The University of Osaka
Institutional Knowledge Archive

Title	Development of Dynamic Mesh Refining Method for Large Scale Thermal and Mechanical Analysis in Welding and Line Heating
Author(s)	Huang, Hui; Murakawa, Hidekazu
Citation	Transactions of JWRI. 2013, 42(1), p. 63-70
Version Type	VoR
URL	https://doi.org/10.18910/26595
rights	
Note	

The University of Osaka Institutional Knowledge Archive : OUKA

https://ir.library.osaka-u.ac.jp/

The University of Osaka

Development of Dynamic Mesh Refining Method for Large Scale Thermal and Mechanical Analysis in Welding and Line Heating†

HUANG Hui *, MURAKAWA Hidekazu **

Abstract

Simulation for real scale structures such as automobile and airplane is now gradually being achieved with the aid of computer technology, which can predict the behaviors concerned practically. For welding or heating problems, thermal elastic-plastic FEM has shown its wonderful capability of computing welding distortion and residual stress. In the recent years, many efforts have been made to improve the performance of simulation on large scale structures in aspects of thermal mechanics and computer science. In this paper, we introduced a new method— dynamic mesh refining method (DMRM) which can ensure the accuracy for stress and strain while reducing large amounts of computational time. A background mesh with fine elements in the vicinity of welds or heating lines is designed to update solutions if necessary. Combined mesh for computation has fine elements only near the current welding or heating position. The element connectivity on interfaces between coarse element and fine element is realized by a transformation matrix based on interpolation function. The model is dynamically refined with movement of the heat source. Several numerical examples demonstrated the accuracy and efficiency of proposed method, and a model consisting of over 2 million elements has been accomplished in 66 hours.

KEY WORDS: (Welding), (Line heating), (Thermal elastic-plastic FEM), (Dynamic mesh refining method), (Background mesh), (Computation efficiency)

1. Introduction

Computer simulation plays an important role for design and production in modern industry. Now it becomes possible to perform a systematic evaluation of practical structures by coupling multiple fields. In welding mechanics, the deformation and residual stress are the two main issues of concerned to engineers. So far, several kinds of simulation methods have been proposed to predict the results before practice. Thermal elastic-plastic FEM[1] can reproduce the transient temperature, stress and strain, phenomenon such as phase transformation and work hardening can be taken into account. The inherent strain method makes use of residual strain after welding which is a summation of different strain components. It has been successfully applied to computation of welding distortion in large structures such as stiffened plate and line heating models [2,3]. Estimation of residual stress in multi-pass welding has also been realized by means of inherent strain[4]. Coupled thermal-fluid-solid analysis deals with interaction between different phenomena which are typically presented in friction stir welding[5]. Though it describes the problem more realistically, currently the application is just limited to very small models.

To enhance the performance of thermal elastic-plastic FEM in engineering application, several notable works have been reported. Goldak[6] made the composite mesh for a pipe welding model, in which each mesh part is independent and the continuity of solution across the interface is maintained by constraints. B. Souloumiac [7] developed a new local–global approach for predicting distortions of welded steel component. Murakawa[8] proposed an iterative substructural method (ISM) by partition of the model into strongly nonlinear region and weakly nonlinear region during the welding process. Shibahara[9] extended the dynamic explicit method to welding analysis in which mass matrix and damping matrix are idealized. Research on remeshing has also

† Received on July 8, 2013
* Graduate Student
** Professor

Transactions of JWRI is published by Joining and Welding Research Institute, Osaka University, Ibaraki, Osaka 567-0047, Japan

been reported. Lindgren[10] used the graded element which has variable nodes to simulate the electron beam welding of a large copper canister, Duranton[11] introduced linear constraints by a penalty method on the interfaces of graded elements in calculation of multi-pass welding on a stainless pipe. However, all of these remeshing methods employ coarsening of the rear part which will inevitably bring some error to the analysis. This effect will become more sensitive in multi-pass welding or other process with many heating and cooling cycles, since exact stress and strain of fine elements have been lost during coarsening.

In this study, a dynamic mesh refining method (DMRM) is developed based on the characteristic of welding. We employed a fine mesh (background mesh) to record detailed solutions of the model along time history. Combined fine and coarse mesh (computational mesh) is used to calculate temperature for thermal analysis and incremental displacement for mechanical analysis. The solution field can be kept consistent during mesh refining and coarsening through feedback of stress and strain from fine mesh. Linear hexahedral elements are employed through the model to keep the simplicity of formulation and also high accuracy for plasticity[12]. Mesh density varies from the heat source region to outer regions by grading. Nodal temperature and incremental displacements are linearly interpolated on interfaces between graded elements. Transformation matrices of element stiffness and load vector are built based on interpolation functions of the elements with slave nodes. Nodal variables computed by combined mesh are then interpolated to nodes in the background mesh for updating solutions such as stress and strain. Several numerical examples are shown in this paper to verify the accuracy and efficiency of DMRM.

2. Dynamic mesh refining method

In welding and line heating processes, the material property strongly depends on the temperature which could range from room temperature to several thousands degree. To get accurate solutions by FEM, the mesh grid should be fine enough to capture corresponding material properties at any position. Besides, heat flux can only be exactly distributed on an element which is fine enough. Since temperature has large gradient in the vicinity of heat source while it becomes smoother in other regions, strongly nonlinear state and weakly nonlinear response can be expected for the two regions, respectively. Based on these considerations, the model can be made with fine mesh near heat source and coarse mesh elsewhere, and fine part can be moved with the heat source in simulation. As mentioned in the introduction, a background mesh is employed in the analysis for keeping detailed solutions. All the solutions are updated on background mesh when the model is changed to another computational mesh, as shown in **Fig. 1**.

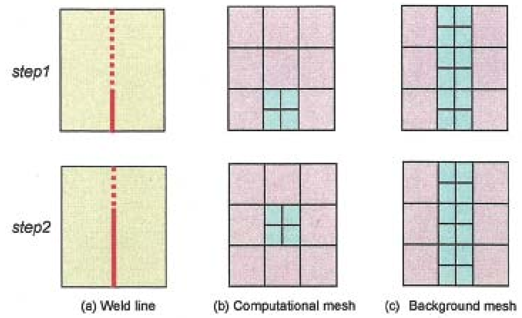


Fig. 1 Basic idea of dynamic mesh refining method

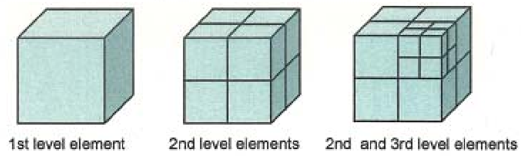


Fig. 2 Hierarchical modeling

2.1 Mesh generation

In the present study, mesh generation is carried out by two steps. Firstly, coarse mesh of the model is generated by any kind of software like Patran and Abaqus. Secondly, an in-house refinement package is employed to subdivide the coarse mesh into fine mesh hierarchically. All hexahedral elements are used for the model, refinement are realized by subdivisions with linear constraints over interface of graded elements (**Fig. 2**). During refining process, interpolation functions for each node are recorded for both stiffness transformation and solution interpolation.

2.2 Refinement criterion

To optimize the mesh refinement, temperature and stress distribution should be taken into account. The reason is that material properties strongly depend on temperature while plasticity depends on stress level. Compared with temperature field, stress distribution will develop in a different way which requires another type of refinement. In this study, two prescribed conditions are proposed for simplicity of the programming. Rycalin has derived the equation for temperature of a plate with finite thickness from semi-infinite body model[13], as shown in **Eq. (1)**.

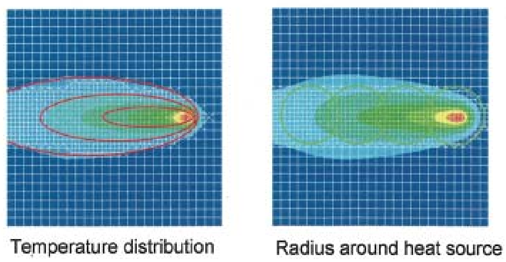


Fig. 3 Two ways to define refinement area

$$\sum_{i=1}^{N}\frac{Q}{2\pi\lambda R_i}\exp\left(-\frac{V(x_i+R_i)}{2a}\right)\geq T \qquad (1)$$

In this equation, λ is thermal conductivity and a is thermal diffusivity, they are assumed to be constants. Q is the net heat input, V is the travelling velocity of heat source. N denotes the number of periodically reiterated layers. x_i is x coordinate of estimated point and R_i is distance from that point to center of heat source with respect to ith layer. It should be pointed out that the equation is only valid for heat sources moving along straight lines, and the effect of plate length or width can not be considered.

$$\left\|\{x_{Heat}\}-\{x_{Element}\}\right\|\leq R \qquad (2)$$

Another option is the radius control around the heat source (**Eq. 2**). Here, $\{x_{Heat}\}$ denotes coordinates of the center of the heat source, and $\{x_{Element}\}$ denotes coordinates of the element centroid. This condition can be easily applied in any kind of welding case.

In addition to the two criteria mentioned above, the combined criterion is also established. It means that if either T or R criterion is satisfied, the element will be refined. It is obvious that smaller value of R and larger value of T are more appreciable from the view point of efficiency.

3. Application of DMRM

To verify the dynamic mesh refining method, several numerical examples were employed to carry out analysis with both original TEP-FEM in a straight forward way and the new FEM. In this study, the material for the models was assumed to be SS400. Work hardening and phase transformation behavior were neither considered. The period for refining was 3 sec for each model. Sequentially coupled scheme was adopted in the simulations. Firstly, heat transfer analysis is performed to obtain transient temperature data for a time interval. Secondly, mechanical analysis will run with the temperature history as load input. All calculations were performed using 1 CPU on Intel Xeon W5580 with Quad Core 3.20GHz, memory size 144GB.

3.1 Bead on plate welding

The welded plate has a dimension of 200×200×6mm, and heat input is 2000W with welding velocity 5mm/s. The heat flux is assumed to follow the Gauss distribution. Size of the coarse element is 8mm×8mm×3mm, and each is subdivided into 8 elements by cutting into two elements in three directions for the background mesh.

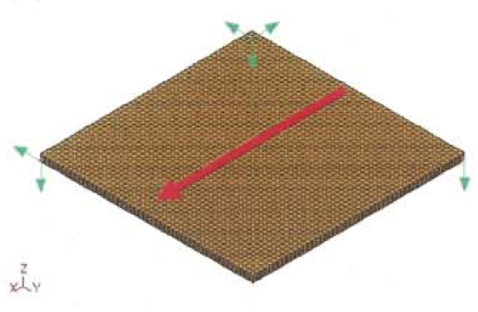


Fig. 4 Bead on plate welding model and boundary conditions

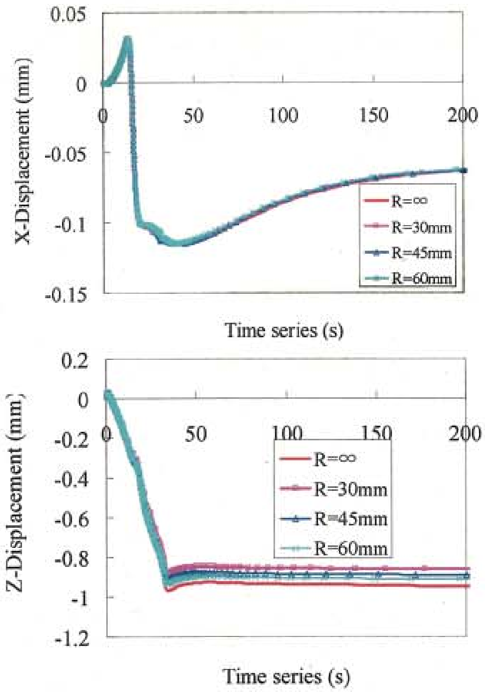


Fig. 5 Transient displacements in X and Z direction of R-criterion models

Boundary conditions in mechanical analysis eliminate the rigid body motions only, as can be seen in **Fig. 4**. To examine the effect of different criteria, eight cases with three refinement criteria were studied (**Table 1**). Here, both R=∞ and T=-∞ correspond to the fine mesh model using original FEM. Transient displacements at the center of plate in X and Z direction are recorded to compare the different cases (**Fig. 5-Fig.7**).

Table 1 Studied cases with different refinement criteria

Refinement criterion	R-criterion (mm)			T-criterion (℃)			Combined-criterion R .or. T (mm, ℃)	
Case	1	2	3	4	5	6	7	8
Parameter (R, T)	30	45	60	200	150	100	100,30	100,45

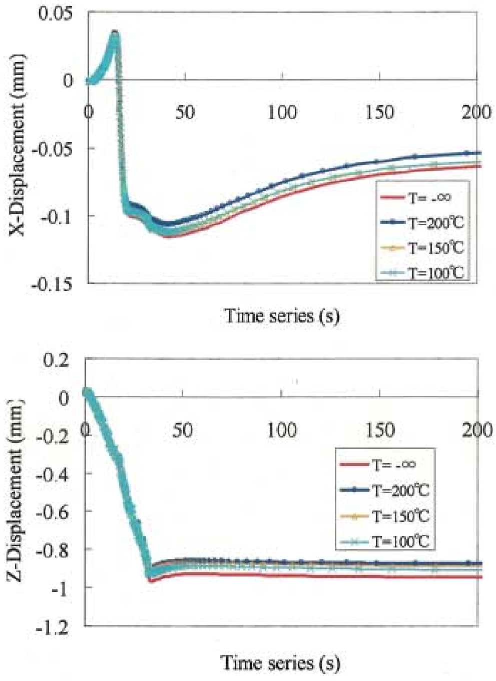


Fig. 6 Transient displacements in X and Z direction of T-criterion models

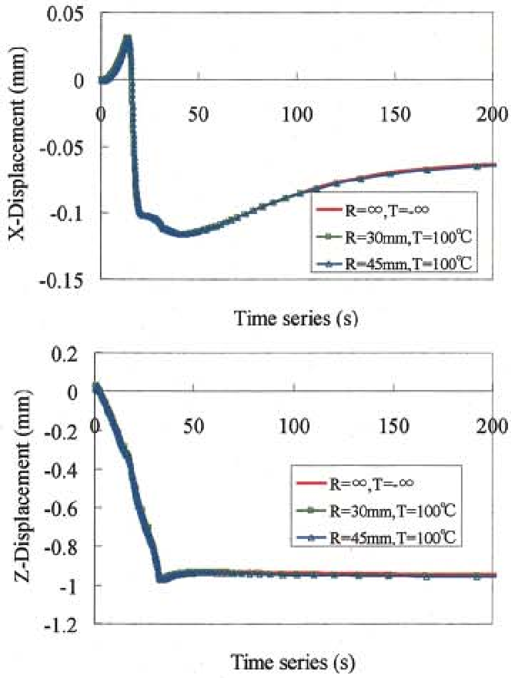


Fig. 7 Transient displacements in X and Z direction of Combined-criterion models

In the case of the radius criterion, the effect on X-Displacement is less sensitive than Z-Displacement. It can be seen that observable difference exists for Z-Displacement even if R becomes the relatively large value of 60 mm.

In the case of temperature criterion, as the value of parameter T decreases, the result of displacements in X and Z direction become closer to the standard one. Still, the slight discrepancy remains for T=100℃ which is very low compared with melting temperature.

To improve accuracy while not increase element number significantly, case 6 is modified to case 7 and case 8 by accounting the effect of refinement with radius around heat source. It can be found that very good agreement between results from mesh refining method and original method is obtained.

For case 8, the residual stresses along weld line and middle cross section of the plate are plotted in **Fig. 8** and **Fig. 9**. In this example, computational time has been reduced by 50%.

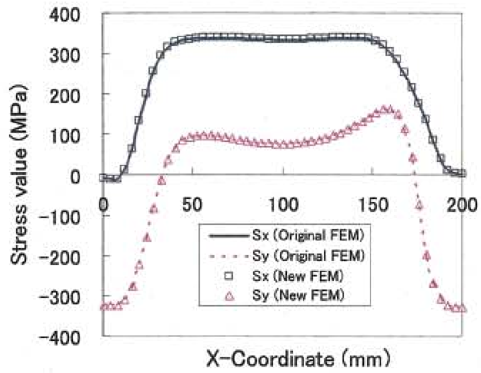


Fig. 8 Comparison of residual stresses along weld line

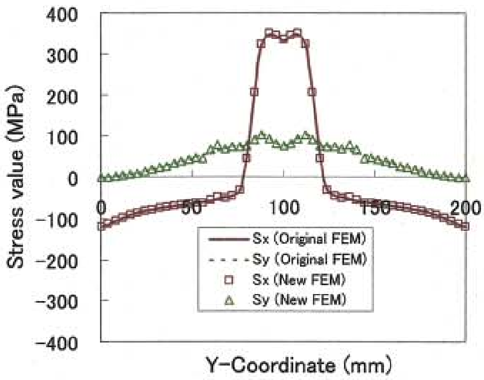


Fig. 9 Comparison of residual stresses along middle cross section

3.2 Line heating on a rectangular plate

For undevelopable surface, it is always difficult to obtain from flat plate by conventional mechanical process like bending and pressing which is effective to form developable surface. Line heating along multiple paths can generate necessary inherent strain to achieve complex target shape. To examine the performance of the proposed FEM on practical line heating, we chose a large scale plate with 18 heating lines in which line 1~9 lies on the top surface and line 10-18 on the bottom surface, as shown in **Fig. 10**.

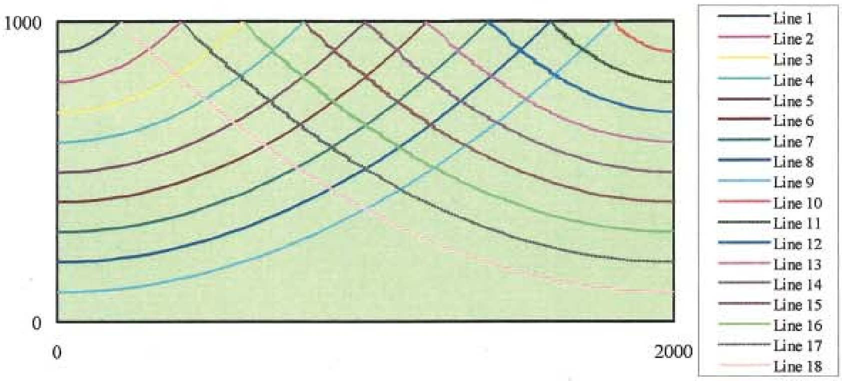


Fig. 10 Configuration of heating lines

The plate has a dimension of 2,000mm in length, 1,000mm in width and 8mm in thickness direction. Each heating line takes the form of parabola. The corresponding equations are as follows:

$$y=\begin{cases}\dfrac{x^2}{400n}+(1000-100n) & (1\le n\le 9)\\[2mm] \dfrac{(x-2000)^2}{400(n-9)}+(1900-100n) & (10\le n\le 18)\end{cases} \quad (3)$$

Gaussian surface heat source was employed in the heat transfer analysis and net heat input is 3,600W with a moving speeding 26mm/s for all heating lines. For this simulation, gravity force and supporting system which exist in practical cases have been neglected.

The model used in original FEM consists of 258,405 nodes and 204,800 elements with element size 6.25×6.25×2.00 mm. For the new FEM, mesh in the vicinity of heating paths has the same size with that in original finite element model. Twisted distortion has occurred after all heating lines were completed, as indicated by the contour of Z-displacement in **Fig. 11**.

Residual stress contours for x component and y component are shown in **Fig. 12** and **Fig. 13**, respectively. Results of analysis by DMRM are plotted on the background mesh from which the interfaces between coarse mesh and fine mesh can be seen. Through comparison between stresses from two analyses, very similar distribution can be observed.

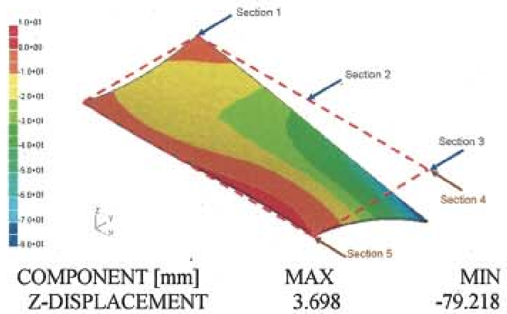
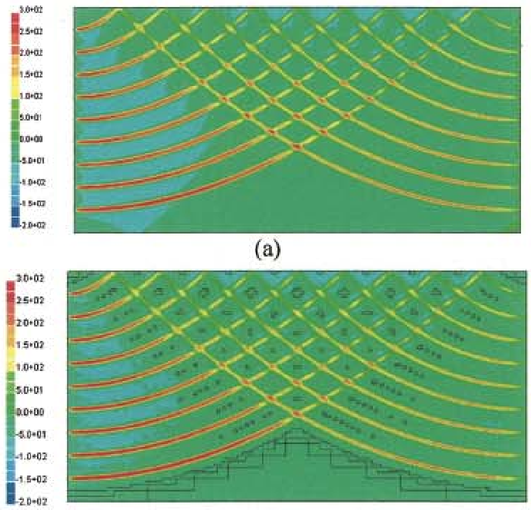


Fig. 11 Contour of Z-displacement at final stage (Scaled by 5 times)

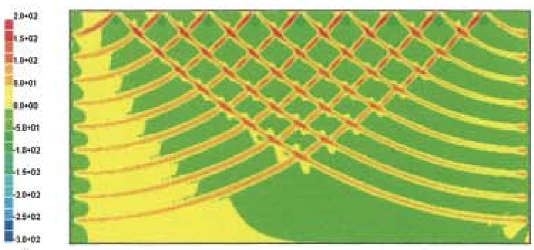
To quantitatively examine the results, out of plane deformation on typical sections are evaluated to make comparison between the two different methods. **Figure 14(a)** shows the results along the top surface of the Section1-3 in transverse direction. It can be seen that deflection on the three sections indicates the double curvature of the final deformed shape. **Figure 14(b)** shows the results along the top surface of Section4 and Section 5. The maximum value of deflection almost reaches 80 mm. From the two figures, it can be found that deflections in the two analyses correspond very well along each evaluated sections.



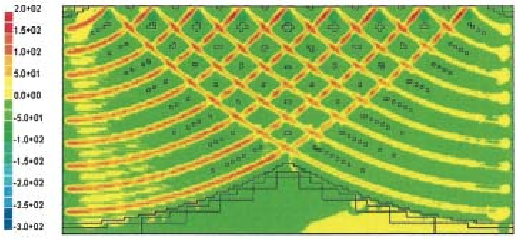
(a)

(b)

Fig. 12 Contour of residual stress in x direction: (a) Original FEM, (b) New FEM

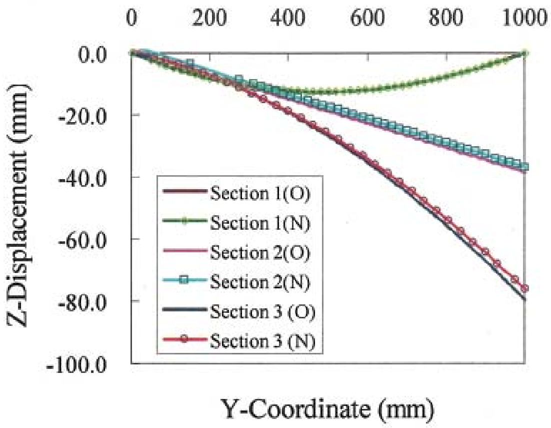


(a)

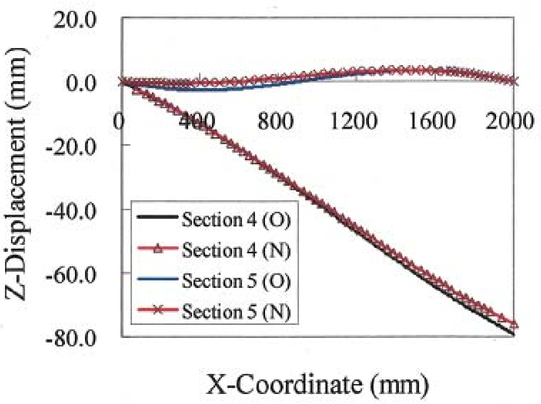


(b)

Fig. 13 Contour of residual stress in y direction: (a) Original FEM, (b) New FEM



(a)

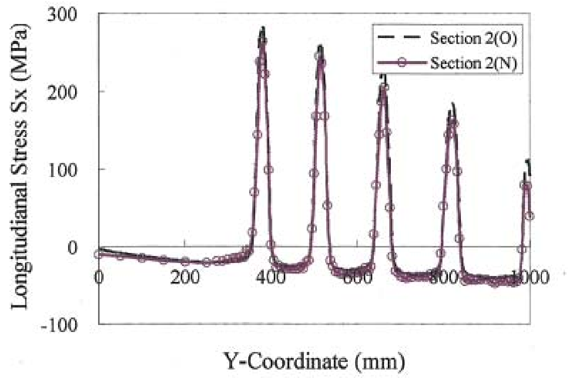


(b)

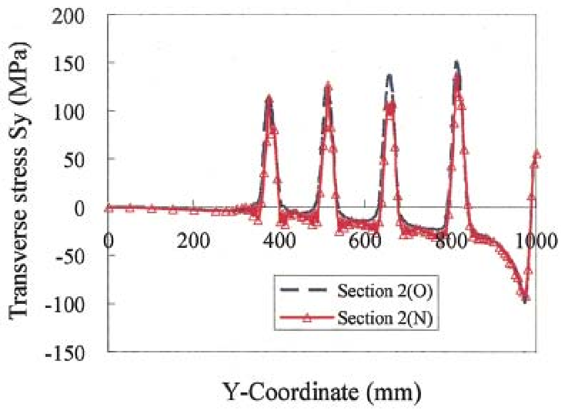
Fig. 14 Comparison of deflections obtained with original FEM and new FEM: (a) Along transverse direction (b) Along longitudinal direction

Residual stresses in longitudinal direction and transverse direction for Section 2 are plotted in **Fig. 15**. The data points include the five cross points of heating lines, where repeated heating and cooling took place. The results obtained by new FEM agree very well with that by original one, except for small difference in the vicinity of cross points.

In this example, elements in the mesh refining model were graded into four levels. Total analysis steps for the simulation is 8899. The calculation with original FEM took 267 hours while the one with new FEM took only 21 hours. The computational speed and accuracy can be further improved by optimizing refinement process, choosing appropriate refining period and hierarchical mode.



(a)



(b)

Fig. 15 Comparison of residual stresses: (a) Longitudinal stress (b) Transverse stress

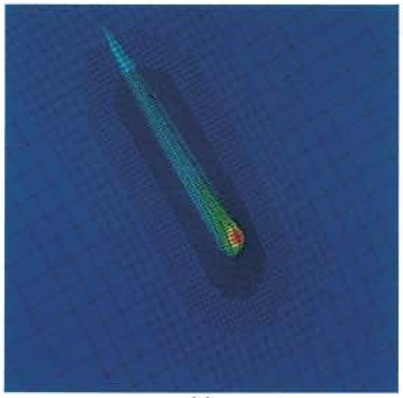
3.3 Circumferential heating on a large pipe

For large scale simulation of welding, it usually requires large amount of computation time and physical memory. On the other hand, the mesh could become very difficult to generate for a model with complex welding or heating paths. The dynamic mesh refining method can be used to refine model and obtain the optimized mesh easily. In the following example, analysis of circumferential heating on a large pipe with over 2 million elements was performed by DMRM (**Fig. 16**). Heating condition and model information are shown in **Table 2**. The computational mesh of the model has five level elements and size of smallest element is 2.5mm by 1.25mm by 0.375mm. Initial coarse mesh has 30 divisions in axial direction, 2 divisions in thickness direction and 314 in circumference direction. Adaptive refinement has been made automatically for the computational mesh and background mesh.

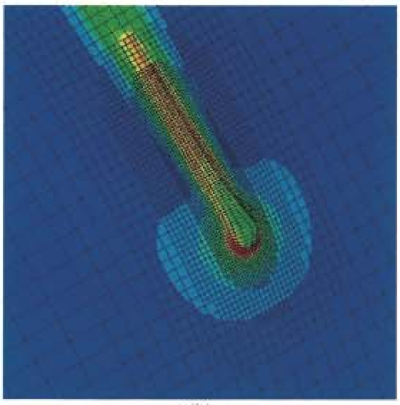
Table 2 Heating condition and model information of the pipe

Dimension L×R×T (mm)	Heat input Q (W)	Velocity mm/s	Heating lengths mm	Node No.	Element No.	DOF No.
1,200×1,000×12	4,800	30	6,283	2,403,984	2,231,912	7,211,946

Fig. 16 Background mesh of the pipe model



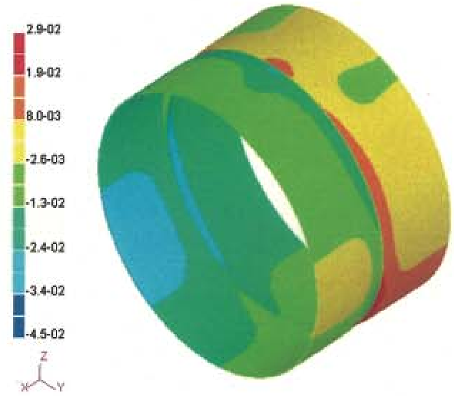
(a)



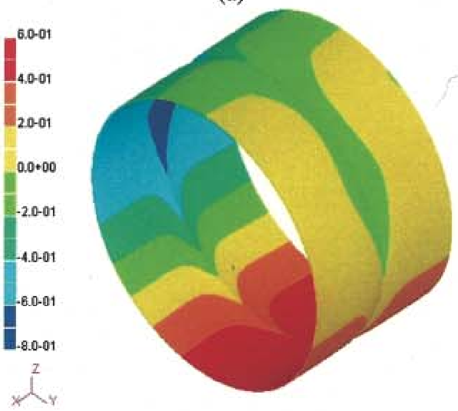
(b)

Fig.17 Contours of (a)Transient temperature and (b)Mises stress

Figure 17 shows the transient temperature distribution and the Mises stress in the vicinity of heat source, and local refinement can be seen from this figure. From the contours, it can be found that the distribution of temperature and stress through regions with different mesh density are very continuous and smooth. It proves that representation of the model using mesh with variable density is reliable. In this way, the unknowns to describe the model have been reduced, which leads to the acceleration of computation. It should be noted that the distribution of the two fields are quite different, the stress field dominates the region to be refined. To achieve high accuracy of solution, trial analysis can help to obtain appropriate parameters for refinement.



(a)



(b)

Fig.18 Heating deformation (mm, scaled by 100 times): (a) Axial component (b) Radial component

Residual deformation components are plotted in **Fig. 18**. It can be observed that transverse shrinkage and angular distortion has been produced by line heating. For this analysis, there were 2134 steps and computation time was 66 hours by one CPU only. Currently, simulation of the model by original FEM could not be performed due to the lack of physical memory.

4. Conclusions

Based on the results from this study, the following conclusions were drawn:

(1) A new Thermal Elastic-Plastic FEM—dynamic mesh refining method has been developed with the idea of background mesh. Accuracy of the method was verified by a bead on plate welding model in respect of displacement and stress.

(2) Combination of radius criterion and temperature criterion for refinement was found to be superior to each criterion alone.

(3) Simulation on a line heating model with 775,209 unknowns and 8899 steps has been successfully accomplished in 21 hours by the dynamic mesh refining method, which suggests a computation speed of over 10 times faster than original method.

(4) Analysis of heating over a large pipe model consisting 7,211,946 unknowns has been completed in 66 hours.

(5) Both CPU time and cache memory has been reduced to a great extent, since the mesh refining model always forms stiffness matrix on combined mesh. It is practically very useful to solve large scale problems.

Reference

1) Y. Ueda, T. Yamakawa, Analysis of thermal elastic-plastic stress and strain during welding by finite element method, Transactions of the Japan Welding Society, Vol. 2 (2), pp. 186-196, 1971

2) D. Deng, H. Murakawa, and W. Liang, Numerical simulation of welding distortion in large structures. Computer Methods in Applied Mechanics and Engineering, Vol. 196 (45), pp. 4613-4627, 2007.

3) Y. Ueda, H.Murakawa, A. M. Rashwan, R. Kamichika, M. Ishiyama, J. Ogawa, Development of computer aided process planning system for plate bending by line-heating (Report IV): Decision making on heating conditions, location and direction. Transactions of JWRI, Vol. 22(2), pp. 305-313, 1993

4) K. Nakacho, T. Ohta, N. Ogawa, N. Ma, H. Hamaguchi, M. Satou, M. Nayama, Measurement of welding residual stresses of reactor vessel by inherent strain method: measurement of residual stresses of pipe–plate penetration joint. Welding International, Vol. 23(6), pp. 439-449, 2009

5) E. D. Schmitter, Modelling massive forming processes with thermally coupled fluid dynamics. Proceedings of the COMSOL Multiphysics User's Conference. 2005.

6) J. A. Goldak, M. Mocanita, V. Aldea, J. Zhou, D. Downey, A. Zypchen, Is real time CWM feasible? Recent progress in CWM. 5 International Seminar Numerical Analysis of Weldability IIW Com. IX, Graz-Seggau Australia, pp. 421-430 , 1999

7) B. Souloumiac, F. Boitout, J. Bergheau, A new local–global approach for the modelling of welded steel component distortions. Mathematical Modelling of Weld Phenomena 6, Institute of Materials, Minerals and Mining, Graz, Austria, 2001.

8) H. Murakwa, I. Oda, S. Itoh, H. Serizwa, M. Shibahara, H. Nishikawa, Iterative substructure method for fast FEM analysis of mechanical problems in welding, Preprints of the National Meeting of JWS, Vol. 75, pp. 274-275, 2004

9) M. Shibahara, K. Ikushima, Development of analytical method for welding mechanics using idealized explicit FEM, Transactions of JWRI, Vol. 39(2) , pp.384-386, 2010

10) L. E. Lindgren, H. A .Haggblad, J. M. J. McDillb, a.s.oddy, Automatic remeshing for three-dimensional finite element simulation 0f welding, Computer Methods in Applied Mechanics and Engineering, Vol. 147(3), pp. 401–409, 1997.

11) P. Duranton, J. Devauxa, V. Robin, P. Gilles, J.M. Bergheau, 3D modelling of multipass welding of a 316L stainless steel pipe, Journal of Material Processing Technology, Vol. 153-154, pp. 457-463, 2004

12) S. E. Benzley, E. Perry, K. Merkley, B. Clark, G. Sjaardama, A comparison of all hexagonal and all tetrahedral finite element meshes for elastic and elasto-plastic analysis. Proceedings of 4th International Meshing Roundtable, Vol. 17, pp. 179-191, 1995

13) D. Radaj, Heat effects of welding: Temperature field, residual stress, distortion, Springer-Verlag, pp. 54-55,199