JESTECH, 15(1), 33-37, (2012)

JESTECH

REPAIRED HSLA-100 STEEL PLATE WITH A CRACK UNDER AXIAL TENSION

Elif Koc*, Hamza Erdogan and Mehmet Yetmez****
*Turkish Hardcoal Enterprise, Zonguldak, Turkey
**Zonguldak Karaelmas University, Zonguldak, Turkey

Abstract

In this study, transverse crack behavior for a welded HSLA-100 plate under tension is investigated. Simple and efficient finite element method for solving the welded plate with a central crack is given. In order to simplify the approximate solution, results of a general purpose finite element code are presented. Stress intensity factor for the plate is determined by using J-integral solution. Experimental results of thickness and loading rate effects on strain fields are also presented with respect to EN ISO 15614-1 welding and ASTM E338 testing procedures. Results show that numerical solutions of the repaired plate are compatible with the experimental evaluations.

Keywords: Welded plate, Crack, J-integral, Finite element, Thickness effect

1. Introduction

Linear elastic fracture mechanics (LEFM) helps to make an acceptable structural design concept which yields a relationship between material properties, crack size and applied load. The principal objective of the analysis of LEFM problems is the calculation of the appropriate fracture mechanics parameters such as stress intensity factor (K) and J-integral (J). Nowadays, these parameters can be calculated easily in a variety of ways. To compute the parameters, researchers have usually used finite element method (FEM). According to FEM, two methods that have been applied are mostly used. First is the singular element method. In this method, specific element (a triangular or collapsed quadrilateral element) is to be represented. Tracey's work may be the best example for this case [1]. Second is the J-integral calculation method. Along any contour Γ, the J-integral method for two dimensional bodies is given by Rice [2]:

$$J = \int_{\Gamma} \left\langle wdy - T.\frac{\partial u}{\partial x} ds \right\rangle \quad (1)$$

In Eqn(1), *u* is the displacement vector, *y* is the distance along the direction normal to the plane of the crack, *s* is the arc length along the contour, *T* is the traction vector and *w* is the strain energy density of the material. This integral is independent of the contour that is selected and characterizes the strain state at the crack tip. Parks's work may be the best example for this case [3]. Parks presents a new finite element implementation that requires no special elements. Additionally, Henshell and Shaw say that special finite elements for crack tips are not necessary for plane stress or plane strain analysis. The whole structure can be analyzed using absolutely standard eight-noded elements [4]. Later on, Harrop shows that there is a particular crack tip element size which minimizes the error in the computed stress intensity factor [5]. In 1990s, not only eight-noded quadratic isoparametric elements,are used as fine meshing, but also four-noded bilinear elements are considered as well [6]. Belytschko, Liu and Moran [7] explain explicitly the element technology of four-noded quadrilateral element. They note that the four-noded quadrilateral element locks in plain strain for nearly incompressible materials when it is fully integrated.

The present work provides a general simple and efficient finite element method (i.e., course meshing) for

Corresponded Author; mehmet.yetmez@karaelmas.edu.tr

solving the welded plate with a central crack by using fully integrated four-noded quadrilateral both plane stress and plane strain elements. In order to verify the finite element results, elastic strains are measured on critical points including the welded zone, near welding boundary zone and crack tip zone.

2. Numerical Model

Regarding to two works [8,9], two general purpose finite element code namely MARC [10] is taken into consideration for the numerical simulation of a welded HSLA-100 plate under tension. To compute the strain and J-integral fields statically, the finite element model which clearly respresents the quarter part of the plate shown in Figure 1 is used with respect to half length L, half width W, half crack length a and half welding width z. Acceptable coarsed mesh are defined to represent the critical parts (i.e., crack tip and welded parts) properly and to calculate reasonably correct strain distributions and J values. Figure 1 also indicates the course meshed model with four-noded quadrilateral plane elements.

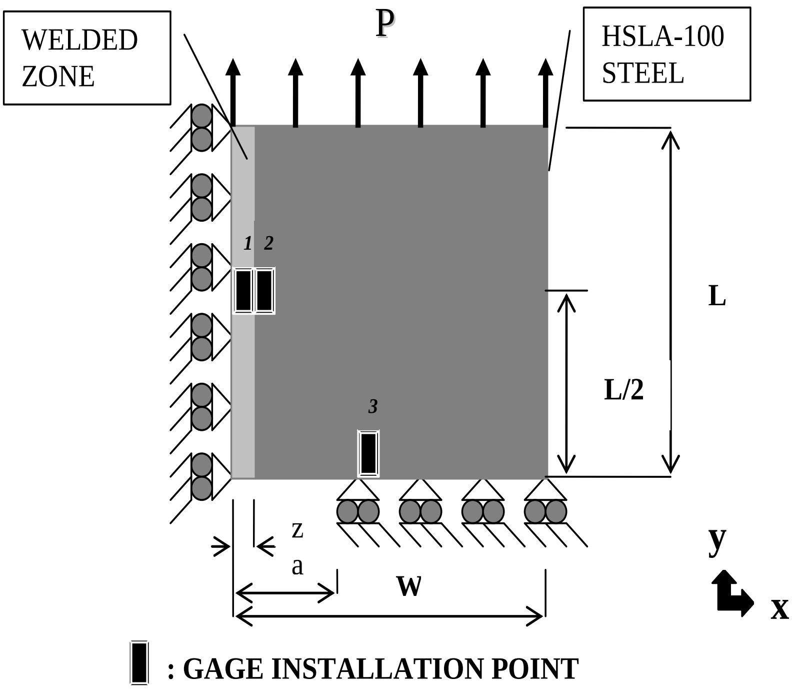

Figure 1. Finite element model of a welded HSLA-100 plate under tension load P.

According to ASTM E338-03 [11] and Figure 1, rectangular HSLA-100 plate of 75×300 mm is used with 2L=150 mm, 2W=75 mm, 2a=25 mm and 2z=4 mm. Corresponding to the plane stress or plane strain effect, thickness range of the plate includes 3, 6 and 8 mm.

The course meshed model with four-noded quadrilateral plane elements is analyzed by 496 nodes and 450 elements without any bias factor.

In this static analysis, full integration technique is preferred. FEM analysis is performed such that the repaired HSLA-100 plate is under small strain condition, i.e., $\varepsilon \leq 0.3\%$. Therefore, a constant axial tensile load of 70000 N is applied.

The HSLA-100 steel is assumed to be linear and elastic. It is also assumed that the material properties of both regions (i.e., welded zone, HSLA-100 steel zone) are the same as E=205 GPa and ν =0.28. In MARC analysis, weld fillers part are set with 3000 °C of melting temperature and 60 seconds of temperature activation time with respect to the welded zone of z×L.

Therefore, strain distributions and J_I values are obtained. By taking stress intensity factor K_I into account, following formulae are used. Under linear elastic, plane stress conditions,

$$J_I = \frac{K_I^2}{E} \qquad (2)$$

and plane strain conditions

$$J_I = \frac{K_I^2}{E}(1-\nu^2) \ . \qquad (3)$$

In order to compare K_I^{MARC} with the reference [12], the formula below is given:

$$K_I = \sigma_y \sqrt{\pi a}(\sec\frac{\pi a}{2W})^{1/2} \qquad (4)$$

where is σ_y the normal stress in y-direction, a is half length of the crack and $2W$ is the width of the plate.

3. Experimental Procedure

Using MAG welding of the general structural steel, the nine repaired HSLA-100 plates are prepared with full penetration. Through the thickness welded zone of z×L is given in Figure 1. MG-2 Wire electrode (TÜV:11071.00) is used. 120 A for 3 mm, 180 A for 6 mm and 195 A for 8 mm thickness are specified for this purpose.

Testing is conducted using a material testing machine (VEB Werkstoffprüfmaschinen, Germany). Three loading rates (i.e., crosshead speeds) are considered at 58.33, 1166.67 and 2333.33 N/s respectively. The loading is of loading-controlled type. As mentioned in the part 2, the load is kept constant at 70000 N.

The 3-channel strain measurements are completed using a microprocessor-based data acquisition system, namely SoMat eDAQ-lite and SoMat Test Control Environment (TCE) software (HBM, Inc., USA).

For the strain measurements in the y-direction, 6 mm foil resistance strain gages and an adhesive are used. The FLA-6-11 strain gages (gage factor: 2.12±0.01 and gage resistance: 350±1Ω) are products of TML, Tokyo Sokki Kenkyojo Co.,Ltd., Japan. The adhesive is produced by 3M, EU. For each of nine specimens, the unidirectional strain gages are mounted by considering three critical points shown in Figure 1 and 2.

4. Results and Discussions

Table 1 presents strain J-integral results of general purpose finite element code MARC [10] at the selected points as shown in Figure 1.

Table 1. Strain and J-integral results of MARC at the gage installation points

Thickness (mm)	ε_y^1 *(με)*	ε_y^2 *(με)*	ε_y^3 *(με)*	J_I *(MPamm)*
3	1770.0	1558.0	4318.0	19.950
6	813.0	719.0	1941.0	4.600
8	609.8	539.3	1456.0	2.588

Using J_I values in Table 1 and Eqn (4), K_I is computed and given in Table 2. When considered on a 1-to-1 basis of $K_I^{[12]}$ and K_I^{MARC} in Table 2, it is particularly important to say that the coarse meshed model of the repaired plate allows one to compare the strain field of FEM results and that of experimental results at the critical points selected.

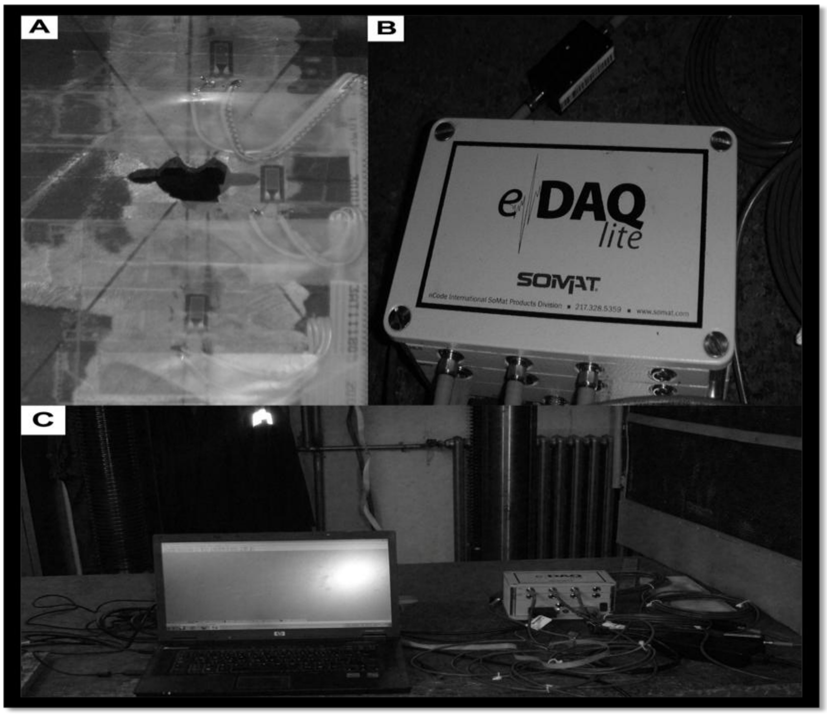

Figure 2. Setup of a welded HSLA-100 plate: (A) test specimen, (B) data acqusition system, (C) measurement configuration.

Table 2. Comparison of computed K_I values with that of reference [12]

Thickness (mm)	$K_I^{[12]}$ *(MPamm$^{1/2}$)*	K_I^{MARC} *(MPamm$^{1/2}$)*
3	2095.963	2022.313
6	1047.985	1011.544
8	785.989	758.731

Results of average measured strain values under various loading rate are given in Table 3. In this table, it is clearly seen that when the loading rate (R) and the plate thickness increase, only the results of coarsed meshed finite element analysis are roughly similar to that of experimental analysis.

While this type of plane stress through plane strain differences is commonly known, the repaired plate with welded zone under small strain conditions is affected by loading rate given in Table 3. In addition to the acceptable behavior regarding to thickness, it is expected that differences between the numerical and the experimental results.

Table 3. Average measured strain values considering the loading rates ([*]:R=58.33, [**]:R=1166.67and [***]:R=2333.33 N/s)

Thickness (mm)	ε_y^1 *(με)*		ε_y^2 *(με)*		ε_y^3 *(με)*	
	NUMERICAL	*EXPERIMENTAL*	*NUMERICAL*	*EXPERIMENTAL*	*NUMERICAL*	*EXPERIMENTAL*
3	1770.0	1453.157*	1558.0	1311.387*	4318.0	5443.567*
6	813.0	931.420**	719.0	805.085**	1941.0	1866.780**
8	609.8	479.587***	539.3	463.207***	1456.0	1153.853***

From the simple and coarse meshed efficient finite element model presented in this study, one may conclude that the loading rate is as important as the welding zone and its thickness effects. For the further studies, in order to use FEM efficiently in design problems (e.g., [14]), one may also say that a more simplified numerical model and its experimental research can be investigated by adding the effect of plasticity to the parameters thickness and loading rate at the interface between the steel part and the welded part. During the design process, this type of simplified approach may help for making a quick-repairabilty-result.

5. References

1. Tracey, D.M. Finite elements for determination of crack tip elastic stress intensity factors, Eng. Fracture Mech. vol. 3, 255-265, 1971.
2. Rice, J.R. and Rosengren, G.F. Plane strain deformation near a crack tip in a power law hardening material, J. Mech. Phys. Solids. vol. 16, 1-12, 1968.
3. Parks, D.M. A stiffness derivative finite element technique for determination of crack tip stress intensity factors, Int. J. Fracture. vol. 10, 487-502, 1974.
4. Henshell, R.D. and Shaw, K.G. Crack tip finite elements are unnecessary, Int. J. Num. Methods in Eng. vol. 9, 495-507, 1975.
5. Harrop, L.P. The optimum size of quarter-point crack tip elements, Int. J. Num. Methods in Eng. vol. 18, 1101-1103, 1982.
6. Reedy,Jr., E.D. Intensity of the stress singularity at the interface corner between a bonded elastic and rigid layer, Eng. Fracture Mech. vol. 36, 575-583, 1990.
7. Belytschko T., Liu W.K. and Moran B., Nonlinear Finite Elements for Continua and Structures. John Wiley&Sons Ltd, Chichester, 2000.
8. Yetmez, M. and Gecit, M.R. Stress analysis of a cracked finite strip with rigid ends, Turkish J. Eng. Env. Sci. vol. 29(6), 383-392, 2005.
9. Dong, Z., Wei Y. and Xu Y. Predicting weld solidification cracks in multipass welds of SUS310 stainless steel, Computational Mat. Sci. vol. 38, 459-466, 2006.
10. MARC Mentat 2010.1.0, MSC. Software Corparation, USA.
11. ASTM E338-03: Standard Test Method of Sharp-Notch Tension Testing of High-Strength Sheet Materials, ASTM International, West Conshohocken, PA,USA, 2003.
12. Skinn D.A., Gallagher J.P., Berens A.P., Huber, P.D. and Smith J., Damage Tolerant Design Handbook. Purdue Reserch Foundation, Dayton, 1994.
13. BS EN ISO15614-1: Specification and Qualification of Welding Procedures of Metallic Materials-Welding Procedure Test, London, 2004.
14. Bhat, S. And Ukadgaonkar, V.G. Dugdale cohesive zone modeling to evaluate J integral at the interface of strength mismatched steels: a simplified numerical approach, Finite Elements in Analysis and Design. vol. 46, 601-610, 2010.