Modeling and Analysis of Distortion in Metal-Inert-Gas Arc Welded Automotive Component

Gene Y. Liao
Wayne State University

Abstract

This paper presents the simulation of Metal-Inert-Gas (MIG) arc welded part distortion and its correlation with test for an automotive engine cradle. A Finite Element Analysis (FEA)-based method is developed to predict the welding distortion. The developed technique has elastic-plastic constitutive material model with temperature-dependent material properties. The inherent shrinkage of weld and two weld sequences are simulated. In addition to distortion, residual stress and plastic strain distributions are obtained from the analysis. The heat sink effect is also analyzed using transient thermal-mechanical analysis. The predicted welding distortion is in reasonable accuracy as it correlated with test data.

1. Introduction

Due to many desirable properties of aluminum material (such as low weight-to-strength ratio and excellent resistance to salt water corrosion), the aluminum has been increasingly used in vehicle structures, such as cradle. The cradle is a welded structure to mount engine in a vehicle. Besides extrusion, Metal-Inert-Gas (MIG) welding is a primary manufacturing process for aluminum cradle. The distortion in welding aluminum part is more severe than that of steel because: (1) aluminum has a higher heat conductivity (approximately five times of that of steel), (2) aluminum has a larger coefficient of linear thermal expansion conductivity (approximately three and one-half times of that of steel), and (3) the elastic modulus of aluminum is one third of that of steel.

Since the design and production of engine cradle and its subsequent integration with other vehicle body structures rely on the achievement of tight tolerance, a modeling and analysis of welding distortion can contribute to the precision data-driven manufacturing process. It can be used to mathematically prove various part designs and welding procedures without physical builds. The technique developed in this project meets the industry's need. This project also serves as a case study for the graduate students to predict the quality of a real industrial part using Computer-Aided Engineering (CAE) in manufacturing.

There are two kinds of Finite Element Analysis (FEA)-based MIG welding approaches: detailed approach [1-4] and simplified one [5-8]. A detailed approach is a straightforward transient analysis of moving heating source. It computes the heat transfer during welding then uses transient temperature history as the mechanical analysis to obtain the stress-strain solution. Temperature-dependent nonlinear material properties, filler metal deposition, solidification shrinkage, phase transform, plastic strain history and other physical phenomena are considered and formulated into the analysis. The major obstacle in the detailed analysis is the enormous requirements of computing time and processing memory. The CPU requirements practically prohibit the use of the thermal, elastoplastic and moving heating source model for the prediction of the distortion and residual stress on large structures. The simplified approach has been devised to yield fast prediction of distortion as well as residual stress. Instead of conducting lengthy transient analysis, static analysis is executed within this approach. The natural of the simplified approach is to simulate the welding deformation through applying equivalent mechanical load, equivalent mechanical temperature load, or equivalent plastic strain.

This paper describes the application of a simplified FEA method, an inherent shrinkage approach, for welding distortion prediction. The analysis platform of the inherent shrinkage method is available in many commercial FEA packages. The material model consists of Young's modulus, Poisson's ratio, yield strength, elastic-plastic stress-strain relationship, and thermal expansion coefficient. All the material properties are temperature-dependent. The equivalent weld shrinkage is applied as temperature load. The elastic-plastic model enables the analysis to capture the plastic deformation around the weld where is the primary cause for distortion. The accumulation and interaction of the plastic zones of multiple welds enable the analysis to compute the contribution of individual to the global rigidity and therefore accurately compute the effect of welding sequence on distortion.

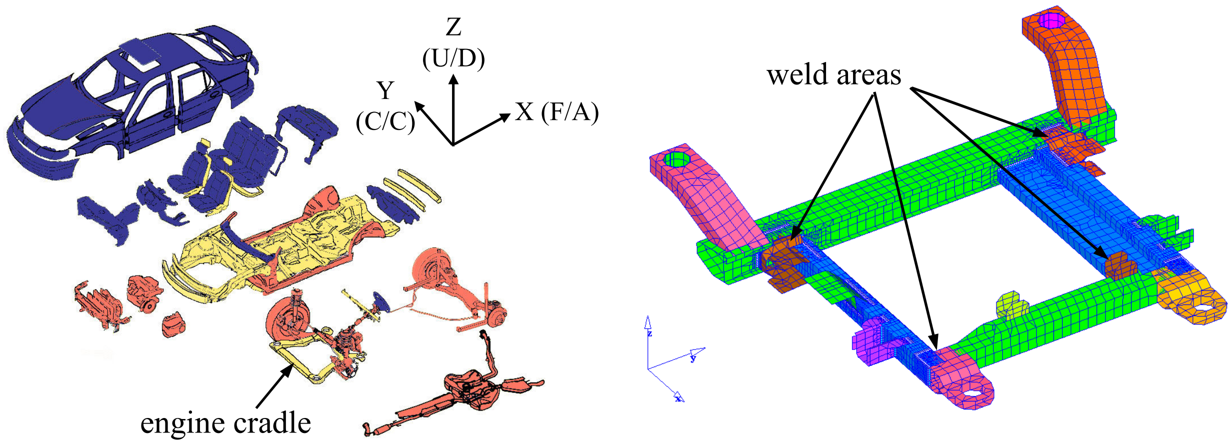
2. Modeling Procedure

A Computer Aided Design (CAD) file of the engine cradle was provided by an industrial partner. Hypermesh [9] was used as pre and post processor to build a finite element model from CAD file and to display simulation results. Abaqus [10] was utilized as a solver to compute the welding distortion. The whole cradle geometry is represented by a shell model, i.e., four-node and three-node shell elements in the analysis. The finite element model contains of 8017 nodes and 9037 elements.

In this study, there are some assumptions that are commonly used in FEA, such as small deformation assumption, homogenous and isotropic material property assumption. It is assumed that a weld is laid on the part simultaneously from its start to end, ignoring the continuous deposition effect of the actual weld. In other words, the travel speed of the weld arc is assumed to be infinitely high. The inherent shrinkage method is not designed to simulate the transient thermal-mechanical process during welding. It is not suitable to utilize it to study the transient phenomena such as thermal history and phase transformation, which can be studied using transient analysis approaches such as the moving source model.

Figure 1 illustrates the typical location of an engine cradle and its FEA model. The general orientation of a vehicle assembly is illustrated in Fig. 1(a), where (1) X-axis direction is in the fore-aft (F/A) direction, (2) Y-axis direction is in the cross-car (C/C) direction, and (3) Z-axis direction is in the up-down (U/D) direction of the vehicle. As shown in Fig. 1(b), the grid is graded from fine to coarse as moving away from the welds which are marked as red color elements. The typical element length in the welds is 5 to 7 mm and it is 20 to 25 mm in the areas away from the welds. Boundary conditions are applied to the FEA model according to the specification and experimental procedure. To simulate the locating and fixturing scheme during welding, the boundary conditions shown in Fig. 2 are imposed on the FEA model during the welding steps. After the welding steps, the boundary conditions representing the fixturing are removed to simulate the fixture release after welding. At the same time, a new set of boundary conditions is applied to the model, which is the same as the locating scheme used in the Coordinated Measuring Machine (CMM) measurement. The boundary conditions described above comply with the actual welding procedure.

The moving heat source model is employed to investigate the heat sink effect caused by ribs in extruded parts. It is an uncouple transient thermal-mechanical analysis. First, a transient heat transfer analysis is conducted to simulate the welding heat transfer, and then the computed thermal history is used as thermal load in the mechanical analysis to calculate distortion and residual stresses. Two weld sequences are simulated: sequence A and B.



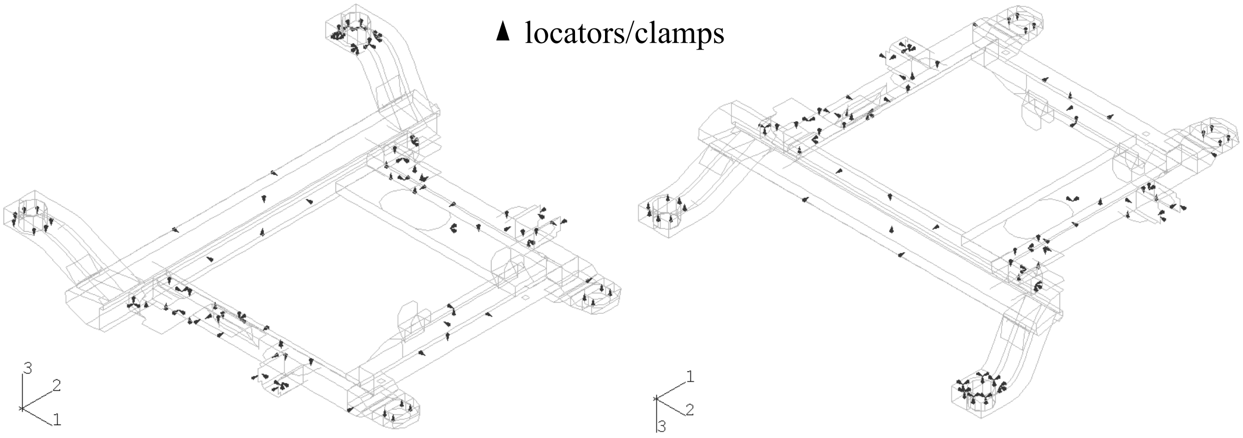
(a) Typical location of an engine cradle

(b) FEA model of engine cradle

Figure 1. A typical engine cradle and its FEA model

3. Simulation Results

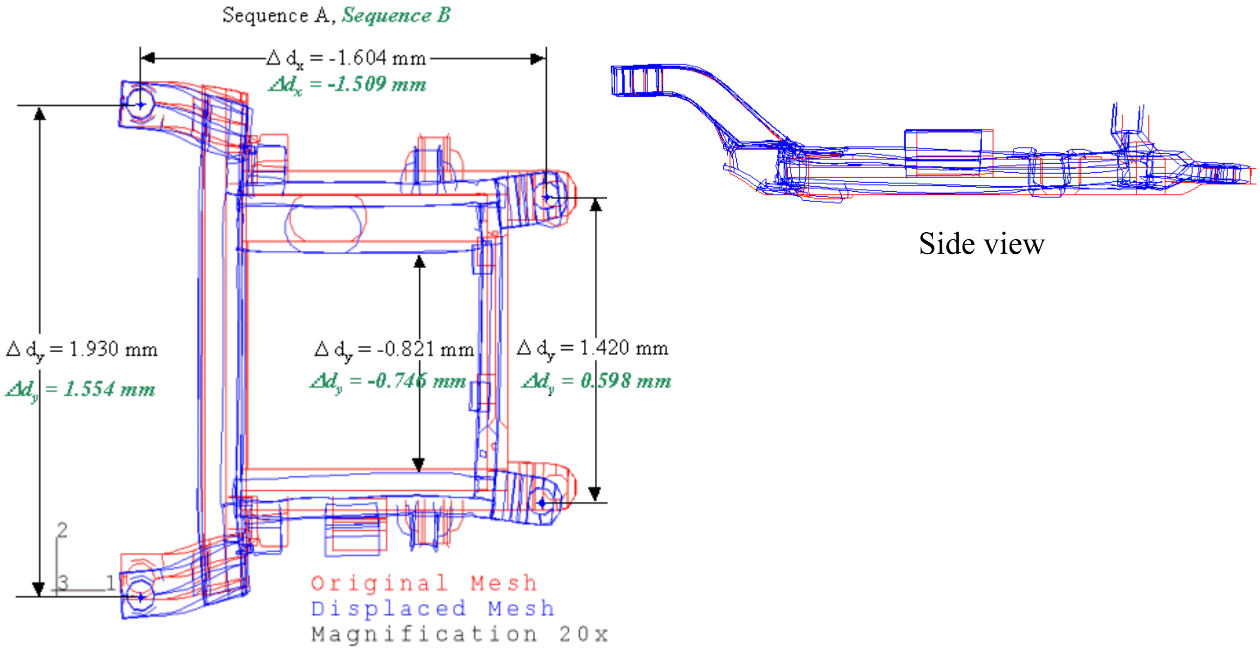
The distorted shape of cradle is illustrated in Fig. 3. As to the effect of welding sequence on distortion, welding sequence A causes larger deformation than sequence B does. The largest deformed magnitude is 1.93 mm which occurs in the y-axis direction. Figure 4 shows the residual stress, or Von-Mises stress, in the cradle after welding from sequence B. The maximum residual stress is 200 MPa resulted from welding process. Figures 5, 6 and 7 respectively display the comparisons of simulated results with CMM measured data in the X-axis (F/A), Y-axis (C/C), and Z-axis (U/D) directions. The deflection data is compared at 31 and 26 points in the left-hand and right-hand sides of the cradle respectively. The tolerance specifications at these points are also plotted in these figures.   Figure 5 indicates that respectively 10 and 5 measured points have F/A deflection greater than tolerance specifications in the left-hand and right-hand side. Figure 6 indicates that respectively 8 and 10 measured points have C/C deflection greater than tolerance specifications in the left-hand and right-hand side. Figure 7 shows that the U/D deflections meet the specification in the left-hand side of the cradle. However, 2 measured points have U/D deflection greater than tolerance specifications in the right-hand side of the cradle.



Top view

Bottom view

Figure 2. Locators and clamps scheme for welding



Side view

Top view

Figure 3. Deformed shape of welding sequences A and B

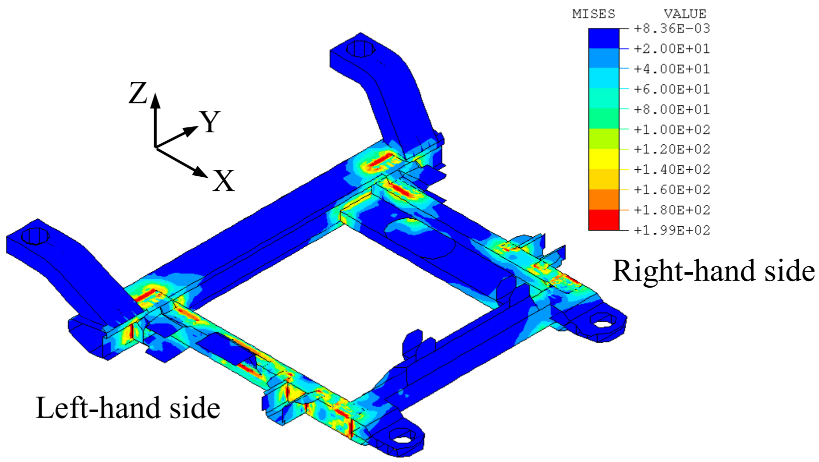
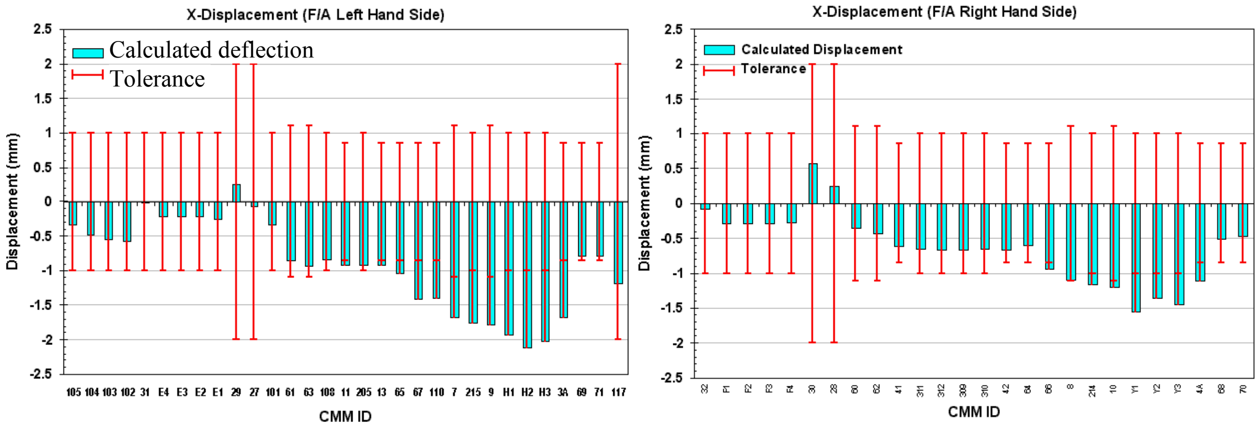


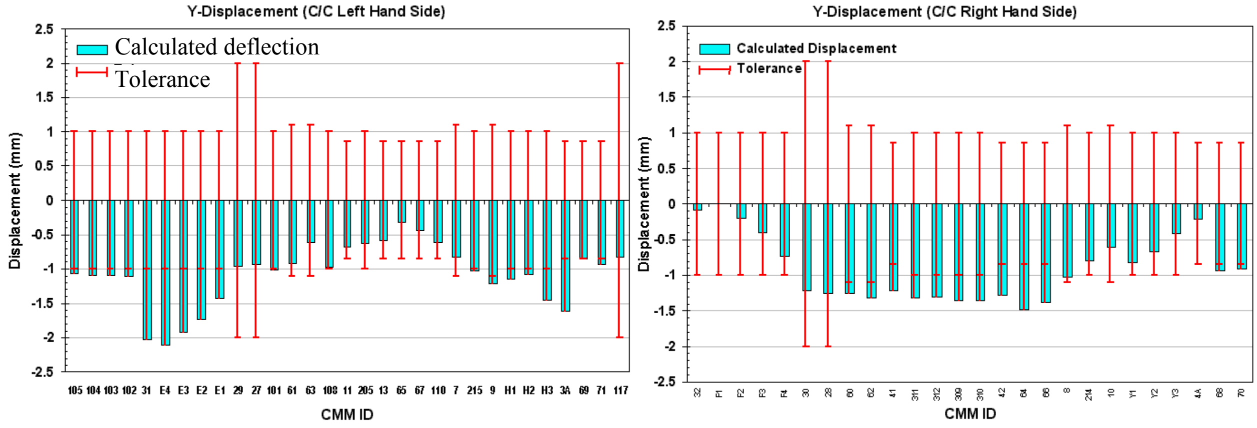
Figure 4. Contour of residual stress after welding from sequence B



Left-hand side

Right-hand side

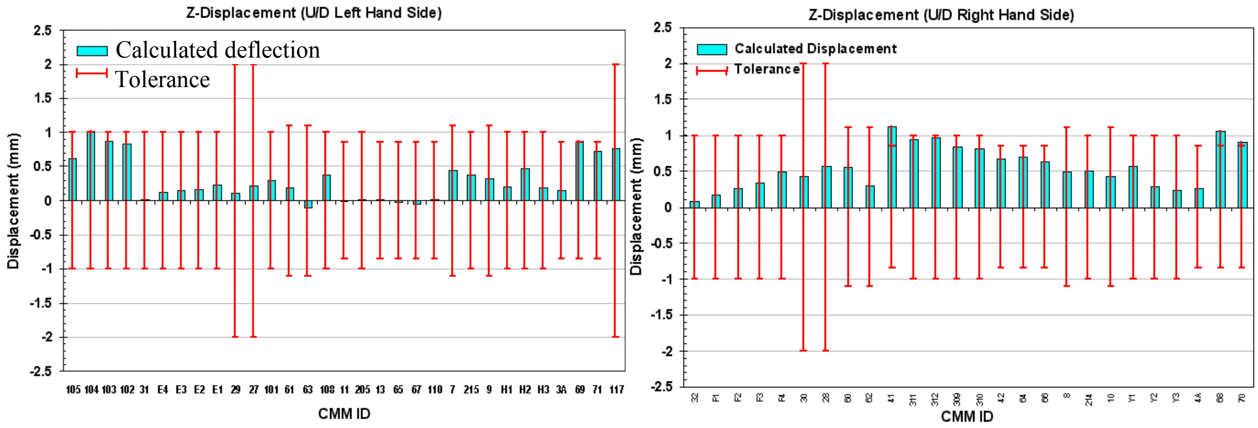
Figure 5. Comparison results of deflections in the X-axis (F/A) direction



Left-hand side

Right-hand side

Figure 6. Comparison results of deflections in the Y-axis (C/C) direction



Left-hand side

Right-hand side

Figure 7. Comparison results of deflections in the Z-axis (U/D) direction

4. Conclusions

The objective of this study is to provide FEA-based analysis on the MIG welding distortion of aluminum cradle. The technique developed in this project meets the industry's need. It can be used to mathematically prove various part designs and welding procedures without physical builds.

The welding distortion of the cradle can be predicted with reasonable accuracy. The distortions, especially those of key product characteristics are within the tolerances. In addition, the possible welding sequences resulting in minimum distortion can be determined from the simulation.

This project also serves as a case study for the graduate students to predict the quality of a real industrial part using CAE in manufacturing.

References

1. Masubuchi, K, "Analysis of welded structures – residual stresses, distortion and their consequences," Welding *International Series on Material Science and Technology*, Vol. 33, 1980, Pergamon Press, Oxford.
2. Shim, Y., Feng, Z.L., Lee, S., Kim, D.S., Jaeger, J., Paparitan, J.C., and Tsai, C.L., "Determination of residual stress in thick-section weldments," *Welding Journal*, Vol. 71, 1992, pp. 305-312.
3. Feng, Z.L. and Michaleris, P., "Evaluation of 2D and 3D FEA models for predicting residual stress and distortion," *ASME PVP Proceedings*, Vol. 347, 1997.
4. Josefson, B.L. and Karlsson, C.T., "FE-calculated stresses ina multi-pass butt-welded pipe – a simplified approach," *Int. J. of Pressure Vessels and Piping*, Vol. 38/3, 1989, pp. 227-243.
5. Dike, J., Cadden, C., Corderman, R., Schultz, C., and McAninch, M., "Finite element modeling of multipass GMA welds in steel plates," 4th Int. Conf. of Trends in Welding Research, Gatlinburg, TN, 1995.

6. Tsai, C.L., Cheng, W.T. and Lee, H.T., "Modeling strategy for control welding-induced distortion," AIME, *J. of Minerals*, 1995, pp. 335-345.
7. Tsai, C.L., Park, S.C., and Cheng, W.T., "Welding distortion of a thin-plate panel structure," Welding Journal, Vol. 78/5, 1999, pp. 156-165.
8. Mourgue, P., Gooroochurn, Y., Bergheau, J.M., Boitout, F., Porzner, H. and Niedenzu, P., "Modeling weld sequence on industrial parts," SYSWELD research paper, 2000.
9. HyperMesh version 5.0 manuals, 2001, Altair Engineering, Inc., Troy, MI.
10. Abaqus/Standard User's Manual, version 6.4, 2004, Hibbitt, Karlsson & Sorensen, Inc., Pawtucket, RI.

Y. GENE LIAO
Dr. Liao received the B.S.M.E. from National Central University, Taiwan, M.S.M.E. from the University of Texas, Mechanical Engineer from Columbia University, and the Doctor of Engineering degree from the University of Michigan, Ann Arbor. He is currently an assistant professor at the Wayne State University. He has over 15 years of industrial practices in the automotive sector prior to becoming a faculty member.