Development of High-speed and High-precision FEM for Analysis of Mechanical Problems in Welding

NISHIKAWA Hiroyasu*, ODA Isamu**, SERIZAWA Hisashi*** and MURAKAWA Hidekazu****

Abstract

Finite element method is a powerful tool for predicting welding distortion. However, the mechanical phenomena in welding are strong non-linear transient problems and thermal-elastic-plastic FE analysis requires very long computational times. In this paper, based on the calculation of large-scale three-dimensional welding distortion problems, the limitation of the applicability of commercial software is discussed. Next, in order to accelerate the thermal-elastic-plastic FE program, the solver was changed into a sparse matrix iterative solver from a skyline solver. Further, an interactive substructure method is proposed as a method to reduce the computing time in three-dimensional welding distortion problems and its effectiveness is demonstrated. The effect which combined the interactive substructure method and the sparse matrix iterative solver was investigated.

KEY WORDS: (Finite Element Method), (Three-dimensional Analysis), (Large-scaled Problem), (Welding Distortion), (Computing Time)

1. Introduction

In recent years, numerical simulation of various phenomena in the construction processes such as welding becoming popular. The purpose of such simulation is to compensate for the shortage of skilled workers, to minimize human error, eliminate inefficient trial and error process, and to establish construction technology on a scientific basis. In the manufacturing industries, the simulation of press forming and casting has reached the level of practical use, but the simulation of welding has still not been used widely. The reason for this is that it requires huge computing time since welding phenomena are strong non-linear and transient problems.

Large number of researches have been done on various mechanical problem of welding, such as residual stresses and distortions. Finite element methods based on thermal-elastic-plastic theory have been widely employed for these researches. When the finite element was first applied, the computation is limited to only two dimensional problems and three dimensional analysis of distortions of actual structures was out of scope.

To overcome this limitation, a method based on the concept of inherent strain was to compute the welding distortion. Since the problem is treated as an elastic problem, the computational time can be greatly reduced even for large scale three dimensional structures. However to use this method, it is necessary to have a database of inherent strain in advance. For the construction of such database for new materials, it is necessary to conduct many experiments or nonlinear thermal-elastic-plastic FE analysis which require time and cost. Thus, this method can be effective for limited problems for which the database of inherent strain is available.

To estimate the welding distortion of large structures with complicated geometry, it is necessary to perform a three-dimensional welding simulation. When the structure to analyze becomes large, the computational time increases drastically and this limits the computation using the available commercial code even in the present computer environment.

Various attempts were made to accelerate the three-dimensional welding simulation. The adaptive meshing procedure is a method which embeds the solid refined mesh into the whole structure model and moves it with the welding torch. The solid refined mesh represents the zone near the welding heat source. The composite mesh is also effective to describe the detail of the zone near the welding heat source. The dynamic substructure is a method which divides a welding part

* Received on November 30, 2004
** Kawasaki Heavy Industries, Ltd.
*** Graduate Student
**** Associate Professor

Transactions of JWRI is published by Joining and Welding Research Institute, Osaka University, Ibaraki, Osaka 567-0047, Japan
from the whole structure and rezones it with movement of the welding heat source. However, it is still not possible to calculate the mechanical phenomena of welding within the practical computing time by these methods.

Therefore, the authors attempted to develop a welding simulation method which is fast enough to predict the distortion of large scale structures manufactured in industries with reasonably high-precision.

2. Analysis of welding distortion using commercial program

In welding phenomena, temperature changes from room temperature to the melting temperature of the material. When the temperature dependency of physical properties of the metal is taken into consideration, the local stiffness of the structure changes from place to place in the thermal-elastic-plastic analysis. In the region near the welding torch where the temperature is near the melting point, the rigidity is extremely small compared to the region away from the torch. Therefore, the mechanical simulation of welding becomes a very strong but localized nonlinear transient problem.

In general, huge computational time is necessary to solve such a strong nonlinear and transient problem. Even using the well established existing general-purpose commercial codes, unrealistically large computational time is necessary to complete the simulation when the model is larger than specimens in the laboratory.

Some of the commercial codes are specially developed for analysis of welding problems such as SYSWELD and Quick Welder. Since price of Quick Welder is low and it operates on a personal computer and it is easy to operate, and it is becoming popular in manufacturing engineering sections of Japanese companies. In order to evaluate the capability of the commercial codes for the simulation of the actual structures assembled in manufacturing engineering sections, welding distortion analysis is performed using Quick Welder, as an example.

The welding simulation of a disk composed of a disk formed by mechanical bending process and a machined flange part as shown in Fig.1 is carried out by Quick Welder. In order to obtain the distortion distribution along the circumferential direction, the model was analyzed as a three-dimensional problem. The disk was modeled using solid elements. The total number of nodes was 6,768.

The computed temperature distributions and deformation modes are shown in Fig.2. The computing time of this calculation is shown in Table 1. Judging from the fact that it takes 17.3 hours to complete the simulation using a personal computer Xeon-2.8GHz, it is difficult for an engineer in manufacturing engineering to conduct such simulations as part of everyday business. If the size of the problem is larger than this case, it can’t be handled due to the restriction of the program. Form the experience of this test computation, the authors reached the conclusion that it is necessary to develop a high-speed and high-precision finite element method to simulate large-scale problems in welding.

<table>
<thead>
<tr>
<th>Table 1: Computing time.</th>
</tr>
</thead>
<tbody>
<tr>
<td>CPU</td>
</tr>
<tr>
<td>Xeon-2.8GHz</td>
</tr>
</tbody>
</table>
3. Improvement of speed by sparse matrix iterative solver

The authors developed a thermal-elastic-plastic finite element method program (it is referred to as Original hereafter) and carried out the welding simulation. Since the “Original” employs a skyline matrix solver, it also takes large computational time. When the computing time of each routine in the “Original” is analyzed, it is found out that the computing time in the skyline solver (which solves the simultaneous equations of whole structure) occupies 92% of the whole computing time.

In order to improve the computational speed and the capability for carrying out large-scale welding programs, the solver is replaced with the “Incomplete Cholesky Conjugate Gradient” method (it is referred to as ICCG hereafter) which is a sparse matrix iterative solver. The modifications necessary are limited only to the subroutines constructing whole stiffness as a sparse matrix and that solving the sparse matrix.

In order to measure the computing time, bead welding of low carbon steel plate with a length of 200 mm, a width of 250 mm, and a thickness of 25 mm was used as an example. The assumed welding conditions were: effective heat input was 420 J/mm with the heat source moved at a fixed welding speed was 1.0 mm/s. The heat input is assumed to be uniformly distributed over the zone 2 mm in width, 1 mm in depth and 5 mm in length.

Element divisions and the boundary conditions used for the simulation are shown in Fig. 3. The temperature dependency of the material constants are shown in Fig. 4. The number of nodes of the model is 6,426 and the number of unknowns is 19,164. The computer used for the analysis is the personal computer, Xeon-2.8GHz.

The temperature distribution at 50 seconds after the start of welding is shown in Fig. 5 and the deformation and stress distribution are shown in Fig. 6. For this example, the same stress and displacements are obtained from both ICCG and “Original”.

The total computing time, the ratio of the matrix solution in computation and the size of memory necessary to execute the computation are shown in Table 2. The computing time of ICCG is 7.8 hours which is 11.2 times faster than the 87.6 hours with “Original” and the ratio of the time used by the matrix solver to the

---

![Fig.3 Finite element model.](image)

![Fig.4 Material and physical properties.](image)

![Fig.5 Temperature distribution.](image)

![Fig.6 Deformation and stress distribution.](image)

<table>
<thead>
<tr>
<th>Table 2</th>
<th>Comparison between Original and ICCG.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Program</td>
<td>Computing time</td>
</tr>
<tr>
<td>Original</td>
<td>87.6 Hr</td>
</tr>
<tr>
<td>ICCG</td>
<td>7.8 Hr</td>
</tr>
<tr>
<td>Original/ICCG</td>
<td>11.2</td>
</tr>
</tbody>
</table>
Development of High-speed and High-precision FEM for Analysis of Mechanical Problems in Welding

whole computing time is also reduced from 92 to 43%. However, 7.8 hours computational time is still not short enough from the practical point of view.

The memory required for ICCG is 58 MB which is 1/7.8 reduced from the 452 MB of the “Original”. This means that ICCG can calculate a 7.8 times large-scale problem computed to “Original” with the same memory.

From this result, the improvement of the speed using ICCG solver does not reach the practical computing speed, but it is quite effective for computing large-scale problems.

4. Improvement of Computational Speed by Iterative Substructure Method

In order to further accelerate the computation of a welding simulation, it is necessary to improve the algorithm of simulation in addition to adopting the sparse matrix iterative solver.

To improve the speed of computation, it is necessary to take advantage of the characteristics of the welding problems. The most distinguished characteristic is the fact that the problem is highly nonlinear, but that the area exhibiting nonlinearity is limited to a small area moving with the welding torch. Thus most of the area of the structure remains in the elastic state and linear solution procedures can be applied. The time consuming nonlinear computation is necessary only for the nonlinear region near the moving welding torch.

On the other hand, from the aspect of FEM analysis, even when only small part of the model behaves nonlinearly, it is still necessary to solve the whole system as a nonlinear problem. In addition to this, it is necessary to carry out solutions of large simultaneous equations with coefficient matrix changing with time $N_s \times N_t$ times. Here, $N_s$ is the number of time steps and $N_t$ is the number of iterations to get convergence. If the coefficient matrix is constant and only the load changes, the computing time for solving simultaneous equations $N_s \times N_t$ times can be shortened to the same order for solving the same equation once.

According to the above discussion, the structure under welding can be divided into a linear region A and a non-linear region B. However, since the region B moves with a welding torch, the stiffness matrix of the region A changes with the motion of the torch. In order to avoid the change of the stiffness matrix for the region A, the original non-linear problem of $(A+B)$ is divided into the linear problem of $(A+B')$ and the non-linear problem of $B$ as shown in Fig.7. The region $B'$ is the fictitious region which corresponds to the region B at a certain moment of the welding process in the past and the region $B'$ is updated every ten or twenty steps in the incremental solution procedure. In this way, the stiffness of $(A+B')$ does not change until updated.

When the boundary between the regions A and B is $\Gamma$, both the displacement and the traction on the boundary $\Gamma$ must satisfy the continuity. As readily noticed, these continuities are not satisfied between $(A+B')$ and B regions. To ensure the continuities, the following iterative procedure called “interactive substructure method” is employed.

The flow chart of the interactive substructure method is shown in Fig.8. The procedure consists of the following five steps.

(a) Solve $(A+B')$ and compute displacements on the boundary $\Gamma$. At the same time, save the stiffness matrix after forward elimination.
(b) Solve (B) by using the displacements on $\Gamma$ computed in step (a) as the displacement boundary condition. In this way, the continuity of the displacement on the

![Fig.7 Iterative substructure method.](image)

![Fig.8 Flow chart of iterative substructure method.](image)
boundary $\Gamma$ is maintained. This solution process is non-linear and Newton-Raphson iterative method is employed.

(c) Compute the unbalance between the reaction forces from \{(A+B')-(B')\} and (B).

(d) To recover the continuity of the traction, feedback the above unbalanced force to step (a) and compute the correction for the displacement on the boundary $\Gamma$.

(e) Repeat steps (a) through (d) until the convergence is reached.

According to the above algorithm, the “Original” program was modified to the interactive substructure method (it is referred to as ISM hereafter).

Noting that the nonlinearities in a welding problem come from the plasticity and the temperature dependent material properties, the nonlinear region B in the present ISM, is selected as the sum of the elements belong to three regions defined as follows.

(i) The elements in which the current temperature exceeds 300 °C.

(ii) The elements in which the current temperature exceeds 200 °C and under plastic loading.

(iii) The elements in which the temperature difference exceeds 50 °C and under plastic loading.

The same skyline solver as in the “Original” program is used and the stiffness matrix of the whole region (A+B') after forward elimination is saved. The matrix after forward elimination is updated every ten steps.

In order to examine the effectiveness of the combination of the interactive substructure method and the sparse matrix iterative solver, the ISM program in which the skyline solver is replaced by the ICCG is also developed (It is referred to as ISM+ICCG hereafter). Since the stiffness matrix after forward elimination of the region (A+B') cannot be saved in ICCG, the region (A+B') is solved over again at each time.

In order to measure computation speed, the same example as the preceding chapter was calculated by using the ISM program and the ISM+ICCG program.

The total computing time, the ratio of the matrix solution in computation and the required memory size for analyzing the same problem are compared among the four programs in Table 3. The computing time of ISM is 0.6 hours which is within the limit for practical application. It is accelerated 146 times from 87.6 hours of the “Original”. Similarly that of ISM+ICCG is 2.7 hours and is accelerated 32.4 times compared with the “Original”. But it is 4.5 times longer compared with ISM.

The memory required for ISM is 850 MB which is twice that of “Original” which requires 452 MB. This implies that, ISM can calculate only a half model compared with “Original” if the same computer is used. On the other hand, the memory used for ISM+ICCG is 48 MB. As far as the size of memory is concerned, ISM+ICCG can simulate 9.4 times larger models compared with “Original”, or 17.7 times larger models compared with ISM.

Although computational efficiency is improved about 100 times by the ISM, the required memory increases twofold. On the other hand, ISM+ICCG, which is effective in reducing the memory size, can reduce the memory size to 1/10. But the computational efficiency is improved only about 30 times.

As a future study, the algorithm of the iterative substructure method will be improved to take advantage of the sparse matrix iterative solver in both high-speed computation and in low-memory.

5. Conclusions

The welding simulation software which can calculate the mechanical phenomena of welding with high-speed and high-precision was developed to provide a tool for manufacturing engineers to solve their welding problems of practical interest. From this study, the following conclusions were obtained.

(1) Through the investigation of the applicability of commercial software for large scale welding problems, it is found that large computational time is required and this limits their application. Thus, the authors decided to develop a high-speed and high-precision finite element method to calculate large-scale problems in welding.

(2) In order to accelerate computation of the thermal-elastic-plastic problem by the finite element method, the solver was changed to a sparse matrix iterative solver from a skyline solver in the newly developed FE program. It turns out that computing speed is accelerated about 10 times and the amount of the memory used is reduced to 1/8.

(3) In order to achieve further improvement in the computing speed, the interactive substructure method was developed. This method takes full advantage of the characteristics of welding phenomena that the area which exhibits strong nonlinearity is very localized, but it moves. The test using a model with 19,164 degrees of freedom shows that the speed of computation becomes about 100 times better though it requires two times larger memory size.

(4) In order to reduce the memory size, a sparse matrix iterative solver is introduced in the interactive substructure method. Though the speed of computation becomes 30 times less than the above

<table>
<thead>
<tr>
<th>Program</th>
<th>Computing time</th>
<th>Solver time ratio</th>
<th>Memory size</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>87.6 Hr</td>
<td>92 %</td>
<td>452 MB</td>
</tr>
<tr>
<td>ICCG</td>
<td>7.8 Hr</td>
<td>43 %</td>
<td>58 MB</td>
</tr>
<tr>
<td>ISM</td>
<td>0.6 Hr</td>
<td>83 %</td>
<td>850 MB</td>
</tr>
<tr>
<td>ISM+ICCG</td>
<td>2.7 Hr</td>
<td>75 %</td>
<td>48 MB</td>
</tr>
<tr>
<td>Original/ICCG</td>
<td>11.2</td>
<td>2.1</td>
<td>7.8</td>
</tr>
<tr>
<td>Original/ISM</td>
<td>146.0</td>
<td>1.1</td>
<td>0.5</td>
</tr>
<tr>
<td>Original/ISM+ICCG</td>
<td>32.4</td>
<td>1.2</td>
<td>9.4</td>
</tr>
</tbody>
</table>
Development of High-speed and High-precision FEM for Analysis of Mechanical Problems in Welding

method, the memory size is reduced to 1/10 by this modification.
As a future research, the algorithm of the interactive substructure method will be improved to achieve both high computational efficiency and reduction of memory size.

References