# Modeling of Residual Stresses in a Butt-welded Joint with Experimental Validation

Vivek Srivastava* and AG Rao
Naval Materials Research Laboratory (NMRL), Defence Research and Development Organization (DRDO), Ambernath

*Corresponding author: NMRL-DRDO, Shil-Badlapur Road, Anand Nagar MIDC PO, Additional Ambernath-East, Thane, Maharashtra, India – 421506
Email: vivek.sri5@gmail.com

**Abstract:** Fatigue behaviour of welded structures is critically dependent on the residual stresses present in the weld joints. However, fatigue analysis requires the prior information of the magnitudes of the residual stresses, either predicted by experimental techniques or by FEM simulation of arc welding process. In this study, complex multi-physical phenomenon of arc welding is 2D-modeled using COMSOL with thermal-structural coupling to predict the residual stresses distribution across thickness-oriented plane of a butt-welded joint of an HSLA steel. To experimentally validate the above model, residual stresses were also experimentally measured using X-Ray Diffraction (XRD) method. It was found that there is close agreement (within 15 %) between the simulated and experimental values of in-situ residual stresses validating the accuracy of COMSOL simulated FEM model of the studied butt-weld joint.

**Keywords:** Welding, Butt Joint, Residual Stress, HSLA Steel, FEM

## 1. Introduction

Marine structures are constructed by joining many assemblies or sub-assemblies by numerous welding operations using medium to high thickness HSLA steel plates. Fatigue behaviour of welded structures is critically dependent on the type (tensile or compressive) and magnitudes of in-situ residual stresses as well as their variations with thickness of the weld joints. Dependence of fatigue behaviour on the residual stresses has been studied in butt joint [1, 4] and T-joint [1, 2] weld configurations. However, fatigue analysis requires the prior information of the magnitudes of the residual stresses measured in-situ by experimental techniques (destructive or non-destructive). In-situ measurements of residual stresses are not always feasible or accurate enough, given the limitations of the experimental techniques. Hence, it is of paramount significance that the weld joints are modelled to conduct FEM simulation of the arc welding process for predicting the in-situ residual stresses existing in these joints. Since, arc welding is a complex process consisting of thermal / structural/mass transport / metallurgical / magnetic interactions, COMSOL being a multi-physical simulation platform, can be utilized efficiently to simulate arc welding [3] and predict the residual stresses with adequate accuracy. In this study, 2D simulation of butt-welded joint of HSLA steel was conducted to predict residual stresses distribution primarily across the thickness of the weld joint. Heat transfer and solid mechanics modules were interactively coupled to simulate the temperature evolution during welding of weld joint from two HSLA steel plates. Finally residual stresses were predicted from steady-state solution of the Von Mises stresses evolved during welding.

## 2. Problem Description

Two HSLA steel plates (1 and 2) of dimensions (500 mm x 500 mm x 40 mm) each were weld simulated as per schematic drawing in Figure 1. Welding process parameters details considered for simulation are listed in Table 1. Since, this geometry was 2D-modeled for weld simulation hence it was assumed that the weld electrode arc contact occurred with a circular span radius of 1.5 mm with upper contact point of two joining plates as its centre. This arc contact area was used for calculating the input surface heat flux during welding.

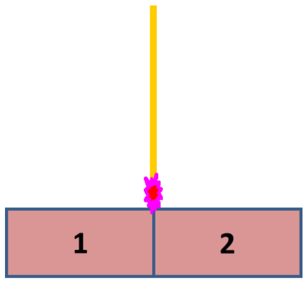


**Figure 1**: Schematic Geometry of Butt-Weld Joint used for this study

**Table 1:** Weld Process Parameter Details

| Parameter | Value |
|---|---|
| Plates | 1 and 2 |
| Material | Structural Steel |
| Weld Joint Type | Butt |
| Weld Pre-Heat / Interpass Temperature | 150 $^{o}$C (423 $^{o}$K) |
| Heat Input | 1.5 kJ / mm |

## 3. Multi-Physical Model Description with use of COMSOL

Accurate and reliable residual stress predictions are essential for structural integrity and fatigue assessment of components containing residual stresses. However, finite element simulation of residual stresses due to welding involves in general many phenomena e.g. non-linear temperature dependent material behaviour, 3D nature of the weld pool and the welding processes and microstructural phase transformation. Despite the simplification by excluding various effects, welding simulations are still CPU time-demanding and complex. Hence, simplified welding simulation procedures are required in order to reduce the complexity and thus maintain the accuracy of the residual stress prediction. Figure 2 shows the simulation scheme and coupling fields in welding analysis used in this study on a 2D model of the butt-joint.

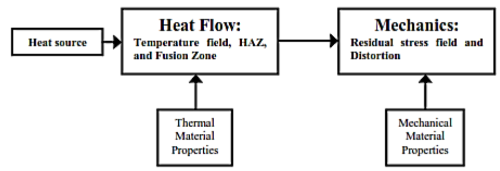


**Figure 2:** Simulation scheme in welding analysis

### 3.1 Heat Source

Heat source was modeled as a surface heat flux following Gaussian distribution. Entire geometry of the model with arc contact line AOB is shown in Figure 3. Surface heat flux from weld heat input, weld speed and contact length was calculated to be equal to 5e8 W/m$^2$. This was applied as a thermal boundary condition on length AOB of the top surface.

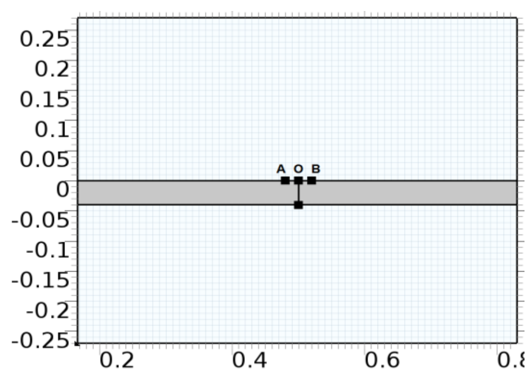


**Figure 3**: Model geometry with arc contact length AOB along with two weld joining plates

### 3.2 Thermal Behavior

Prediction of temperature field in welding process is one of the main factors in determining plastic strains and welding residual stresses. Heat transfer in solids physics model in COMSOL was used to describe the thermal behavior in the modeled domain in accordance with the thermal behavior discussed in literature [4, 5]. It included conductive heat transfer with phase change domain as described in Equation set 1. Convective heat transfer with appropriate boundary conditions was applied on the open surfaces of the geometry except arc contact length AOB as described in Equation set 2. Convective heat transfer coefficient was taken as h = 50 W/m$^2$-K whereas ambient temperature was taken as $T_{ext}$ = 293 K. Initial condition was selected as initial temperature as weld pre-heat / inter-pass temperature of 150 $^o$C (423 K).

A physics dependent fine mapped mesh defined in COMSOL was adopted in the butt-joint geometry for FEM simulation of welding process. Material thermo-physical properties such as density, specific heat, thermal conductivity, thermal expansion coefficient etc. were chosen from the "Structural Steel" material library of COMSOL with defined temperature dependence. Liquidus temperature of the steel i.e. melting point of the steel was taken as $T_{pc,1\rightarrow 2}$ = 1790 K where phase 1 was solid phase and phase 2 was liquid steel. Also latent heat of fusion was taken as L = 245 kJ/kg. s

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T = \nabla \cdot (k \nabla T) + Q + Q_{vh} + W_p$$
$$k = \theta k_{phase1} + (1-\theta) k_{phase2}$$
$$C_p = \theta C_{p.phase1} + (1-\theta) C_{p.phase2} + L\frac{d\alpha}{dT}$$
$$\rho = \frac{\theta \rho_{phase1} C_{p.phase1} + (1-\theta)\rho_{phase2} C_{p.phase2}}{\theta C_{p.phase1} + (1-\theta) C_{p.phase2}} \qquad (1)$$

$$-\mathbf{n} \cdot (-k\nabla T) = h \cdot (T_{ext} - T) \qquad (2)$$

### 3.3 Structural Behavior

In order to describe the structural behavior during welding, solid mechanics physics module of COMSOL was used. Linear Elastic material domain was selected with constitutive stress-strain behavior with thermal effect as shown in Equation set 3. Thermal elastic-plastic behavior was described by a plasticity model [5, 6] based on Von Mises yield criteria and isotropic hardening model as shown in Equation set 4. Tangent modulus, $E_{Tiso}$ = 1 GPa was selected as input in the isotropic hardening model. Zero values of initial stress, strain, displacement and structural velocity fields were initial conditions. Considering the restrained welding configuration during the course of this study, prescribed displacement in vertical direction were taken as zero ($u_y$ = 0) at points 1-8 as shown in Figure 4.

$$-\nabla \cdot \sigma = F_V, \quad \sigma = s$$
$$s - S_0 = C : (\epsilon - \epsilon_0 - \epsilon_{inel})$$
$$\epsilon = \frac{1}{2}\left[(\nabla \mathbf{u})^T + \nabla \mathbf{u}\right] \qquad (3)$$

$$s - S_0 = C : (\epsilon - \epsilon_0 - \epsilon_{inel}), \quad \epsilon_{inel} = \epsilon_p$$
$$F(\sigma, \sigma_{ys}) \le 0, \quad \dot{\epsilon}_p = \lambda \frac{\partial Q}{\partial \sigma}$$
$$F = \sigma_{mises} - \sigma_{ys}, \quad Q = F$$
$$\sigma_{ys} = \sigma_{ys0} + \frac{E_{Tiso}}{1 - \frac{E_{Tiso}}{E}} \epsilon_{pe}$$
$$\epsilon_{th} = \alpha (T - T_{ref}) \qquad (4)$$

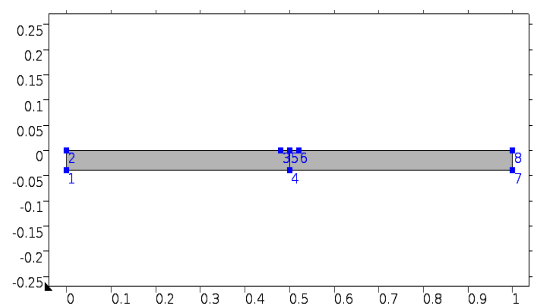


**Figure 4:** Restrained welding configuration leading to zero vertical displacements at points 1-8

Multi-physics coupling was done for thermal-structural interaction by considering thermal expansion (thermal strain) and temperature coupling to source from heat transfer module solution to solid mechanics solution.

## 4. Results and Discussion

This welding simulation 2D model was solved with applied initial and boundary conditions. Time duration was varied upto 300 seconds with the time step size of 10 seconds for heat transfer solver. Figure 5 presents the temperature field evolution snapshot at an intermittent time step (t = 120 seconds) in the weld fusion zone showing the location and value of the temperature maxima. It shows the temperature evolution within the weld fusion zone showing symmetry in the initial arc heat flux transfer, subsequent heating and cooling cycles.

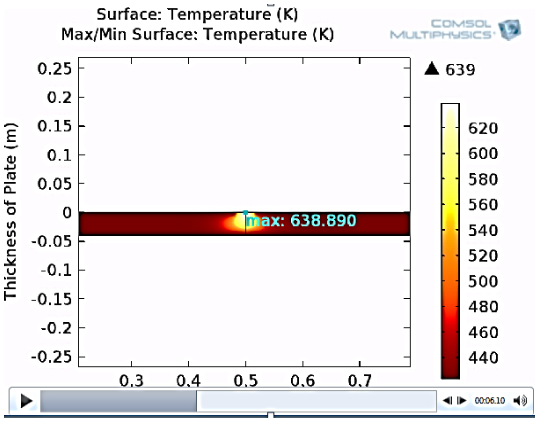


**Figure 5:** Temperature field evolution in the fusion zone screenshot at t = 120 s

Using temperature coupled solid mechanics solver, steady state solution obtained for the final state of Von Mises stress was the measure of residual stresses evolved in the butt-joint after simulated welding. Predicted Residual stresses distribution across the thickness-oriented plane in the form of iso-stress contours is shown in Figure 6.

In order to validate this COMSOL model, X-Ray Diffraction (XRD) method was used to experimentally measure residual stresses present in the weld zone of a physically butt-welded HSLA steel with the identical welding parameters as the simulation. Residual stress levels were measured (experimental by XRD and simulated by COMSOL) at three discrete locations along the weld line in thickness direction i.e. at the depths of 10, 20 and 30 mm. Table 2 shows the comparative validation between COMSOL predicted and XRD measured residual stresses with percentage deviations. It can be observed that there is close agreement (within 15 %) between the simulated and experimental values of in-situ residual stresses validating the accuracy of COMSOL simulated FEM model of the studied butt-weld joint.

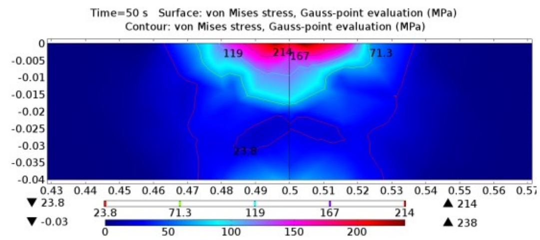


**Figure 5:** Residual stress distribution across thickness plane showing iso-stress contours

**Table 2:** Comparative validation between COMSOL predicted and XRD measured residual stresses with percentage deviations

| *Depth (mm)* | *Residual Stress (Simulation) – COMSOL* | *Residual Stress (Experimental) – XRD* | *Percentage Deviation (%)* |
|---|---|---|---|
| 10 | 116.1 | 100.8 | 15.2 |
| 20 | 65.3 | 73.5 | 11.2 |
| 30 | 32.1 | 34 | 5.6 |

## 5. Conclusions

.

1. In this study, complex multi-physical phenomenon of arc welding for a butt-joint configuration was 2D-modeled using COMSOL with thermal-structural coupling.
2. Inward surface heat flux from weld electrode arc, conductive and convective heat transfer with appropriate initial and boundary conditions were applied to describe the temperature evolution within the fusion zone and plates.
3. Further constitutive stress-strain behavior was solved with thermal-elastic-plastic effect using a plasticity model based on Von Mises stress and isotropic hardening.
4. Steady state solution achieved for final state of the stress was the measure of residual stresses distribution across thickness-oriented plane of a butt-welded joint of an HSLA steel.
5. Experimental validation of the above model was performed by measuring

residual stresses using X-Ray Diffraction (XRD) method. Close agreement (within 15 %) between the simulated and experimental values of in-situ residual stresses was found validating the accuracy of COMSOL simulated FEM model of the studied butt-weld joint.

## 6. References


1. Z. Barsoum, I. Barsoum, Residual stress effects on fatigue life of welded structures using LEFM, *Engineering Failure Analysis* **16** (2008) 449-467

2. T. Mann et al, Fatigue of Welded Aluminium T-Joints, *European Conference of Fatigue* **ECF15** (2015)

3. F. Roger and A. Traidia, Modeling Residual Stresses in Arc Welding, *Proceedings of the COMSOL 2010 Boston* (2015)

4. A. Khurram and K.Shehzad, FE Simulation of Welding Distortion and Residual Stresses in Butt Joint using Inherent Strain, *International Journal of Applied Physics and Mathematics* **2** (2012) 405-408

5. Z. Barsoum and A Lundback, Simplified FE welding simulation of fillet welds – 3D effects on the formation residual stresses, *Engineering Failure Analysis* **16** (2009) 2281-2289

6. A. Bower, *Applied Mechanics of Solids*, 820, CRC Press 2011


## 7. Acknowledgements


Authors are thankful to K Sridhar, B Basu, RV Kalmekar, VP Deshmukh and Director NMRL for their guidance and support for this work. Sincere thanks are also due to ARAI Pune for conducting Residual stress measurements by XRD experimental method on physically butt-joint welded coupons.