

Stress Intensity Factor Solutions for Gas Metal Arc Welds in Lap-Shear Specimens

Catherine M. Amodeo
Johnson Controls

Jwo Pan
University of Michigan

ABSTRACT

In this paper, mode I and mode II stress intensity factor solutions for gas metal arc welds in single lap-shear specimens are investigated by the analytical stress intensity factor solutions and by finite element analyses. Finite element analyses were carried out in order to obtain the computational stress intensity factor solutions for both realistic and idealized weld geometries. The computational results indicate that the stress intensity factor solutions for the realistic welds are lower than the analytical solutions for the idealized weld geometry. The computational results can be used for the estimation of fatigue lives in a fatigue crack growth model under mixed mode loading conditions for gas metal arc welds.

CITATION: Amodeo, C. and Pan, J., "Stress Intensity Factor Solutions for Gas Metal Arc Welds in Lap-Shear Specimens," *SAE Int. J. Mater. Manf.* 8(3):2015, doi:10.4271/2015-01-0708.

INTRODUCTION

Weld geometry is one of the key factors affecting the cyclic loading performance of gas metal arc weld joints [1]. Niu and Glinka [2] investigated the effect of weld geometry on the stress intensity factor solutions for welded T-butt joints under a variety of loading conditions. Nguyen and Wahab [3] examined the effect of weld geometry and residual stresses on the fatigue life of butt welded joints based on the stress intensity factor solutions. Feng et al. [4] developed a fatigue life prediction model with the consideration of the crack initiation life at the weld toe, which was estimated using the notch strain approach, and the crack propagation life based on the stress intensity factor solution for cracks at the weld toe. Cerit et al. [5] investigated the effects of the weld geometry on the stress intensity factor solutions in cylindrical fillet-welded joints.

In this paper, the mode I and mode II stress intensity factor solutions for the pre-existing cracks near gas metal arc welds in lap-shear specimens are investigated. Two-dimensional, plane strain finite element analyses were carried out in order to obtain the computational stress intensity factor solutions for the realistic and idealized weld geometries. The computational solutions are compared with the analytical solutions based on the beam bending theory. Finally, some conclusions are made.

SPECIMENS

The lap-shear specimens examined in Amodeo et al. [6] are studied in this investigation. Lap-shear specimens of HSLA steel sheets with a thickness t of 2.0 mm were robot welded together using the gas metal arc welding process. 60 mm wide sheet specimens with the weld lengths varying from 10 mm to 40 mm were made. The welds were centered on the sheets. Some of these specimens are shown in Figure 1.

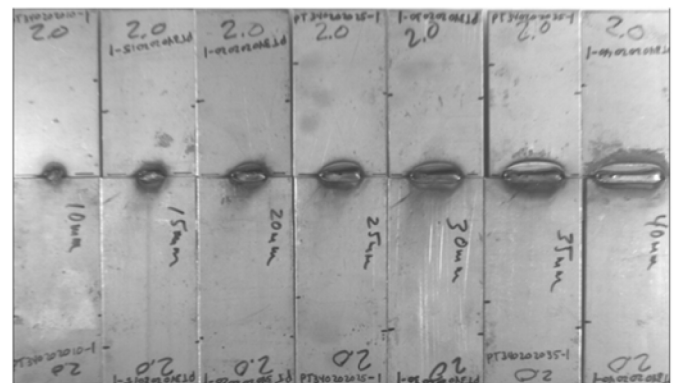


Figure 1. Wide lap-shear specimens with the weld length varying from 10 mm to 40 mm.

For the finite element analyses in this investigation, the cross sectional geometry of the weld metal used to represent a realistic weld was designed to match the micrograph of the cross section of a specimen with a weld length of 25 mm as shown in Figure 2. The

specimen has the sheet thickness of 2.0 mm. The pre-existing crack tip at the interfacial plane of the two sheets at the root of the weld is labeled in the figure.

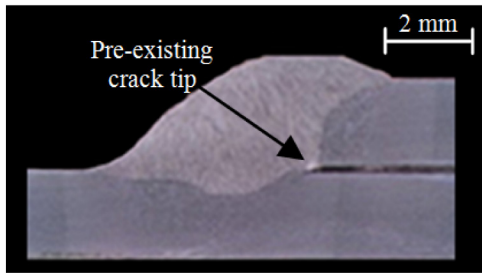


Figure 2. The micrograph of the cross section of a specimen with a weld length of 25 mm and a sheet thickness of 2.0 mm.

ANALYTICAL STRESS INTENSITY FACTOR SOLUTIONS

A schematic of a lap-shear specimen near a realistic weld is shown in Figure 3(a) where the shaded area represents the weld metal. Both the upper and lower sheets have the same thickness t . The weld width is represented by w in the figure. The geometry of the weld shown in Figure 3(a) is idealized to a square shape that represents two joined beams as shown in Figure 3(b). The protrusion and shape of the weld metal, as shown in Figure 3(a), are not considered in the analytical solutions in this paper. As shown in Figure 3, F/b represents the lap-shear force per unit weld length applied along the interfacial surface. The weld metal is assumed to have the same elastic properties as those of the base metal.

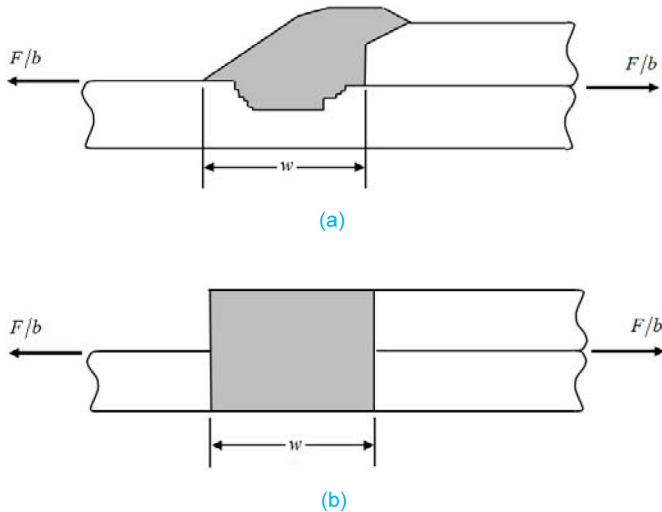


Figure 3. A schematic of a lap-shear specimen near (a) a realistic weld and (b) an idealized weld.

The loading condition shown in Figure 3(b) gives different opening and shear modes of the crack. The K_I and K_{II} solutions can be derived from the elastic beam bending theory, as in Sripichai et al. [7] when w/t is large. The analytical stress intensity factor solutions for the lap-shear specimen are expressed as

$$K_I = \frac{\sqrt{3}}{2} \frac{F}{b\sqrt{t}} \quad (1)$$

and

$$K_{II} = \frac{F}{b\sqrt{t}} \quad (2)$$

FINITE ELEMENT ANALYSIS

Two-dimensional, plane strain finite element analyses were carried out in order to obtain the computational stress intensity factor solutions for the pre-existing cracks between the upper and lower sheets near the roots of the welds with realistic weld geometries. Figure 4(a) shows a schematic of a two-dimensional finite element model of a lap-shear specimen. The specimen has the sheet thickness t , overlap length g , weld width w , weld height h , and weld toe angle, θ . Both the upper and lower sheets have the same thickness. The coordinate system is shown in the figure. The left edge has a fixed displacement condition at the upper surface while the right edge has a concentrated force per unit width, F/b , applied at the lower surface in the $+x$ direction.

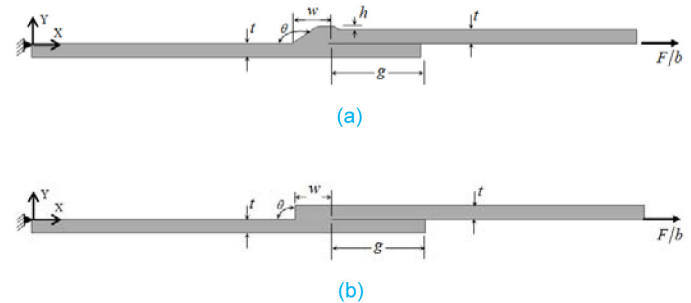


Figure 4. (a) A schematic of a two-dimensional finite element model of a lap-shear specimen with a realistic weld and the boundary conditions. (b) A schematic of a two-dimensional finite element model of a lap-shear specimen with an idealized weld and the boundary conditions.

Computations were performed using the commercial finite element code ABAQUS Standard v611 [8]. The weld metal, heat affected zone, and base metal are assumed to be linear elastic with Young's modulus $E = 207$ GPa and Poisson's ratio $\nu = 0.3$. Second-order, continuum, plane strain elements (CPE8) are used in the models. Figure 5(a) shows the mesh near the weld. The element size near the weld is about $150 \mu\text{m}$. Figure 5(b) shows the mesh near the preexisting crack tip at the root of the weld. A finer mesh is used near the pre-existing crack tip with the element size of $10 \mu\text{m}$ closest to the crack tip.

The two-dimensional plane strain finite element model has the overlap length $g = 13.4$ mm, the weld width $w = 5.29$ mm, the weld height $h = 0.53$ mm, and the toe angle $\theta = 145^\circ$. In order to obtain the stress intensity factor solutions for various realistic weld geometries, the sheet thickness in the finite element model was varied while keeping the overlap length, the weld width, the weld height, and the weld toe angle constant. Figure 6 shows the meshes near the welds for the finite element models for the specimens with the 1.5 mm, 2.0 mm, and 2.5 mm sheet thicknesses for both the upper and lower sheets.