Simulation-based numerical optimization of arc welding process for reduced distortion in welded structures

M. Islam a,*, A. Buijkb, M. Rais-Rohani a,c, K. Motoyama a,c

a Center for Advanced Vehicular Systems, Mississippi State University, MS 39762, USA
b Simufact-Americs LLC, Plymouth, MI 48170-4347, USA
c Department of Aerospace Engineering, Mississippi State University, MS 39762, USA

A R T I C L E   I N F O

Article history:
Received 11 November 2013
Received in revised form 29 January 2014
Accepted 11 February 2014

Keywords:
Welding simulation
Numerical design optimization
Finite element analysis
Genetic algorithm
Welding-induced distortion

A B S T R A C T

This paper presents an effective numerical approach for welding process parameter optimization to minimize weld-induced distortion in structures. A numerical optimization framework based on coupled Genetic Algorithm (GA) and Finite Element Analysis (FEA) is developed and implemented for a low and a high fidelity model. Classical weakly coupled thermo-mechanical analysis with thermo-elasto-plastic assumptions is carried out for distortion prediction of numerical models. The search for optimum process parameters is executed by direct integration of numerical models and GA-based optimization technique. The developed framework automatically inserts the process parameters into the simulation models, executes the FE-based welding simulations and evaluates the required simulation output data for iterative evolutionary optimization. The optimization results show that the proposed approach can contribute substantially to enhance final welded product quality while facilitating and accelerating the product design and development.

© 2014 Elsevier B.V. All rights reserved.

1. Introduction

Arc welding is a major joining process used in automotive, shipbuilding, and other industries. It is prominent over other joining methods due to its competitive advantages such as reduced cost, enhanced joint strength, and wide range of applications. However, one of the major problems of welding is weld-induced distortion in the welded assembly. Distortion affects performance of welded structures in the form of reduced joint strength and dimensional accuracy. Despite tremendous development in arc welding technology over the years, weld-induced distortion is still one of the major obstacles for welding industry to ensure adequate reliability of welded structure’s performance.

Correction of distortion often requires additional after-weld reworks, which are usually costly, time consuming, and practical only in the most crucial applications. The best practice to minimize or control distortion is proper welding process design through careful selection of various welding input parameters. Several process parameters influence welding distortion. Better control of these parameters will eliminate the conditions that promote distortion [1].

Control of distortion is mostly performed empirically using experiments [2]. A set of experiments is conducted in a defined range of conditions and the experimental results are used to determine the set of parameters that closely meets the joint requirements. However, this type of experimentally determined optimal criteria does not always guarantee the most ideal setting or applicability to the tested conditions.

In recent years, rapid growth in computing power and numerical algorithms has made it possible to model real-world welding processes through computer simulations. For more than three decades, finite element method (FEM) has been the most popular and powerful tool for simulating the thermomechanical behavior of a structure during welding process [3]. Several researchers investigated the problem of distortion during welding using FEM. Lindgren has provided an extensive review on finite element (FE) modeling for welding residual stress and distortion prediction [4–6]; the initial welding simulations were highly simplified based on two-dimensional (2D) approach and plane strain condition [4]. Although the 2D analysis gives indications of the welding residual stresses involved in quasi-static, plane strain conditions, it does not provide the total out-of-plane deformations [7]. Brown and Song [8] used both two 2D and three-dimensional (3D) models to investigate fixturing impact on large structures and concluded that full 3D models are essential in predicting welding distortion. Michaleris et al. [2,9] used two step numerical analysis approach that combines 2D welding simulation with 3D structural analysis for predicting buckling distortion; 2D welding simulation substantially reduces the computation time of their analysis. Tsai et al. [1]
investigated distortion mechanism and the effect of welding sequence on panel distortion using FEM based on inherent shrinkage method. Jung and Tsai [10] developed a plasticity-based distortion analysis and applied it to investigate the relationship between cumulative plastic strains and angular distortion in a fillet welded T-joint. Camilleri et al. [7] proposed a method to improve the computational efficiency of generic FEM-based distortion prediction technique; they modified the full transient thermo-elasto-plastic analysis into an uncoupled thermal, elasto-plastic and structural treatment. More recently, Deng et al. [11–14] conducted substantial research on predicting welding distortion of welded structures. They have effectively used thermal elastic-plastic finite element analysis (FEA) to predict welding distortion in small or medium welded structures. However, the authors concluded this method is inapplicable to simulate the welding distortion for large welded structures because of the large amount of computational time [15]. They proposed an elastic FEM to predict welding distortion in large structures considering both local shrinkage and gap. Murakawa et al. [16] extended the application of inherent strain theory and interface element formulation to compute distortion in thin plate structures. They developed a practical distortion prediction system to compute the accumulated distortion during the welding assembly process based on inherent strain theory and interface element formulation. All these works have provided important and useful information about weld induced distortion phenomena. However, the advantage of knowledge associated with distortion mechanism can be augmented tremendously if FE-based welding simulations are implemented with numerical optimization techniques. The integration of numerical optimization and welding simulation makes it possible to find optimum process parameters computationally with less time and cost.

Optimization of welding process to minimize weld-induced distortion in final structure has been an active research area for several decades. Two optimization approaches (i.e., experimental and computational) can be implemented to determine the optimum welding conditions. The first approach, where actual welding experiments are used, still dominates the published literature. An extensive review of experimental optimization of welding process can be found in Refs. [17,18]. In experimental optimization, DOE, Taguchi method, Evolutionary Algorithms and Artificial Neural Networks are the most frequently used methods.

In numerical optimization, computational models replace the expensive experimental evaluations. As such, design optimization can be conducted using computers rather than real experiments [19] for determining optimum structural sizing, shape and topology parameters in automotive and aerospace industry [20]. In welding industry, this approach is yet to be adopted in full scale. Very few research works have been conducted in the domain of welding optimization via integration of welding simulation and optimization algorithm. Kadivar et al. [21] linked Genetic Algorithms (GA) with a transient 2D FE model to determine optimum welding sequence of a circular pipe for minimization of distortion. Song et al. [22,23] investigated sensitivity of thermo-mechanical responses of welded joints to variation in material properties and optimized quasi-static weakly coupled thermo-elastic-plastic FE process for side heater design. Zhang et al. [24] and Ertas et al. [25] conducted FE based design optimization of spot welded structures under maximum fatigue life considerations.

Furthermore, most of the research conducted in this domain is devoted to investigate the influence of specific parameter variation on welding distortion and residual stress. Teng et al. [26] performed thermo-elastic-plastic analysis using FEM to evaluate residual stress and angular distortion in T-joint fillet welds and analyzed the effect of flange thickness, welding penetration depth and restraint condition on angular distortion. Gannon et al. [27] used FEM to study the influence of welding sequences on the distribution of residual stress and distortion generated when welding a flat-bar stiffener to a steel plate. Schnk et al. [28,29] studied the influence of the clamping time, the release time and the influence of clamp preheating on welding distortion.

It is believed that numerical design optimization has the potential to improve not only the manufacturing side of welding process but also the design side as well. The present study focuses on this topic. Specifically, the objective of this paper is to investigate the implication of numerical approach for design and optimization of industrial welding processes to minimize weld induced distortion in welded structures. The proposed computational approach is to employ a global optimization method through GA in conjunction with FEM for welding process design.

In this paper, a coupled thermo-elastic-plastic FE modeling approach is adopted for welding simulation. Two 3D FE models are developed for a plate lap joint specimen and an automotive structure specimen. The developed models are analyzed through FE-based thermo-mechanical welding simulations. The simulation results are presented for the temperature time history and distortion pattern of the structures. Since the ultimate objective of this paper is distortion minimization through numerical optimization, discussion of residual stress generation is beyond the scope of current study. Next, the models are integrated with optimization algorithm to establish an automated and iterative optimization system. The system is subsequently implemented to perform numerical optimization of the two test problems. Weld-induced distortion is set as objective function (that will be minimized) and minimum weld quality requirement in the form of sufficient weld fusion or melting is set as design constraint. The important parameters including welding speed, input current, arc voltage and welding direction or sequence, etc. are treated as design variables during the analysis and bounded by upper and lower limits. The optimization results are presented and discussed.

2. Numerical optimization framework

The proposed numerical design optimization system consists of four computer programs: (1) an optimization program, (2) a simulation input generation program, (3) a welding simulation program and (4) a simulation output evaluation program. The structure of the system is illustrated in Fig. 1. The four programs are integrated sequentially in a closed loop to establish the automatic and iterative optimization system.

The optimization program is the main controlling program of the system. It executes GA as optimization solver. It also takes important decisions of triggering other three programs when required and stopping the analysis by checking the stopping criteria in each iteration. Furthermore, it also keeps records of results, model information and constraint violations. MATLAB® GA solver is used as the optimization tool. A customized GA variant is developed by varying different default solver properties to meet the problem-specific requirements.

Welding simulation program is a commercial FE-based welding simulation program named simufact.welding®. It executes the welding simulation based on input file and stores the desired output results. It has the ability to simulate welding process models with multiple welding robots working at the same time or different times, flexibility to define or modify welding parameters, paths, directions and simple automatic batch running option useful for simulation-based numerical optimization.

In numerical optimization, the objective function and constraints are evaluated by numerical methods such as FEA. In such cases, the mapping from design variables to objective function and constraints is strictly implicit. Therefore, conversion of the original simulation model and its responses into standard mathematic function values recognizable by the optimization algorithm is
required. For this purpose, two computer programs are developed in the MATLAB® programming environment and embedded inside the optimization framework.

The programs are named as ‘Simulation Input Generator (SIG)’ and ‘Simulation Output Evaluator (SOE)’. Both programs serve as interpreters between optimization and welding simulation program. Simulation input generation program or SIG takes new values of design variables as input, adjusts those values in the simulation input file and passes the updated input file to the welding simulation program as output. SOE reads the simulation output files, extracts the specified results and feedbacks the optimization program with those extracted results. The optimization program uses the extracted results to produce next generation, and in this way, the analysis loop repeats until stopping criterion of the system is satisfied.

3. Finite element modeling and welding simulation

3.1. Geometry modeling

3.1.1. Lap-joint model

The first test model is a single pass welded lap-joint specimen of two thin steel plates. The plate dimensions are $170 \times 35 \times 3.2$ mm and the weld length is 70 mm at the approximate middle section of the plates. FE model for this case study was validated by experimental results. The experimental setup and experimental sample specimen have been illustrated in Fig. 2. A full 3D FE model of the specimen was developed and the necessary correlations were implemented to make the model as accurate as possible. The entire FE model of the lap-joint specimen consists of three geometries: two to represent the base plates and one for weld bead as shown in Fig. 3. As mechanical boundary conditions, four clamps (red parts in Fig. 3) were used on the top surfaces of the plates and the holding force of clamps was set equal to 500 N.

The model contains 6840 eight-node hexagonal elements and 10,347 nodes. To reduce computational time, adaptive meshing was implemented to refine the mesh in the vicinity of weld bead. A refinement level of two is used and heat source area is treated as refinement criterion. The criterion is set by means of a scaling factor (equal to 2), which is a multiplier of the heat source size for the local refinement area around the heat source.

3.1.2. Lower arm model

The next model is an automotive suspension part (lower arm) with nonlinear welding path and weld length of 160 mm. The lower-arm FE model consists of three geometries: two to represent the curvilinear parts to be joined and one for weld bead as shown in Fig. 4. However, the weld bead is subdivided into three sub-welds as shown in the zoomed views of the fillet element in Fig. 4. Four clamps were used for this model as well. The lower-arm FE model contains 15,405 eight-node hexagonal elements and 31,816 nodes.

3.2. Thermal analysis

The conservation of energy is the fundamental principle in thermal analysis of welding [3]. Therefore, during thermal analysis, stress, strain, and displacement are ignored and only energy is considered. In this step, temperature history at each node of the whole FE model is calculated by the transient heat conduction equation given as

\[ \rho C_p \frac{\partial T}{\partial t} = \nabla \cdot q + Q \]

where $\rho$ is the density of the material (g/mm$^3$), $C_p$ is the specific heat capacity (J/(g °C)), $T$ is the current temperature (°C), $q$ is the heat flux vector (W/mm$^2$), $Q$ is the internal heat generation rate (W/mm$^3$), $t$ is the time (s), and $\nabla$ is the spatial gradient operator ($\partial / \partial x$, $\partial / \partial y$, $\partial / \partial z$).

The heat flux vector is defined by Fourier’s law for isotropic material as follows

\[ q = -k \nabla T \]

where $k$ is the temperature-dependent thermal conductivity matrix (W/mm °C) and $\nabla T$ is the temperature gradient.

Typically, the complex physics of heat generation or weld pool is simplified considerably and replaced by a heat input model [30]. In this study, a double ellipsoid heat source model, first proposed by Goldak et al [31], is used to simulate the arc welding heat input. This heat input model combines two ellipsoidal heat sources to achieve the expected steeper temperature gradient in front of the heat source and a less steep gradient at the trailing edge of molten pool [31]. The power density distribution or heat generation rate

![Framework of numerical design optimization system.](image-url)
for each ellipsoidal heat source is given as

\[ Q = \frac{6\sqrt{3}Q_\text{in}t}{bd\pi \sqrt{\pi}} \]

(3)

where \( a, b \) and \( d \) represent the length, width and depth of each ellipsoidal heat source, respectively, \( f \) is the fraction factor defining the fraction of heat deposited in each heat source. \( Q_\text{in} \) is the input power which is determined from input current, arc voltage and arc efficiency. Physically, the width and depth correspond to the weld bead cross-section dimensions. As such, the heat source dimensions are adjusted to obtain the correct heat flux input and correct shape of the melted zone. The heat source parameters used for both models in the current analysis are shown in Table 1.

To solve the differential equation in Eq. (1), boundary conditions must be defined. The boundary conditions may be prescribed temperature or prescribed heat flux. In this study, the latter is used. By including convective, contact and radiation heat losses, the flux equation is written as

\[ q_n = h(T - T_\infty) + \varepsilon\sigma(T^4 - T_\infty^4) + \alpha(T - T_{\text{contact}}) \]

(4)

where the first term is convection heat loss and \( h \) is the convective heat transfer coefficient. The second term is emissive heat loss and \( \varepsilon \) is the emissivity factor. The third term is the contact heat loss and \( \alpha \) is the contact heat transfer coefficient. The relevant parameters are given in Table 2 and the same parameters are used in both models. The material model used for both models is ASTM A591 sheet metal steel. Temperature dependent thermal and mechanical properties are considered as shown in Fig. 5.

3.3. Mechanical analysis

In mechanical analysis, the basic equations are the equilibrium equations, constitutive stress–strain relations and geometric compatibility equations [32]. Temperature history obtained from thermal analysis is input as thermal loading into the structural model to calculate the stress–strain field. The change in the temperature distribution contributes to the deformation of the body through thermal strains and influences the material properties. Large deformation and large plastic strains are accounted for and the additive decomposition of elastic, plastic and thermal strain contributions is utilized for the stress recovery process. The analysis uses a regular elasto-plastic material model with von Mises yield criterion. Phase transformation effects are not considered in the current analysis. The modeling of the fluid flow is also not included because the effect of the fluid flow on the deformation and stress field can be considered as negligible [3].

3.4. Welding simulation approach

The welding simulation is conducted using the built-in MSC Marc solver of simufact.welding®. Thermal analysis is performed separately from mechanical analysis with the assumption of weakly coupled analysis, that is, temperature history is not affected by stress and strain. Marc uses a staggered solution approach in coupled thermo-mechanical analysis where it first performs thermal analysis, then mechanical analysis. It is convenient, as the thermal analysis does not significantly increase total computational time. The dynamic creation of fillet material is achieved by the deactivated element method where elements are first deactivated along the weld path, then revived as the moving heat source takes position within the elements.

3.5. Simulation results

3.5.1. Lap-joint model

The simulation time required to run the complete coupled thermo-mechanical analysis for the lap-joint FE model is approximately 2 h using a desktop computer with 2.30 GHz Intel (R) Core (TM) i5-2410M CPU with 8 GB Ram. The welding parameters used are 200 A, 20.5 V and 10 mm/s. The total simulation time is 137 s, 7 s for welding time and 130 s for cooling. The main driving force in welding simulation is heat generation process. Lindgren [32] stated that if the weld pool boundary is correct, then temperature field outside this region will also be correct. Therefore, the heat source model was validated with respect to the weld macrograph of experimental weld cross-sections and a fairly good agreement was achieved in terms of weld pool boundary shape and size as shown in Fig. 6. The red zone indicates that temperature of the portion is above the melting point.

After validating the heat source, the simulation predicted out-of-plane distortion was compared with the experimental results...
During the experiments, only out-of-plane distortion was measured. In experiment, the maximum positive out-of-plane distortion occurs in the middle section along the edge of the lower plate and its magnitude is 0.53 mm. The maximum negative out-of-plane distortion is 0.401 mm. The maximum out-of-plane distortion obtained by simulation was 0.49 mm and 0.35 mm, respectively in positive and negative z axis.

To compare out-of-plane distortion quantitatively with the experiments, three different lines along the two edges and along the midsection of the weld were considered. The corresponding distortion plots are shown in Fig. 8(b)–(d), respectively. It is worth mentioning that the laser scanner used in the experiments recorded thousands of data points over the surface with a reference frame not identical with the simulation model. The experimental data points were picked randomly and thereby some deviation from the exact line has occurred during manual point selection. Although there is some discrepancy between the experimental and simulation results, the general trend of the plots indicates that the simulation predictions are sufficiently accurate on the basis of quantitative comparison as well.

Furthermore, total distortion pattern obtained by simulation has been shown in Fig. 9. The total distortion prediction by simulation also confirms that out-of-plane distortion (z axis) is the main contributing factor for distortion in the structure. Maximum distortion (0.58 mm) has occurred at middle section of the lower plate as well. Since total distortion could not be measured during experiment, simulation predicted total distortion has been used as reference in later section of the paper.

### Table 1
Heat source parameters.

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Lap joint model</th>
<th>Lower arm model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Front length, (a_f) (mm)</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>Rear length, (a_r) (mm)</td>
<td>2.50</td>
<td>1.00</td>
</tr>
<tr>
<td>Width, (b) (mm)</td>
<td>3.25</td>
<td>2.50</td>
</tr>
<tr>
<td>Depth, (d) (mm)</td>
<td>4.00</td>
<td>2.25</td>
</tr>
</tbody>
</table>

### Table 2
Heat transfer coefficients.

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Convective coefficient, (h) (W/m²·K)</td>
<td>20</td>
</tr>
<tr>
<td>Contact coefficient, (a) (W/m²·K)</td>
<td>100</td>
</tr>
<tr>
<td>Emission coefficient, (\varepsilon)</td>
<td>0.6</td>
</tr>
</tbody>
</table>

The maximum negative out-of-plane distortion is 0.401 mm. The maximum out-of-plane distortion obtained by simulation was 0.49 mm and 0.35 mm, respectively in positive and negative z axis.

3.5.2. Lower-arm model

The simulation time required to run the complete thermo-mechanical analysis for the lower-arm FE model was approximately
4 h on the same PC used for the lap-joint model analysis. It should be noted that this model could not be validated due to unavailability of any relevant experimental or simulation works on this particular model. However, based on the lap-joint model validation experience, a good effort was made to make the simulation model as accurate as possible. The welding parameters used are 18 volts, 180 amp and 15 mm/s welding speed for this case study. The total simulation time is 50 s, with welding time of 10 s approximately and cooling time of 40 s. As the first step of modeling, the heat source model was calibrated to obtain sufficiently accurate weld pool shape.
The weld bead was modeled as triangular shaped with the resultant molten weld pool shape for sub-weld 1 depicted in Fig. 10 by dotted line.

Fig. 11 shows the typical distortion distribution over the structure. The distortion pattern of the structure indicates that the lower part has undergone higher distortion than the upper part and the maximum distortion has occurred in the middle section of the lower part with a magnitude is 0.59 mm.

4. Numerical optimization

4.1. Optimization problem formulation

4.1.1. Lap-joint model

The maximum distortion is treated as the objective function. Through welding simulation, distortions in all \( N \) nodes are first calculated as the sum of square roots of nodal distortions in all three directions. Then, the maximum distortion value is selected and used as the objective function value for iterative optimization via GA. Thus, the objective function is defined as

\[
F(X) = \max(D_i)
\]

\[
D_i = \sqrt{(dx_i^2) + (dy_i^2) + (dz_i^2)} \quad i = 1, 2, 3 \ldots N
\]

Welding speed (X1), arc voltage (X2), input current (X3) and welding direction (X4) are defined as design variables. Details of design variables are shown in Table 3.

For the lap-joint model, X4 can take six numerical values to represent six possible welding directions as shown in Table 4. Two welding directions are designed with one robot and are represented by integer values 1 and 2, depending on the robot’s left-right or right-left movement direction, respectively. Similarly, the remaining four welding directions are designed with two robots and are represented by an integer from 3 to 6 depending on each robot’s left-right or right-left movement direction. For the two robot welding cases, it is assumed that both robots will start and stop welding at the same time.

The optimization algorithm chooses design variables automatically. As such, selected design variables do not always guarantee good welding quality. For example, if the heat input (current \( \times \) voltage) is very low, welding quality will be poor due to incomplete fusion or insufficient weld penetration. Sufficient welding quality is ensured if the temperature in the welding zone is higher than or equal to melting temperatures of base metals and weld beads during welding. As such temperature constraints are required in the optimization problem. During FE simulations, temperatures at three different weld bead cross-sections of both models are monitored to check temperature constraint. Constraint violation has been implemented by adding a penalty term to the objective function \( F(X) \) and the resultant combined function called augmented function \( \phi(X) \) is used as revised objective function. The penalty term is defined as proportional to severity of constraint violations. In this work, the penalty term is proportional to the number of bead element nodes (\( N_c \)) that have temperature less than melting value. Thereby, the augmented function \( \phi(X) \) is defined as

\[
\phi(X) = \begin{cases} F(X), & N_c = 0 \\ F(X) + 100N_c, & N_c > 0 \end{cases}
\]

where \( F(X) \) is the original objective function value and \( 100N_c \) is the penalty term. The penalty term increases the original objective function value and indicates to GA that the associated model is infeasible. An infeasible model represents poor welding quality even though the weld-induced distortion may be small.
4.1.2. Lower-arm model

For the lower-arm model, the objective function definition is also the same as that in Eq. (5). Since the welding path is nonlinear and sufficiently long, the impact of welding order or sequencing is very critical on weld-induced distortion. As such, for this case study, welding sequences are treated as design variables in conjunction with the other process dependent variables. The entire weld path is divided into three sub-welds and the order of occurrence of each sub-weld is treated as design a variable. Therefore, there are six design variables in this case study, the first three are input current, arc voltage and weld speed defined in Table 5 and the latter three representing the sub-weld orders (X4, X5, X6) are shown in Fig. 12.

The design variables associated with sub-weld orders are treated as discrete design variables. Each sub-weld can occur in three different orders and each can have two directions (forward or backward). Thus, each sub-weld design variable has six discrete status values that it can possess as listed in Table 6. For example, physical meaning of sub-weld order for a typical combination is shown in Fig. 12. Total number of combinations (N) possible from three sub-welds is 48 ($2^3 \times 3!$). Similar to the lap-joint model, temperatures at three different weld bead cross-sections are monitored to check the temperature constraints. The corresponding objective function is also converted to the augmented function of Eq. (6).

### Table 5

<table>
<thead>
<tr>
<th>Design variable</th>
<th>Lower arm model</th>
</tr>
</thead>
<tbody>
<tr>
<td>X1</td>
<td>Current</td>
</tr>
<tr>
<td>X2</td>
<td>Voltage</td>
</tr>
<tr>
<td>X3</td>
<td>Speed</td>
</tr>
<tr>
<td>X4, X5, X6</td>
<td>Weld order</td>
</tr>
<tr>
<td><strong>Definition</strong></td>
<td><strong>Unit</strong></td>
</tr>
<tr>
<td>Amp</td>
<td>Volt</td>
</tr>
<tr>
<td>20</td>
<td>25</td>
</tr>
<tr>
<td>15</td>
<td>22</td>
</tr>
</tbody>
</table>

4.2. Genetic algorithm based optimization procedure

In numerical optimization, since the mapping from design variables to objective function and constraints is strictly implicit, it is hard to judge whether these functions are continuous and differentiable as the convergence conditions of some optimization methods (i.e., gradient-based methods) may require. Gradient-based methods, thus, may not be appropriate for these optimization problems. Derivative-free techniques such as GA can be easily adopted in these problems. GA requires only objective and constraint function values during the search process. As such, it is very suitable for a general nonlinear optimization problem such as the one considered here.

GA based optimization initiates by creating a set of random candidate solutions called population. Each individual in the population is called chromosome, representing a solution to the problem. For example, a chromosome represents a set of values for welding speed (X1), arc voltage (X2), input current (X3) and welding direction (X4) for the lap-joint model. The chromosomes evolve through successive iterations, called generations. During each generation, the chromosomes are evaluated by welding simulation program. The program stores each individual and its fitness value so as to ensure not to reevaluate twice the same individual in successive generations. To search for better solution, a new generation is formed by selecting, according to objective function values (maximum distortion), some of candidate solutions and forming new prospective candidate solutions. New chromosomes or candidate solutions are formed by either merging two chromosomes from current generation using a crossover operator or modifying a chromosome using a mutation operator. The optimization algorithm runs until the maximum number of generations is reached or the cumulative change in the objective function value over five generations is less than or equal to predefined objective function tolerance. The flowchart of the system algorithm is shown in Fig. 13.

The relevant GA parameters used in the optimization system are given in Table 7. The number of individuals in each iteration or population is 10 and maximum number of generations is limited to 20. Therefore, maximum number of simulations possible in the system is 200. However, due to repetition of promising candidates and consideration of elite candidates in each population, total simulation requirement is always less than the maximum value. A crossover rate of 80% is used and crossover operator is executed using scattered crossover function. Scattered crossover function creates a random binary vector and selects the genes where the vector is a 1 from the first parent, and the genes where the vector is a 0 from the second parent, and combines the genes to produce new chromosome. An adaptive mutation function has been used that randomly generates directions that are adaptive with respect to the objective function value and considers elite candidates in each generation. The fitness calculation involves evaluating each chromosome in the population, and selecting the elite chromosomes. The elite chromosomes are combined at random to create new chromosomes.

### Table 6

<table>
<thead>
<tr>
<th>Design variable Status</th>
<th>Welding order</th>
<th>Welding direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>First</td>
<td>Forward</td>
</tr>
<tr>
<td>2</td>
<td>Second</td>
<td>Forward</td>
</tr>
<tr>
<td>3</td>
<td>Third</td>
<td>Forward</td>
</tr>
<tr>
<td>4</td>
<td>First</td>
<td>Backward</td>
</tr>
<tr>
<td>5</td>
<td>Second</td>
<td>Backward</td>
</tr>
<tr>
<td>6</td>
<td>Third</td>
<td>Backward</td>
</tr>
</tbody>
</table>

![Fig. 12. Symbols of sub-weld design variables.](image)
to the last successful or unsuccessful generation. The mutation chooses a direction and step length that satisfies bounds and linear constraints.

5. Optimization results

5.1. Lap-joint model

Since GA is not a deterministic algorithm, three trials were conducted to determine the optimum parameters for this case study. The optimization results are given in Table 8. The best optimization result is found in trial 2 for the maximum total distortion of 0.46 mm, which is ~21% less than the maximum total distortion found by simulation in the experimental condition (0.58 mm). Furthermore, it is seen that the optimum heat input is 2000 W, which is just 49% of the heat input of the experimental condition (4100 W). The max distortions found with the other two trials are the same (0.48 mm) and very close to the best optimization result. Therefore, GA was successful in determining the optimum set of parameters that would result in the reduced weld induced distortion.

Given the computational expense, the aim is to optimize the process with a maximum of 100 FE simulations. For all three trials, the optimization converged with 10–15 iterations and at the maximum cost of 78 FE simulations in trial 3. The fastest convergence was achieved in trial 1 with 58 FE simulations. For all the case studies, the optimum point was reached within five or six iterations. But the optimization algorithm was designed to run extra five iterations to confirm the validity of the optimum point. Thus, for all three trials, the optimization algorithm stops searching for optimum point when cumulative change in objective function value is less than the predefined limit (10E–06) for five consecutive generations. Fig. 14 shows the optimization result convergence history with respect to the calculation generations or iterations for trial 2. The convergence history also reveals that the z-directional or out-of-plane distortion is the dominant part of total distortion and it is the most sensitive distortion component to the change in design variables considered.

Table 8
Optimization results of lap joint model.

<table>
<thead>
<tr>
<th>Condition</th>
<th>Trial#</th>
<th>Optimum values of design variables</th>
<th>Maximum total distortion, (mm)</th>
<th>No of simulations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optimum (Best)</td>
<td>1</td>
<td>200, 15, 10, 3</td>
<td>0.48</td>
<td>58</td>
</tr>
<tr>
<td>2</td>
<td>200, 10, 5, 3</td>
<td>0.46</td>
<td>63</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>200, 15, 10, 3</td>
<td>0.48</td>
<td>78</td>
<td></td>
</tr>
</tbody>
</table>

5.2. Lower-arm model

The optimization results are illustrated in Table 9. In this case, given the high computational expense, the aim is not to find the optimum result but to evaluate how much distortion can be reduced through the optimization process with a maximum of 100 FE simulations. The best result or smallest distortion found for this case study is 0.35 mm, which is ~41% less than that of the initial condition. Furthermore, it is seen that heat input for the best parameter set is 2870 W, which is 11% less than the heat input (3240 W) of initial condition. Fig. 15 illustrates the best welding sequence found for this case study. Thus, it is also possible to reduce the weld-induced distortion considerably by GA at the cost of 100 FE simulations. For this case study also, GA is successful in reducing the overall distortion of structure in addition to reducing the maximum distortion.
6. Conclusion

Due to increasing requirements for improved performance of welded structures, it has become essential to take into account process variability during the design phase of a welding process. Traditional experiment based welding process variable optimization is quite expensive and is not always guaranteed to provide the optimum parameter combination. Furthermore, such approach cannot also effectively control several critical parameters such as weld quality. In this context, this study introduced a robust numerical optimization system based on integrated computational tools, which allow automatic optimization of welding process parameters without the requirement of expensive experiments. The system is capable of exploring the effects of several design variables at a time with limited modification of the simulation model. Thereby, the developed tool can be effectively implemented for the process design purpose of a large-scale industrial welding process. In addition, the proposed tool will also be a useful tool for performing early-stage design investigations like parametric study or sensitivity analysis. This study also widens the automated and customized applications of CAE tools in manufacturing process design, analysis and optimization.

The illustrative example of lap joint welding specimen optimization presented in this work showed that the proposed GA-FEM coupled method is able to search for optimum set of parameters, especially under the critical constraint of weld quality coupled method is able to search for optimum set of process parameters. This example illustrated that the proposed computational method could be redesigned easily without any major modification in the system when the objective is to find a compromise between optimum design and computational cost.

Although computational efficiency is a critical limitation of the proposed GA-FEM coupled optimization system, it is evident that the methodology is quite successful in converging towards optimum point. To increase the computational efficiency of the developed system, integration of parallel computing facility with the system can be an excellent extension of current work. Consideration of additional design variables such as clamping position, clamp apply/release time and cooling time between sub-welds will also be among the topics for future research.

Acknowledgments

This research is based on the work supported by the US Department of Energy under Award Number DE-EEO002323. Additional support provided by Center for Advanced Vehicular Systems (CAVS), Simufact Americas LLC and F.tech R&D North America Inc. are acknowledged and greatly appreciated.

References

[22] J. Song, J. Peters, A. Noor, P. Michaleris, Sensitivity analysis of the thermo-
[23] J. Song, J.Y. Shanghvi, P. Michaleris, Sensitivity analysis and optimization of
thermo-elasto-plastic processes with applications to welding side heater
[27] L. Gannon, Y. Liu, N. Pegg, M. Smith, Effect of welding sequence on residual stress
575–582.
[29] T. Schenk, I.M. Richardson, M. Kraska, S. Ohnimus, A study on the influence of
[31] J. Goldak, A. Chakravarti, M. bibby, A new finite element model for welding