Prediction of Welding Deformation of Automotive Components Using Large-scale Thermal Elastic Plastic Analysis

by IKUSHIMA Kazuki**, MAEDA Shintaro***, UCHIMURA Taro***, KAWAHARA Atsushi**, KUWABARA Hitoshi****, KANETAKE Hiroaki***** and SHIBAHARA Masakazu**

Welding is often used to join automotive components during production. Deformation may occur during welding, which leads to increased production costs. Therefore, prediction of welding deformation is highly demanded in automobile production. To predict the welding deformation of real automotive components, it is necessary to conduct a numerical simulation within a reasonable computing time. In this study, an efficient simulation was achieved with a large-scale thermal elastic plastic analysis method, called the idealized explicit finite element method (IEFEM), to predict deformation of a side rail, which is a component of the rear suspension member of a car. In addition, the analysis method was also applied to the prediction of deformation in a front suspension member having more than 2 million degrees of freedom. The accuracy of the analyses was evaluated by comparison with experimental measurements. The results indicated that the IEFEM can accurately predict welding deformation and can be used to conduct simulations within a reasonable computing time.

Key Words: Finite element method, Thermal elastic plastic analysis, Idealized Explicit FEM, Multigrid method, Welding deformation

1. Introduction

Currently, automatic welding technology is used to increase production efficiency and reduce labor costs in automobile manufacturing. In addition, to decrease the weight of a vehicle in response to global environmental problems, thinner plates are used by adopting high-strength material. However, a large degree of deformation can occur when thin plates are welded, and this may cause problems. To increase the use of automatic welding, it is necessary to predict welding deformation in advance of production and incorporate this into the design.

Welding deformations can be predicted by using thermal elastic plastic (TEP) analysis, which is based on the nonlinear finite element method (FEM). In TEP analysis, the mechanical process during welding, in which a material is melted by heat input from a welding torch, is consecutively analyzed. Thus, large amounts of computing time and memory are necessary for TEP analysis. Currently, the progress of analysis technology can realize a more practical computation. However, the scope of conventional TEP analysis is still limited. Especially for automotive components, the prediction of welding deformation using TEP analysis is not widely utilized since they have complex shapes and a huge computing time is necessary.

In this study, the Idealized Explicit FEM (IEFEM) is employed to predict welding deformation in large-scale complex automotive structures within a reasonable computing time. To efficiently analyze the welding deformation in a real complex structure, the concept of multigrid method is introduced to the IEFEM. The predicted welding deformations are compared with measured results to show the accuracy of the proposed method. Through these discussions, the applicability of the proposed method to the prediction of welding deformations in a real complex structure is investigated.

2. Large-scale TEP analysis method

In this study, to achieve an efficient analysis of large-scale complex structures, the IEFEM with the algebraic multigrid method (AMG) is employed. The concept of the proposed method (MGIEFEM) is shown in Fig. 1. In the MGIEFEM, first, displacement vector $U_1$ is calculated by the Idealized Explicit method for several time steps based on the following equation:

$$M_1^* \ddot{U}_1 + C_1^* \dot{U}_1 + K_1 U_1 = F_1$$

where $M_1^*, C_1^*, K_1$ and $F_1$ are the modified mass matrix, the modified damping matrix, the stiffness matrix and the given load vector on the 1st level grid, respectively. The mass and damping matrices are modified to obtain the static equilibrium state fast. The details of these matrices are shown in the literature. To calculate the displacement $U_1$, the central difference is applied to Eq. (1) for discretization of time. After that, the residual force vector $R_1$ is calculated on the finest grid (1st-level grid) by Eq. (2):

$$R_1 = F_1 - K_1 U_1$$

The calculated residual force vector $R_1$ is restricted to the coarse grid (2nd-level grid) as load vector $F_2$.
where $P_2$ is the interpolation matrix. The details of the interpolation matrix are presented in the literature\(^9\), and the pseudo-Laplacian method is used in the present study. By using the load vector obtained by Eq. (3), the displacement on the 2nd-level grid is calculated based on the Idealized Explicit method. These procedures are recursively applied until the procedure reaches the coarsest grid ($n$-th-level grid). On the coarsest grid, the displacement is calculated by the direct method using the lower-upper (LU) decomposition performed in advance. After calculation of the displacement on the coarsest grid, the displacement is prolonged to the finer grid (($n$-1)-th-level grid), and the displacement on the finer grid is updated:

$$U_{n-1} \leftarrow U_{n-1} + P_{n} U_n$$ \hspace{1cm} (4)

The displacement on the finer grid is calculated again using the Idealized Explicit method. After calculation of the displacement, the displacement is prolonged to the finer, ($n$-2)-th-level, grid and the displacement is calculated in the same manner as for the ($n$-1)-th-level grid. These procedures are applied recursively until the calculation reaches the finest grid (1st-level grid), and then the iterations for the multigrid method are completed.

Analysis flow of this method is as shown in Fig. 2. First, as an analysis preprocess, multiple grids with different spatial resolution are generated according to the algebraic characteristics of a stiffness matrix according to the smoothed aggregation method\(^9\) and LU decomposition is performed and stored on the coarsest grid. After that, the temperature field is renewed and the stiffness matrix for the whole structure is calculated. Here, the stiffness matrix is calculated only on the finest grid. This is to improve the followability to the plastic deformation by renewing the stiffness matrix on the finest grid, that is, the inputted mesh, because the nonlinear deformation due to temperature change with the progress of welding is a local phenomenon and limited to the vicinity of the weld torch. The stiffness matrix for each grid is stored in the form of a sparse matrix to reduce memory consumption. Then, the displacement for a load step is determined by iteratively calculating the displacement until the whole system reaches a state of static equilibrium according to the multigrid procedure as shown in Fig. 1. The calculation is iterated for each load step from the beginning of heating to the end of cooling. The detailed procedures are discussed in the literature\(^10\).

Using this analysis procedure, the proposed method can be expected to have better convergence for complex structures than the normal IEFEM.

3. Analyses of real complex structures

3.1 Side rail model

To demonstrate the applicability of the proposed method to a real structure, it was applied to the analysis of welding deformation in the side rail model shown in Fig. 3. The side rail is part of the rear suspension member in an automobile. The thickness of the side rail plates ranged from 1.8 to 4.5 mm. The element size in the welding direction, the perpendicular direction to the welding direction and the thickness direction near welded part are 1.5 mm, 1.0 mm and 0.75 mm, respectively. After mesh division, the number of nodes, elements and degrees of freedom (DOFs) were 161,071, 119,816 and 483,213, respectively. The number of welding lines was 15. The moving rectangular heat source is employed in this analysis. In this heat source, uniform body heat is...
given to the welded part. The welding current was 169.22 to 220.06 A, the welding voltage was 17.32 to 24.18 V, and the welding speed was 16.67 mm/s. The locations of the welding lines are shown in Fig. 4. The constraint conditions are also shown in Fig. 4. The constraint condition named group 1 in Fig. 4 is released 8 s after the start of welding. Constraint condition group 2 is released after 17 s. After 26 s, all remaining constraint conditions were released and rigid body mode is constrained. These constraint conditions are based on those used in the actual fabrication process such as clamps and pins. The experiment was conducted using the actual work process. The material properties used for the base metal and the weld metal in this analysis are shown in Fig. 5 which were taken by measurement. The computer employed for this analysis had the following specifications: the CPU was an Intel Core i7 3.2-GHz processor and the GPU was an NVIDIA GeForce GTX 980 processor. We also conducted an analysis using the iterative substructure method (ISM) under the same condition. The ISM is well-validated method and the analysis results are compared with those by the ISM.

Figure 6 shows the distribution of equivalent stress after cooling was complete. It can be seen that a large residual stress occurs near the welded joints. Figure 7 shows the distribution of displacement in the z-direction after welding. From the figure, it can be seen that the calculated displacement is up to about 1.5 mm. Figure 8 compares the displacement in the z-direction along line A1 to A6 among the proposed method, the ISM and the experimental measurement. The displacement of the experiment was measured by the coordinate measuring machine.

The accuracy of the analysis is investigated according to the concordance rate defined by Eq. (5):

\[
c = 100\% \times \left(1 - \frac{\sum_{i=1}^{n} |\Delta u_i|}{\sum_{i=1}^{n} |u_i|}\right)
\]

where \(c\) is the concordance rate in percent, \(n\) is the number of measurements, \(\Delta u_i\) is the difference between the analysis and the measurement, and \(u_i\) is the measured data. The concordance rate of the proposed method and the experiment along line A1-A6 was
85%. That of the proposed method and the ISM was 75%. So, in this analysis, it can be said that the result obtained by the proposed method agree very well with both the experimental measurement and that by the ISM.

3.2 Front suspension member model

To demonstrate the applicability of the proposed method for even more complex and large problems, it was applied to the analysis of welding deformation in a front suspension member model. The analysis model is shown in Fig. 9. The plate thickness ranged from 2.0 mm to 7.0 mm. The element sizes near welded part are as follows: 1.5 mm in the welding direction, 1.0 mm in the perpendicular direction to the welding direction and 0.75 mm in the thickness direction. After mesh division, the number of nodes, elements and DOFs in the model were 700,687 and 521,623, 2,102,061, respectively. The same heat source as that in the previous section is employed. The welding conditions were current of 170 A or 180 A, voltage of 21 V or 22 V, and speed of 13.33 mm/s. The number of welding lines was 40, and their locations are shown in Fig. 10. The welding sequence was separated into four stages. After the 1st welding stage, the part was cooled for 1,800 s. After that, welding proceeded in the following sequence: the 2nd welding stage was performed and the part cooled for 40 s, the 3rd welding stage was performed and the part cooled for 40 s, and the 4th welding stage was performed and the part cooled for 15 s, after which the part was completely cooled. The constraint conditions are also shown in Fig. 9. All the constraints were considered in the 1st welding stage and cooling. In the 2nd, 3rd, and 4th welding stages, constraint condition named group 1 in Fig. 9 was released. The remaining constraints were released and the rigid body motion was constrained until the final cooling. As same as the previous section, the experiment was conducted under the same work process as the actual fabrication, and the constraint conditions are based on the actual fabrication. The material properties used for both the base metal and the weld metal in this analysis are shown in Fig. 11 which were taken by measurement. The same computer was employed as in the previous section.

Figure 12 shows the distribution of equivalent stress after all welding was complete. The figure indicates that large stresses occur near the welding lines. Figure 13 shows the distribution of displacement in the z-direction after welding, as viewed from the negative z-direction. Figure 14 shows the change of distance from the reference plane along line A-B in Fig. 13 before and after welding. The reference plane is defined by points C, D, and E in

---

![Fig. 9](image-url) Analysis model and constraint conditions for front suspension member.

![Fig. 10](image-url) Position of welding lines in front suspension model.

![Fig. 11](image-url) Material properties for front suspension model.

![Fig. 12](image-url) Equivalent stress distribution in front suspension model after welding.
The displacement of the experiment was measured by the coordinate measuring machine\(^{11}\). From Fig. 14, it is found that the deformation obtained by the proposed method well reproduced the tendency of the measured deformation. However, a certain difference can be seen between the analysis and measurement in the range from 6000mm to 9000mm. The concordance rate, defined by Eq. (5), of the proposed method and the experiment along line A-B is 60%. It can also be seen that the lower plate bent along line A-B and raised the center. There are two main reasons for this deformation: the welding line between the lower and upper plates is relatively long compared with the other welding lines and longitudinal shrinkage of the welding line occurs below the neutral surface. These characteristics of the weld can generate a bending deformation in the whole structure.

The total number of load steps in this analysis was 11,340, and the computation took about 38 h. From these results, it can be concluded that the proposed method can analyze the welding deformation in real structures in a reasonable time.

4. Conclusions

In this study, the IEFEM was applied to the analysis of welding deformation in automotive components to show its applicability to complex structures. To achieve an efficient analysis, the multigrid method was employed with the IEFEM.

The proposed method was applied to the analysis of a side rail and front suspension member, which are automotive components. The analyses considered constraint conditions encountered in actual production. The analysis results were compared with the experimental measurements. The concordance rate of the analyzed result and the measurement in the side rail model and front suspension member model were 85% and 60%, respectively.

The number of DOFs in the side rail model and the front suspension member model were 483,213 and 2,102,061, respectively. Using a single PC, the computing times to analyze each model were about 4 h and 38 h, respectively. These results indicate that the proposed method can analyze welding deformation in real structures in a reasonable computing time.

**Reference**

1) Y. Ueda, T. Yamakawa: Analysis of thermal elastic-plastic behavior of metals during welding by finite element method, J. JWS, 42-6 (1973), 567-577.
2) T. Muraki, J. J. Bryan and K. Masubuchi: Analysis of Thermal Stresses and Metal Movement During Welding (Part 1, Analytical Study), J. Eng. Mater. Technol., 97-1 (1975), 81-84.
3) E. Friedman: Thermomechanical Analysis of the Welding Process Using the Finite Element Method, J. Pressure Vessel Technol., 97-3 (1975), 206-212.
4) H. Nishikawa, H. Serizawa, H. Murakawa: Actual application of large-scaled FEM for analysis of mechanical problems in welding, Quarter. J. JWS, 24-2 (2006), 168-173.
5) J. Goldak, M. Mocanita, V. Aldea, J. Zhou, D. Downey, D. Dorling: Predicting Burn-through When Welding on Pressurized Natural Gas Pipelines, Proceedings of 2000 ASME Pressure Vessels and Piping Conference (2000), 23-27.
6) K. Ikushima, M. Shibahara: Prediction of residual stresses in multi-pass welded joint using Idealized Explicit FEM accelerated by a GPU, Comput. Mater. Sci., 93 (2014), 62-67.
7) K. Stüben: Algebraic Multigrid (AMG): An Introduction with Applications, GMD-Report, 70 (1999), 1-127.
8) M. Brezina, R. Falgout, S. MacLachlan, T. Manteuffel, S. McCormick and J. Ruge: Adaptive algebraic multigrid, SIAM J. Sci. Comput., 27-4 (2006), 1261-1286.
9) P. Vaněk, J. Mandel and M. Brezina: Algebraic Multigrid by Smoothed Aggregation for Second and Fourth Order Elliptic Problems, Computing, 56 (1996), 179-196.
10) K. Ikushima, M. Shibahara: Nonlinear Computational Welding Mechanics for Large Structures, J. Offshore Mech. Arct. Eng., 141-2 (2019), 021603.
11) FARO Quantum ScanArm, https://www.faro.com/ja-jp/products/3d-manufacturing/faro-scanarm/, Accessed at May 1, 2020.