

# Effect of residual stresses due to laser welding on the Stress Intensity Factors of adjacent crack

G. Labeas<sup>1</sup>, S. Tsirkas<sup>2</sup>, J. Diamantakos<sup>2</sup> and A. Kermanidis<sup>1</sup>

<sup>1</sup>LTSM, Laboratory of Technology and Strength of Materials  
Department of Mechanical Engineering & Aeronautics  
University of Patras, Patras 26500, GREECE

<sup>2</sup>ISTRAM, Institute of Structures and Advanced Materials  
Patron-Athinon 57, Patras 26441, GREECE

## Abstract

In the present work the effect of residual stresses due to laser welding on the Stress Intensity Factors (SIFs) of cracks developing nearby the welded area is studied. The simulation of the welding process and the calculation of SIFs on the cracked structure are performed using an explicit and an implicit Finite Element code, respectively. The developed residual stresses due to the welding of two flat plates by laser are calculated first, using a thermo-mechanical transient analysis. Consequently, a linear elastic analysis is applied on the calculation of SIFs at the crack tips. The calculated results of the welding simulation are verified by comparing the computed angular distortions to the corresponding experimental values. As SIF values can not be experimentally determined due to the existence of the residual stress field, the verification of the fracture mechanics analysis is performed through comparisons between computed and experimental crack opening displacement (COD) values. Concerning both analysis types, a good agreement between experimental and theoretical results is observed.

## 1. Introduction

The welding processes usually cause development of flaws or cracks in engineering structures which are subjected to complex stress fields. Many researchers have indicated by experimental proof that the weld residual stresses can significantly affect the fatigue crack growth rates in welded joints. The prediction of crack growth rate in such conditions is a real engineering problem. In previous studies, a number of analytical and numerical models have been developed to calculate the SIF values of cracked welded structures, however, the residual stress field imposed is rather hypothetical and not predicted by precise numerical simulation of the laser welding process. The studies comprise analytical [1-3], two-dimensional [4] and three-dimensional [5] numerical models predicting the effect of an hypothetical residual stresses field on the SIF values. Unfortunately, the above models do not have the capability of predicting the real welding produced residual stresses and take into account the real residual stress field into the SIF calculations.

In the present study, a numerical algorithm employing two different FE codes is developed for the prediction of the effect of the laser welding residual stresses on the SIF values of a cracked welded plate. Simulation of the residual welding stresses is performed initially and the simulation results are introduced in a fracture mechanics analysis for the calculation of the SIFs of cracks adjacent to the weld area. For the application of the proposed algorithm, the basic parameters required are the plate dimensions, the laser power, the laser travel speed, the laser beam diameter, the material absorption coefficient, the temperature dependent material mechanical and thermal properties and the material phase transformation data. The computed results include: (i) the time-dependent temperature distribution; (ii) the time-dependent stresses, strains and distortions; (iii) the residual stresses and strains; and (iv) the final shape and (v) the Stress Intensity Factors of the cracked plate.

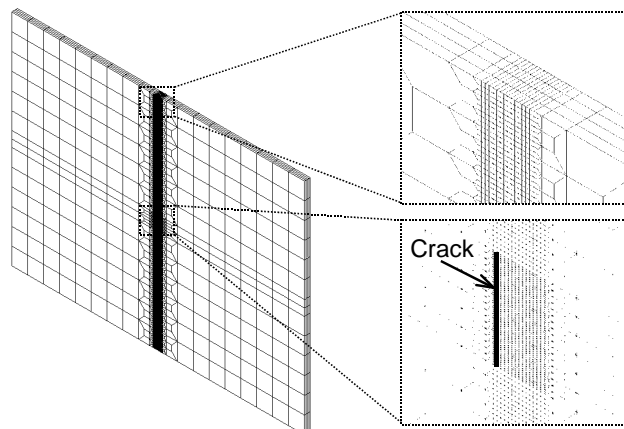
## 2. Numerical analysis

For the fracture mechanics design of welded plates, the computation of SIFs of cracks developing nearby the welded area is required, by taking into account the residual stress field created during the laser welding process. Therefore, two types of numerical analyses are required. In the present work the explicit FE code SYSWELD [6] is used for the simulation of the laser welding process and the determination of the residual stress field around the welded area, while the implicit FE code ANSYS [7] is applied afterwards, for the calculation of SIFs, as well as, stresses and deformations at the cracked structure.

The FE model is initially developed using the preprocessor of ANSYS code. The entities of the FE model are transferred to SYSWELD code by means of a properly transformed ASCII file, which contains the topology of the model (nodes and elements). Upon simulation of the laser heating and cooling process in SYSWELD, the computed residual stresses are exported from SYSWELD and are imported as initial stresses in ANSYS, through an in-house developed interface. The methodology followed utilizes the same FE mesh for both numerical analyses, therefore, it ensures excellent cooperation between the two models. However, both the welding simulation and the fracture mechanics problem are solved with the same FE mesh, although it would be beneficial to refine each mesh separately, according to the specific requirements of each analysis type.

For the entire FE model the 'SOLID45' element type of ANSYS code is used. SOLID45 is a brick element and is defined by eight nodes, having three degrees of freedom at each node, i.e. translations in the nodal x, y, and z directions. During the creation of the model special consideration is taken into account in order to achieve a fine mesh density at the areas of the laser beam welding, as well as, at the crack tips, which is necessary for the successful laser welding simulation and SIFs calculations, respectively.

The developed FE model is presented in figure 1. In the same figure two details are shown, one at the laser beam track around the welded plate edge and one at the crack region. The model comprises 35380 nodes and 27888 elements.



**Figure 1:** Finite element model of the welded plates

### 2.1 Laser welding simulation

During the laser welding process, many mechanisms are taking place in the material of the weldments. A very narrow zone under the laser beam is suddenly heated, vaporized and locally fused. After welding and cooling of the melt material, the assembly of the welded pieces is achieved. On the exposed area a keyhole is shaped. The elevated temperature gradients developed in this area during both heating and cooling phase, along with the sharp decrease of mechanical properties during heating, yield to non-homogeneous permanent strains and residual stresses after the process.

A three-dimensional (3D) finite element model is developed to simulate the laser welding process for the butt-joint of two square plates 150 mm length and 4mm thickness, made of shipbuilding steel AH36 [8], using the commercial code SYSWELD. For the laser welding 2000 Watt power and 300 mm/min laser travel speed, was used.

The geometry of the weld structure is modeled using the pre-processor of ANSYS code. Three-dimensional volume elements having eight nodes and three degrees of freedom per node are utilized. A dense mesh is used in the area along the weld line and at the crack tip regions, as shown in figure 1, while a coarser mesh for the rest of the structure.

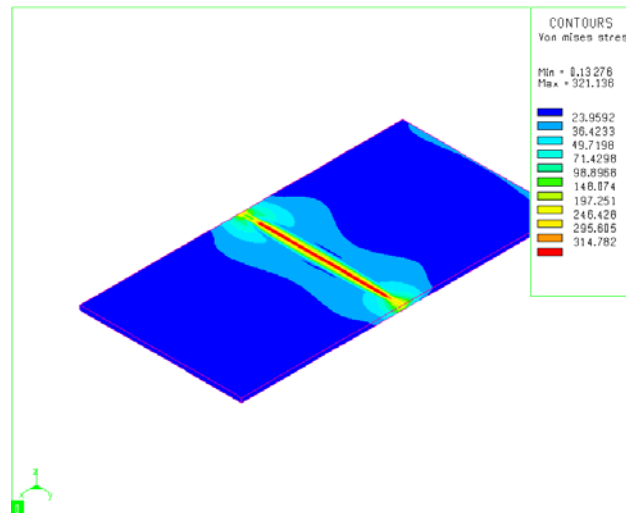
A thermo-elasto-plastic analysis associated with metallurgical transformations is performed using the SYSWELD finite element code enhanced with user subroutines. The solution is generated in two basic steps. First a transient heat transfer analysis is performed and the resulting temperature field is used as input for the second step, which is the mechanical analysis. An appropriate time stepping scheme is applied for each analysis in order to achieve fast convergence of the solution and reasonable accuracy.

The thermal analysis is conducted using temperature dependent thermal material properties, provided by [8]. A metallurgical analysis, based on the material phase transformations laws is applied, in order to simulate the phase transformations taking place during the material heating and cooling due to welding process. During the welding, heat is supplied to the weld pool by the laser beam, which is transferred to the metal by conduction and convection. A part of this heat energy is lost by free convection and radiation.

The heat input to the weld is generally calculated from the energy supplied. The heat input distribution determines the size and shape of the weld pool. In order to simulate the heat distribution and flow in the welding direction, the laser beam is modelled as a three-dimensional moving heat source. The model of the heat source assumes a Gaussian heat flux distribution on the weld pool and the shaped keyhole is simulated by a cone [8, 9]. The heat flux is implemented into the FE code by developing a FORTRAN subroutine.

The heat loss by free convection follows Newton's law, where the coefficient of convective heat transfer is assumed to vary with both temperature and orientation of the boundary [8, 9]. Heat loss due to thermal radiation between the weldments and the environment are important where the temperature difference is high. This radiation is modelled by the standard Stefan-Boltzman relation [8, 9]. The heat loss is implemented into the FE code by developing a FORTRAN subroutine. Prior to welding the material is assumed to be at room temperature.

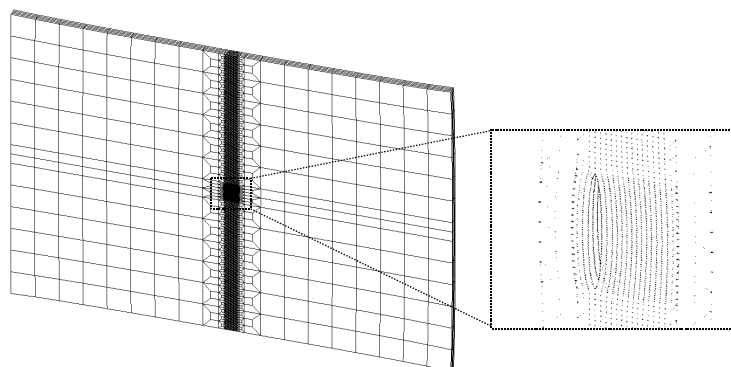
The results of the thermal analysis, i.e. the temperature distributions, are inputted to the mechanical analysis. Proper kinematic boundary conditions to represent the clamping of the plates are introduced in the model. The mechanical analysis is performed using, an elasto-plastic material formulation and the Von-Mises yield criterion coupled to a kinematic hardening rule. The temperature dependent mechanical material properties are introduced in the FE code in a suitable table form. The residual stresses distribution computed as computed from the simulation of the laser welding process is shown in figure 2.



**Figure 2:** Distribution of residual stresses from the laser welding simulation

## 2.2 SIFs calculation

After the two flat plates are welded together, a central notch is created by means of saw cut, which is then pre-fatigued so that cracks start developing at the notch edges. When the total crack length reaches 20mm, Compact Tension specimens are cut from the welded structure and the Crack Opening Displacement (COD) is measured by extensometers. To verify the measured COD and consequently the numerical fracture mechanics analysis, a FE model of the Compact Tension specimens is used for the computation of the COD, SIF and strains – stresses, under the mechanical loading. A comparison between experimentally measured and numerically computed COD values, has shown a good agreement, thus verifying the success of both the residual stress field computation, as well as, the success of the fracture mechanics analysis, as the COD value is dependent on both. Therefore, reliable computation of Stress Intensity Factors may be performed, for which the displacement extrapolation method is applied, using a fit of the nodal displacements in the vicinity of the crack, according to the  $1/(r)^{1/2}$  approximation. Having verified the numerical methodology for SIF computation in the presence of residual stresses, SIFs of the cracked welded plates of figure 1 are numerically calculated. In order to take into account the residual stress field in the SIF calculations, the residual stress values computed from the simulation of the laser welding process are imported to the FE model in the form of initial stresses. External forces vertical to the crack line are applied at the edges of the plate. A linear elastic analysis is then conducted leading to the calculation of stresses and deformations of the plate. The deformed shape of the plate is presented in figure 3, including a detail view of the cracked region.



**Figure 3:** Deformed shape of the plate

To assess the effect of residual stresses on the SIF values, the fracture mechanics analysis is repeated, without taking into account the initial stresses. Calculated SIFs under a residual stress field are compared with respective ones computed for the plate without any residual stresses, in Table 1. A big influence of the residual stress field on the numerical value of the SIF may be observed.

	Calculated SIF (MPa mm <sup>1/2</sup> )
<b>Residual stress field neglected</b>	340
<b>Residual stress field considered</b>	710

Table 1: Calculated SIFs with / without the effect of the residual stress field.

### 3. Conclusions

A numerical methodology is presented for the calculation of the effect of residual stress field on the Stress Intensity Factor of cracks around the welded area of a structure. Both the welding simulation and the fracture mechanics analysis are performed via Finite Element analysis, using two different FE codes.

The numerical COD results are verified using experimental testing. A good agreement between numerical and experimental results has been achieved, thus indicating the successful implementation of the numerical methodology.

### 4. References

- [1] D. Stefanescu, L. Edwards, M.E. Fitzpatrick, Stress intensity factor correction for asymmetric through-thickness fatigue cracks at holes, *International Journal of Fatigue*, 25, 2003, pp. 569-576.
- [2] S. Pommier, C. Sakae, Y. Murakami, An empirical stress intensity factor set of equations for a semi-elliptical crack in a semi-infinite body subjected to a polynomial stress distribution, *International Journal of Fatigue*, 21, 1999, pp. 243-251.
- [3] Nguyen, T. Ninh and M. A., Wahab, The effect of residual stress and weld geometry on the improvement of fatigue life, *Journal of Materials Processing Technology*, 48, 1995, pp. 581-588.
- [4] R.J. Dexter, P.J. Pilarski, H. N. Mahmoud, Analysis of crack propagation in welded stiffened panels, *International Journal of Fatigue*, 25, 2003, pp. 1169-1174.
- [5] Krishnakumar Shankar, Weidong Wu, Effect of welding and weld repair on crack propagation behaviour in aluminium alloy 5083 plates, *Materials and Design*, 23, 2002, pp. 201-208.
- [6] SYSWELD: User's manual, ESI Group, 2001.
- [7] ANSYS: User's manual, ver. 7.1, Swanson Analysis Systems Inc., 2004.
- [8] S.A. Tsirkas, P. Papanikos, Th. Kermanidis, Numerical Simulation of the Laser Welding Process in Butt-joint specimens, *Journal of Materials Processing Technology*, 134(1), 2003, pp. 59-69.
- [9] S.A. Tsirkas, P. Papanikos, K. Pericleous, N. Strusevich, F. Boitout, J. M. Bergheau, Evaluation of distortions of laser welded shipbuilding parts using a local-global finite element approach, *Science and Technology of Welding and Joining*, 8(2), 2003, pp.79-88 .