Xin Long
Sanjeev K. Khanna
Department of Mechanical and Aerospace Engineering,
University of Missouri-Columbia,
Columbia, MO 65211

Numerical Simulation of Residual Stresses in a Spot Welded Joint

An incremental and thermal-electro-mechanical coupled finite element model has been presented in this study for predicting residual stress distribution in a spot welded steel joint. Approximate temperature dependent material properties, including physical and mechanical properties, have been considered. The spot nugget shape and the residual stress distribution were obtained by simulation. The results obtained have been compared with experimental measurements, and good agreement is observed. The highest tensile residual stress occurs at the center of the nugget and the residual stress decreases towards the edge of the nugget. [DOI: 10.1115/1.1543968]

1 Introduction

Spot welding, or resistance spot welding, involves the joining of two or more pieces of sheet metal in localized areas or spots where melting and coalescence of a small volume of material occurs from heating caused by resistance to the passage of an electric current. This process is typically used to obtain a lap joint of sheet metal parts of thickness 0.125 in. (3 mm) or less, using a series of spot welds, in situations where an airtight assembly is not required. A common example is the mass production of automobiles, where a typical automobile may contain more than 5000 spot welds.

When the current is turned off, this volume of molten metal cools down and solidifies, beginning at its outer edges. The volume of metal from the work pieces that has undergone heating, melting, fusion, and resolidification is called the weld nugget. The grain structure in the nugget is considerably coarser than the parent metal. Evidently a spot weld cools down to room temperature non-uniformly. The large temperature gradients created by the intense local heating during the welding process followed by rapid cooling, and also phase changes in the solidifying metal, induce heterogeneous deformations in the metal resulting in the development of internal stresses. These internal or remaining stresses are known as residual stresses.

In order to increase the reliability of products, study on the mechanical properties of the spot welded joint has been attracting a lot of interest [1–6]. It is known that the mechanical properties of a welded joint are not only determined by the microstructure of weld zone metal, but also by the residual stresses introduced by the heterogeneous thermal cycle during welding. Residual stresses play an important role in influencing the fatigue life and other mechanical properties of the spot welded structure. For instance, when the interaction between residual stresses and future loads occurs, the local area that has the highest residual stresses is a potential source for crack initiation and growth in the weld or heat affected zone (HAZ).

There are several methods to measure residual stresses, but most of them are not suitable for a spot welded joint because the spot nugget has a small size (for example, the diameter of spot nugget in the present study is about 5 mm) and it is not easy to be reached (spot nugget exists between the two workpieces). Recently, high sensitivity moiré interferometry [7] and X-ray techniques [8] have been successfully used in the measurement of residual stresses in spot welds, which make it possible to determine the average residual stress distribution in the spot weld. With moiré methods in conjunction with hole drilling, the average through thickness residual stresses are obtained, while the X-ray method gives the residual stresses in the surface layers of the weld, though etching techniques have been used for through thickness stress measurement using X-rays.

With advances in computer hardware and finite element method (FEM) software, numerical simulation now plays an important role in study of manufacturing processes. Since experiments alone cannot easily study the spot welding process, numerical simulation is a potential way to aid in the quantitative study of spot weld residual stress generation. But because of the complexity of the spot welding process, early efforts mainly focused on the heat transfer problem or surface phenomena by mathematical analysis. In 1984, a thermal-electro-mechanical coupled model was introduced by Nied [9]. Temperature dependent materials properties were used in this model. The results in this work show the mechanical deformation of electrodes and workpieces, and an elliptic shaped weld nugget. Since then, coupled spot welding models have been developed by many researchers to focus on different aspects of the spot welding process, such as temperature distribution, nugget growth, electrode design, welding parameters optimization, etc. [10–14]. Since spot welding is a strongly coupled model that cannot be solved directly by the commercial software, some of them used sequentially coupled finite element models [9,10]. A sequentially coupled model simulates the temperature field and stress field in the spot weld in a sequential order. Under this condition, the contact radii between the electrode/workpiece and faying interface are assumed to remain constant during the whole welding process. This is not actually true because contact radii are the result of competition between thermal expansion and electrode squeezing force, thus they vary during the welding process. Some authors used an incrementally coupled thermal-electro-mechanical model [11–14]. That means, with a predetermined small time increment, the contact radii from thermal-mechanical analysis is input to the electrical-thermal analysis as electrical and thermal boundary conditions. Then the temperature distribution from electrical-thermal analysis is input to the thermal-mechanical analysis as a body load. This iterative procedure is applied during the whole welding cycle. The incrementally coupled models were used to simulate the squeezing cycle, welding cycle and sometimes holding cycle during the whole welding cycle. However, to accurately study residual stress formation, the whole welding process, which includes squeezing cycle, welding (heating) cycle, holding cycle, and cooling cycle, should be considered in the model. In Ref. [15], a whole welding process was simulated using a coupled incremental method, and the residual stress distribution in an aluminum spot welded joint was obtained. But, unfortunately, no experimental data was offered for verification.

In this study, an incrementally coupled finite element model has been established to study the whole spot welding process, and used to investigate residual stress generation and distribution in a

Contributed by the Materials Division for publication in the JOURNAL OF ENGINEERING MATERIALS AND TECHNOLOGY. Manuscript received by the Materials Division December 7, 2001; revision received June 20, 2002. Associate Editor: S. Mall.

 Copyright © 2003 by ASME

Downloaded From: http://materialstechnology.asmedigitalcollection.asme.org/ on 01/29/2016 Terms of Use: http://www.asme.org/about-asme/terms-of-use

spot welded steel joint. The residual stress results obtained using the finite element model have been compared to the experimental results obtained previously [7] by employing high sensitivity moiré interferometry. The growth of the spot weld nugget has also been obtained using the finite element. The nugget size was compared with the nugget in an actual spot welded specimen by cutting and etching the nugget cross-section.

2 Numerical Model

The spot welding process involves a complex coupling among thermal, electrical, physical, and chemical processes, which makes it difficult to be modeled. The residual stress simulation of the whole welding process, including squeezing, welding, holding and cooling cycles, has been included in the model presented here. Commercial finite element code ANSYS and the APDL (ANSYS Parametric Design Language) were used to model the coupling between electrical and thermal phenomenon and between the thermal and mechanical phenomenon. The electrode contact and electrode separation problems have also been considered because the electrodes will be removed from the workpiece after the holding cycle.

2.1 Basic Theory. A spot welding model involves electrical field, temperature field, and stress and strain field equations. Due to the axisymmetric nature of the electrode and the workpiece, a quarter of the physical model for two-dimensional analysis is created and is shown in Fig. 1.

All of the equations in this study are based on the column (cylindrical-polar coordinate) coordinate system, as shown in Fig. 1. The governing equation for calculation of the electrical potential φ in the whole model is:

$$\frac{\partial}{\partial r}\left(\sigma\frac{\partial\varphi}{\partial r}\right)+\frac{\sigma}{r}\frac{\partial\varphi}{\partial r}+\frac{\partial}{\partial z}\left(\sigma\frac{\partial\varphi}{\partial z}\right)=0 \quad (1)$$

where r is the radial distance in this column coordinate system, z is the distance in the axis direction of the coordinate system, and σ is the electrical conductivity. By solving Eq. (1), the electrical potential φ is obtained. According to the electric current heat generation (q) rule:

$$q=I^2Rt \quad (2)$$

where I is the current, R is the material electrical resistance, and t is the time for which current is passed. Since $I=\varphi/R$, Eq. (2) can be rewritten as:

$$q=\varphi^2t/R \quad (3)$$

The governing equation for transient temperature field distribution, which involves electrical resistance heat can be written as:

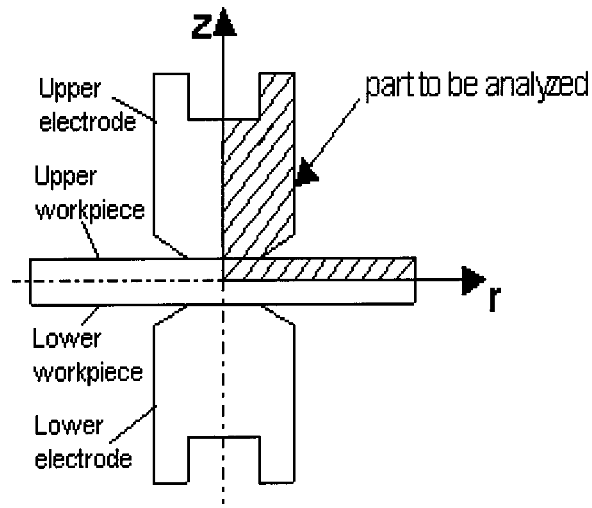
$$\rho c\frac{\partial T}{\partial t}=\frac{\partial}{\partial r}\left(k\frac{\partial T}{\partial r}\right)+\frac{k}{r}\frac{\partial T}{\partial r}+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)+q \quad (4)$$

where ρ is the material's density, c is the heat capacity, T is the temperature, t is the time, k is the thermal conductivity, respectively. The material properties c, k, and σ are temperature dependent (which is further explained in a later section). Substituting Eq. (3) in (4), and solving the resultant equation, the temperature distribution generated by current is obtained.

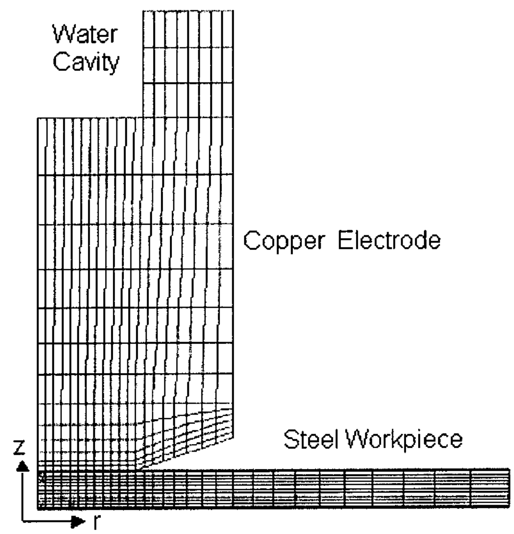
For stress and strain analysis, the governing finite element equation is:

$$H\cdot\Phi+f=0 \quad (5)$$

where Φ is the vector of unknown displacements, f is the vector of loads, and H is the stiffness matrix. Since residual stress is caused by heterogeneous plastic deformation of the material, temperature dependent material mechanical properties were used. Plastic deformation of material was modeled using bilinear kinematic hardening (BKIN) option in ANSYS program, which assumes the total stress range is equal to twice the yield stress and the work hardening is linear.



(*a*)



(*b*)

Fig. 1 (*a*) Axisymetric mode of spot welding setup (*b*) Finite element model for spot welding

2.2 Geometrical Model and Boundary Conditions

Geometrical Model. The geometrical model and the mesh division are shown in Fig. 1. A total of 933 nodes and 835 elements (in case of thermal-mechanical analysis, there were 60 more contact elements) were used in this study. A denser mesh was used for the weld zone and the electrode portion in proximity to the workpiece.

Electrical Boundary Conditions. In the welding cycle, a root-mean-square (RMS) value of total current was applied uniformly on top of the copper electrode. At the faying surface of the workpieces (that is the contact regions between the two sheets being welded), the voltage was set to zero.

Downloaded From: http://materialstechnology.asmedigitalcollection.asme.org/ on 01/29/2016 Terms of Use: http://www.asme.org/about-asme/terms-of-use

Table 1 Mechanical Properties for AISI 1010 Steel and Welded Metal

Temperature (°C)	20	100	200	400	600	800
Elastic modulus (GPa)	200	195	190	170	60	10
Yield Stress (MPa)	185	180	165	112	55	10

Note: Poisson's ratio for AISI 1010 steel is 0.3.

Thermal Boundary Conditions. The temperature at the water cooling cavity was restrained to 25°C, and convection heat transfer to ambient air was specified on all the lateral surfaces of the electrode and workpiece that are not in contact.

Mechanical Boundary Conditions. Uniform load was applied at the top of the copper electrode during welding and holding cycle. The electrode was removed at the end of the holding cycle. At the faying surface between the electrode and the workpiece, a contact element was used. At the faying surface between workpiece and workpiece, the vertical displacement of the part of faying surface under electrode was set to zero, and a subroutine program was used to determine if the other part of faying surface is under contact or not. If some nodes are under contact and are under pressure stress, a zero vertical displacement was applied here.

Materials Properties. Temperature dependent physical and mechanical materials properties, including thermal conductivity, coefficient of thermal expansion, electrical resistance, specific heat, density and latent, elastic modulus, yield stress and Poisson's ratio, were used for both electro-thermal and thermal-mechanical analysis [10]. In this model, the contact resistance between electrode/workpiece interfaces and workpiece/workpiece faying surfaces were considered and simulated by assigning temperature and pressure dependent resistance properties to one layer of elements along the contact interfaces, which has a thickness of 0.01 mm. The value of contact resistance (equivalent contact resistance) was obtained from reference [10]. However, considering the difference of squeeze forces and the materials (sheet size) used in present study, a trial study was used to modify the contact resistance values provided in reference [10]. This contact resistance layer was not considered in the thermal-mechanical analysis.

AISI 1010 steel with 1 mm thickness was used in present study. The sheet steel has room temperature nominal yield strength of 185 MPa [17]. The physical and mechanical properties for AISI 1010 steel workpiece at elevated temperatures were referred in [18–20]. Table 1 shows the approximate mechanical properties for AISI 1010 steel sheet and the welded metal used in present study.

Welding Conditions. In this study, the root-mean-square value of A/C welding current used was 11,000 ampere. The welding cycle had time duration of 12 cycles (0.2 s), the holding cycle had a time duration of 0.2 s (current turned off during this cycle, see Fig. 2), and the cooling cycle had a time duration of 50 s. A squeeze force of 2225 N was used. The tapered flat shape electrode had a 3 mm radius contact area with the workpiece.

2.3 Analysis Flow Chart. Since spot welding is a strongly coupled model that cannot be solved directly by commercial software, an incremental method is adopted in studying the spot welding problem. First, a solid element is used to analyze the initial contact radius (between the electrode/workpiece and the workpiece/workpiece) in the model. Then a thermal-electrical coupled element is used to analyze the temperature field in the same model. The contact information from the thermal-mechanical analysis is input to the thermal-electrical analysis to determine the electrical potential distribution. After that, the thermal-mechanical solution is carried out. The temperature field from the former thermal-electrical analysis is input as a body load.

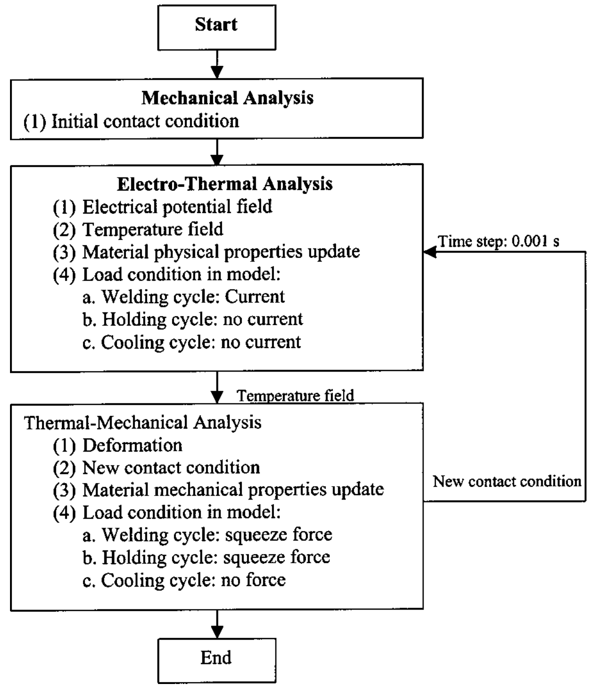


Fig. 2 FEM program flow chart

After finishing the thermal-mechanical analysis, new contact information is gained. Over a given time increment, the interactions between the electrical-thermal and thermal-mechanical analysis are obtained. This iterative procedure is applied during the whole welding cycle. A time increment of 0.001 s was used to ensure sufficient accuracy and good calculation convergence. During the holding stage, the current is set to zero, and convection is the only thermal load in the electro-thermal model. In the cooling stage, the electrode is separated (nocontact), and convection is still the only thermal load in electro-thermal model. The condition in the thermal-mechanical model is a little complicated because there is a stress release and re-equilibrium process in workpieces after separation of the electrodes. Then, the stresses continue changing with the cooling of workpieces in the electro-thermal model. When the temperature decreases to near ambient temperature (25°C), the calculated stress distribution in the thermal-mechanical model is the residual stress in the spot welded joint.

The flow chart of the analysis procedure is illustrated in Fig. 2. Commercial finite element code ANSYS and the APDL (ANSYS Parametric Design Language) were used to deal with the incremental method involving the coupling between the electro-thermal and thermal-mechanical subroutines.

3 Results and Discussion

3.1 Numerical Simulation Results

3.1.1 The Nugget Shape. Figure 3 shows the temperature distribution in the spot welded joint. Considering 1500°C as the melting point of steel, the spot nugget (melted steel) contour appears as curve G-G in Fig. 3. From Fig. 3, it can be seen that the area under G-G has a half-width of 2.20 mm and a half-height of 0.67 mm (the contact resistance layer thickness is not counted). But that is not the final nugget size because there will be some deformation in the welding zone when the weldement cools down due to the electrode pressure and material shrinkage because of temperature change. According to the simulation result, the nugget has an expansion along the width of 0.02 mm, and shrinkage in height of 0.01 mm after cooling down. Therefore, the final simulated nugget size is 2.22 mm in half-width and 0.66 mm in half-height.

Downloaded From: http://materialstechnology.asmedigitalcollection.asme.org/ on 01/29/2016 Terms of Use: http://www.asme.org/about-asme/terms-of-use