Investigation of factors influencing welding deformation of ship block by inherent strain analysis using idealized explicit FEM

Masashi Hata\textsuperscript{1}, Shinsuke Itoh\textsuperscript{1}, Taketoshi Sugihiro\textsuperscript{3}, Yasuo Yamashita\textsuperscript{3}, Atsushi Kamio\textsuperscript{3}, Masakazu Shibahara\textsuperscript{2}, and Masahito Mochizuki\textsuperscript{1}

\textsuperscript{1}Graduate School of Engineering, Osaka University, 2-1 Yamada-oka, Suita, Osaka 565-0871, Japan
\textsuperscript{2}Graduate School of Engineering, Osaka Prefecture University, 1-1 Gakuen-cho, Naka-ku, Sakai, Osaka, Japan
\textsuperscript{3}Sumitomo Heavy Industries Marine & Engineering Co., Ltd., 19 Natsushima-cho, Yokosuka, Kanagawa, Japan

Abstract. It is impossible to avoid welding deformations when steel structures such as ships are assembled. Therefore, quantitative prediction and effective control of residual welding deformation are necessary. However, the current finite element analysis requires very long computational time and memory. To solve this problem, we have developed an inherent strain analysis method using idealized explicit FEM and have applied this proposed method to a ship block model. The simulated results agree well with the measured deformation. Furthermore, the influence of various factors on the welding deformation of targeted ship block is also investigated.

1. Introduction
In ship construction, welding is used to join and assemble components. Welding deformation often causes a deviation from the design. Therefore, corrective work is required. However, since this corrective work relies on the skills of the engineer, it is difficult to estimate the time and the cost. If it was possible to predict the welding deformation accurately, the time and cost of the corrective work could be reduced. Furthermore, if the mechanism of welding deformation is clarified, more effective methods to avoid welding deformation can be developed. Therefore, if welding deformation and corrective work could be accurately predicted, a reduction of total energy is possible in ship construction. However, current methods of predicting welding deformation, such as finite element analysis (FEM), require a long computational time and large memory. Therefore, we developed an inherent strain analysis method using an Idealized Explicit FEM. In this method, the Iterative Substructure Method (ISM) \cite{1} is first used to create a database of inherent strain distributions. Then, using this database, an inherent strain analysis using Idealized Explicit FEM is applied to predict the welding deformation.
2. Analysis method of this study
Generally, inherent strain analysis is used to predict the welding deformation of large welded structures. The computational time for inherent strain analysis is shorter than that for thermal elastic analysis because inherent strain analysis is an elastic analysis. However, to use this method, a database of inherent strain distributions for each joint is necessary. Since the inherent strain distributions must be obtained through a series of experiments, the total time to obtain the results becomes enormous. Therefore, in this study, two improved methods are used. Figure 1 shows the flow of the proposed methods. ISM, which is a high-speed thermal elastic plastic FE analysis, is first used to create the database of inherent strain distributions. Then, by using the database, an inherent strain analysis using Idealized Explicit FEM is applied to predict the welding deformation. The following describes the concepts of the ISM and Idealized Explicit FEM.

2.1. Iterative Substructure Method for thermal elastic plastic analysis [1]
In simulations of welding, two characteristics of welding greatly increase the computing time. One is the strong nonlinearity near the welding torch. However, the area that is nonlinear is very limited and small in comparison with the entire structure. Typically, finite element (FE) analysis must be applied as a total nonlinear solution even when the nonlinear region is a very small part of the whole model. The purpose of ISM is to solve these problems. In ISM, the linear and the nonlinear regions are separated. As shown in Figure 2, these regions are called A and B, respectively. The boundary between regions A and B is designated Γ. Most of the computing time in FE analysis is used to solve large-scale simultaneous equations. In welding problems, the equations must be solved \((Ns \times Ni)\) times, where \(Ns\) is the time step and \(Ni\) is the iteration step. As an example, when using Gauss elimination, the computing time for forward elimination and back substitution is proportional to \((DOF)^3\) and \((DOF)^2\), respectively, where DOF is the number of degrees of freedom. If the stiffness matrix is constant, the matrix for forward elimination can be reused. In this case, the computing time becomes almost the same as that for performing the forward elimination one time because the time required by back substitution is markedly shorter than that for forward elimination. The differences between the conventional substructure method and ISM are summarized in Figure 3. In the conventional substructure method, the whole model is divided into regions A and B. In this case, region A changes as region B moves. Thus, the stiffness matrix for region A changes. In contrast, ISM uses the whole region \((A+B)\) and the nonlinear region B. If region B moves, the whole region \((A+B)\) does not change. The stiffness matrix for the whole region \((A+B)\) after forward elimination is saved. In this way, the process can be repeated to solve the displacement in region A without changing the stiffness matrix.

![Figure 1. Flow of proposed method.](image-url)
After obtaining the displacement of region A, the displacement in region B is computed by using the displacement of the boundary $\Gamma$ between regions A and B. Thus, the computing time is reduced by omitting the forward elimination of the stiffness matrix for the whole region (A+B).

2.2. Inherent strain analysis using Idealized Explicit FEM [2]
To simulate the welding deformation, thermal elastic plastic analysis or inherent strain analysis is used. Thermal elastic plastic analysis can simulate transient welding mechanical phenomena. However, if thermal elastic plastic analysis is carried out using implicit static FEM, the computing time and the memory utilization increase in proportion to the square of the number of elements [3]. In contrast, the computing time for inherent strain analysis is much smaller than that for thermal elastic plastic analysis, because inherent strain analysis computes only one step of an elastic analysis by using the inherent strain database. However, the tendency for the computing time and memory to increase is the same as that for thermal elastic plastic analysis if the computation method is implicit static FEM. In this research, Idealized Explicit FEM is used [2]. In this method, a step is divided into hundreds of time steps and the displacements are computed for each. After computing the displacement, the static equilibrium of the whole system is checked. The displacements are computed until a static equilibrium state is obtained. The concept of Idealized Explicit FEM is shown in Figure 4. Generally, in implicit dynamic FEM, the Wilson theta method or the Newmark beta method is used to solve the dynamic problem. To obtain the displacement vector $\{u\}$, very large simultaneous equations have to be solved. In contrast, explicit FEM does not require solving very large matrix equations. The principal theory of explicit FEM can be described as follows [4]. First, the nodal velocity vector $\{\dot{u}\}$ and the nodal acceleration vector $\{\ddot{u}\}$ are discretized by the central difference method:

$$\{\ddot{u}\}_i = \left(\{u\}_{i+\Delta t} - 2\{u\}_i + \{u\}_{i-\Delta t}\right)/\Delta t^2,$$

$$\{\dot{u}\}_i = \left(\{u\}_{i+\Delta t} - \{u\}_{i-\Delta t}\right)/(2\Delta t),$$

where $t$ is the time, $\Delta t$ is the time increment, and $\{u\}_{i-\Delta t}$, $\{u\}_i$, and $\{u\}_{i+\Delta t}$ are the displacement vectors at times $t - \Delta t$, $t$, and $t + \Delta t$, respectively.

$$[M][\ddot{u}]_i + [C][\dot{u}]_i + \int [B]^T [\sigma] dV = \{F\}_i,$$
Equation (4) is derived by substituting equations (1) and (2) into equation (3).

If the mass matrix $[M]$ and the damping matrix $[C]$ are diagonal matrices, then the displacement vector $\{u\}_{i=M}$ can be easily obtained from the displacement vectors $\{u\}_i$ and $\{u\}_{i=M}$. In this formulation, it is well known that the global stiffness matrix is not required and no large simultaneous equations need to be solved. To calculate the displacement, equation (4) is solved. However, for long-duration phenomena such as welding, the computational steps become enormous because of the very short time increment. The short increment is due to the Courant condition [5]. The Courant condition requires the distance travelled by a stress wave in a time increment to be less than the minimum element size. Idealized Explicit FEM overcomes this limitation.

Figure 4. Concept of Idealized Explicit FEM [3].

3. Experiment and simulation for fillet welding of a T-joint

In the proposed method, thermal elastic plastic analysis by ISM is used to create the database of inherent strain distributions. Therefore, in order to verify the adequacy of thermal elastic plastic analysis by ISM and to obtain the database of inherent strain distribution, an experiment and a simulation are performed. In a ship structure, since many stiffeners are attached by fillet welding of a T-joint, in this study, fillet welding of a T-joint is considered. An illustration of the T-joint is shown in Figure 5. The model and welding conditions are shown in Figure 6 and Table 1, respectively. The components are mild steel. The angular distortion is measured at four points, shown in Figure 5. The angular distortion refers to the difference in angle between the base plate and the stiffener after welding. Figure 7 shows a comparison of the angular distortion for the experiment and the simulation. This figure indicates that there are some differences between the experimental and analysis results. These differences are because the experiment was performed simply to reproduce the actual site. However, taking this into account, the ISM simulation corresponds to the experiment.

Figure 5. Illustration of T-joint model. Figure 6. FE mesh divisions of T-joint model.
Table 1.  Welding conditions.

| Voltage (V) | Current (A) | Welding speed (mm/s) |
|------------|-------------|----------------------|
| 44.0       | 290.0       | 8.0                  |

Figure 7.  Comparison of angular distortion for experiment and FE analysis using ISM.

4. Effectiveness of the proposed method

In the proposed method, inherent strain analysis using Idealized Explicit FEM is used to predict the welding deformation of a large-scale structure. Therefore, inherent strain analysis using Idealized Explicit FEM and thermal elastic plastic analysis by ISM are compared in order to verify the effectiveness of the former method. This analysis uses the inherent strain distributions from both the welding and straightening by line heating calculated by ISM. In this study, straightening by line heating is performed after welding. The model, welding conditions and heating conditions are shown in Figure 8, Table 1 and Table 2, respectively. The position of line heating and the contour of the Mises plastic strain are shown in Figure 9 and Figure 10, respectively. From Figure 10, since the Mises plastic strain can be regarded as constant, the inherent strain is assumed to be constant for all welding lines in this study. Figure 11 shows a comparison of the deformation of the center section weld line for ISM and inherent strain analysis using Idealized Explicit FEM. As shown in this figure, the simulation using inherent strain analysis using Idealized Explicit FEM corresponds to that using thermal elastic plastic analysis by ISM with high accuracy.

Table 2. Line heating conditions.

| Heat input (J/mm) | Heating speed (mm/s) |
|-------------------|----------------------|
| 2160              | 6.0                  |

Figure 8.  FE mesh division.
5. Inherent strain analysis using Idealized Explicit FEM applied to actual ship block

A ship block is assembled using a large amount of fillet welds. The proposed methods are next applied to the actual ship block shown in the photo in Figure 12. The block is near the engine and the problem of welding and line heating deformation occurs at the aperture. Straightening by line heating is performed after welding. The welding and heating conditions are shown in Table 1 and Table 2, respectively. The edges and the crossed areas are welded manually. The analysis model is shown Figure 13. The total number of nodes and solid elements is 875698 and 614110, respectively. The total number of welding lines in this model is almost 600. Figure 14 and Figure 15 show the contour of X-direction displacement and Y-direction displacement, respectively.

To investigate the influence of the heat input (Q) and the welding and heating velocity (v) on the deformation of the block, the values of each are changed in the simulation. Figure 16 shows the measurement and analysis results for the outline of the ship shape with in-plane deformation. From this figure, it is found that the simulated results correspond to the measurement results. Furthermore, it is found that the deformation at the aperture is strongly affected by the heat input.
**Figure 12.** Photo of actual ship block.  

**Figure 13.** Ship block model.  

**Figure 14.** Contour of X-direction displacement.  

**Figure 15.** Contour of Y-direction displacement.  

**Figure 16.** Outline of ship shape with 100 times in-plane deformation.
6. Conclusions
To predict the welding deformation of an actual ship block, an inherent strain analysis method using ISM and Idealized Explicit FEM was developed and applied. The results of the analysis are as follows.
(1) For an actual ship block, the simulated deformation agrees with the experimental results.
(2) The influence of the heat input on the deformation at the aperture is significant.

Acknowledgements
This study was supported in part by Priority Assistance for the Formation of Worldwide Renowned Centers of Research – The Global COE Program (project, Center of Excellence for Advanced Structural and Functional Materials Design) from the Ministry of Education, Culture, Sports, Science and Technology (MEXT), Japan.

References
[1] Murakawa H, Oda I, Ito S, Serizawa H, Shibahara M, Nishikawa H 2005 Iterative Substructure Method for Fast Computation of Thermal Elastic Plastic Welding Problems (Journal of the Kansai Society of Naval Architects) Japan 243 67-70 (in Japanese)
[2] Shibahara M, Ikusima K, Itoh S, Masaoka K 2011 Computational Method for Transient Welding Deformation and Stress for Large Scale Structure Based on Dynamic Explicit FEM (Journal of the Japan Welding Society) 29 (1) 1-9.
[3] Nishikawa H, Oda I, Shibahara M, Serizawa H, Murakawa H 2004 Three-dimensional Thermal-elastic-plastic FEM Analysis for Predicting Residual Stress and Deformation Under Multi-pass Welding (Proc. Of ISOPE’04) Toulon France CD-ROM
[4] Yagawa M 2004 KOUZOUKOGAKU hand book (MARUZEN) 302
[5] Sugimoto S, Yasuzawa K, Kagawa K 2001 Elasto-Plastic Collapse Behavior of Plate and Shell Structure Components by Dynamic Explicit FEA (The Computational Mechanics Conference, The Japan Society of Mechanical Engineers) 11 183-184