming and expensive, specially when applied to large models, typically observed in real welded structures, it is largely recommended to use the non contact analysis which showed to be an effective approach to compute the hot spot stresses in welded joints.

The great benefit of this methodology is the possibility to investigate several weld joints configuration, components stiffness effects, weld location and stress concentration within the virtual environment, contributing for product design optimization. Validation time is then reduced and product quality and reliability achieved.

7. ACKNOWLEDGEMENTS

The authors acknowledge ArvinMeritor for the great support on this research and continuous incentive to studies aiming innovation.

8. REFERENCES

Alexandre, S. F., Camarão, A. F., Fernandes, A. O., Bezerra, M. A., Argentino, M. A. and Ripoli, R. R., 2001, “A Finite Element Approach for Fatigue Analysis of Welded Components”, SAE Brasil International Fatigue Conference, pp. 229-235.

Bäckström, M., 2003, “Multiaxial Fatigue Life assessment of welds based on nominal and hot spot stresses”, VTT technical Research Center of Finland, VTT Publications nº 502, Finland. 107p.

Branco, C. M., Fernandes, A. A. and Castro, P. M. S. T., 1986, “Fadiga de Estruturas Soldadas”, Ed. Fundação Calouste Gulbenkian, Lisboa, Portugal.

BS7608, 1993, “Code of practice for Fatigue design and assessment of steel structures”, British Standard.

Fricke, W., 2003, “Fatigue analysis of welded joints: state of development”, Marine Structures, Vol. 16, pp. 185-200.

Gustafsson, J. and Saarinen, J., 2007, “Multi-axial fatigue in welded details – An investigation of existing design approaches”, Master´s Thesis, Chalmers University of Technology, Sweden, 117p.

Iida, K., 1984, “Application of hot spot strain concept to Fatigue Life prediction”, Weldin in the World, vol 22, n. 9/10,

Kou, S., 2003 “Welding Metallurgy”, 2nd Ed, John Wiley & Sons.

Miki, C., 2003, “Local (Structural) Stress Based Fatigue Design, Retrofit Engineering for Urban infrastructures”, Tokyo Institute of Technology. 25 Aug. 2008, <http://www.cv.titech.ac.jp/~miki-lab/english/class/retrofit_eng/Class6-Structural-Stress-Based/Class-6-Local-Stress-Based-fin.pdf >

Niemi, E., 1995, “Stress Determination for Fatigue Analysis of Welded Components”, Ed. Abington Publishing, Cambridge, England, 69p.

Shigley, J. E. and Mischke, C. R., 1989, “Mechanical Engineering Desing”, McGraw-Hill, Singapore, pp. 222-246.

Susmel, L. and Tovo, R., 2006, “Local and structural multiaxial stress in welded joints under fatigue loading”, International Journal of Fatigue, Vol. 28, pp. 564-575.

5. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.