Title	Study of Temperature Field and Inherent Strain Produced by High Frequency Induction Heating on Flat Plate(Mechanics, Strength & Structural Design)
Author(s)	Luo, Yu; Ishiyama, Morinobu; Murakawa, Hidekazu
Citation	Transactions of JWRI. 2004, 33(1), p. 59-63
Version Type	VoR
URL	https://doi.org/10.18910/8867
rights	
Note	

Study of Temperature Field and Inherent Strain Produced by High Frequency Induction Heating on Flat Plate †

LUO Yu*, ISHIYAMA Morinobu ** and MURAKAWA Hidekazu***

Abstract

In the heating and bending process for ship hull plates, high frequency induction heating is replacing traditional torch flame heating. In this paper, the finite element model of high frequency induction heating source is developed based on the feature of high frequency induction heating. In this model, the temperature field of high frequency induction heating is analyzed, and the validity of this model is confirmed by experimental results. Using the developed heat source model, the deformations have been calculated. Further, based on the knowledge obtained through the study, simple formulae to calculate the inherent strain of high frequency induction heating are proposed. Using the inherent strain given by the formulae, an elastic finite element method is developed to predict the deformation produced by heating and bending.

KEY WORDS: (High frequency induction heating) (Temperature field) (Plate bending) (Inherent strain)

1. Introduction

Plate bending by line-heating plays a very important role in the production of plate structures in shipbuilding. Traditionally, torch flame heating and water-cooling forming is often adopted. However, when the torch flame heating is used, it is very difficult to use a computer because the heating temperature and its varied range cannot be controlled precisely. The traditional method can only be achieved through experience, which results in a series of problems, such as low efficiency, shape error and so on. Recently, high frequency induction heating method has been introduced for plate bending in the shipbuilding industry. Using the high frequency induction heating method, it is possible to employ computer-aided processes for plate bending [1~6]. In order to achieve computer-aided processes of bending plate by high frequency induction heating, the features of the temperature field should be studied first. In this paper, a finite element model is developed based on the features of high frequency induction heating. The temperature field of high frequency induction heating is analyzed with this model and the validity of the model is confirmed by experiments. Then, a specific heating model is used to compute the temperature field and deformation by high frequency induction heating .

Further, based on the knowledge obtained through the study, simple formulae to calculate the inherent strain in high frequency induction heating are proposed. Using the inherent strain given by the formulae, an elastic finite element method is developed to predict the deformation produced by heating and bending.

2.Feature of High Frequency Induction Heating Source

In the process of plate bending by high frequency induction heating, high frequency induced current is utilized in the inductive loop, whereby the plate surface is heated. Therefore, the features of the inductive heat source should be studied first. The heat source can be considered as a round surface heat source because of the high induction frequency. Based on theoretical and experimental analysis, the features of the heat source model can be described as **Fig. 1**. Heat current density is smaller in the center of inductive loops and increases gradually when the radius increases away from the center. When the heat current density achieves its maximum, it keeps a constant value in a small range, and begins to decrease slowly with increasing distance.

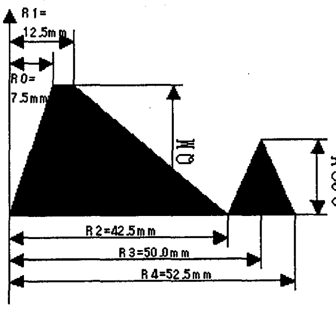


Fig. 1 Heat source model of electromagnetic induced heating.

† Received on June 8, 2004
*Associate Professor (Shanghai Jiaotong University, China)
** IHI Marine United Inc.
***Professor

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

difficult to decide the radius and peak value of heat current density for different heat sources. So, determining the specific parameters of the heat source model is important.

3. Determination of Parameters of Heat Source Model

In this study, the parameters of the heat source are assumed at first, and then confirmed by comparison with a large quantity of FEM to experimental results. The simulation model and finite element meshes are shown as **Fig.2**. In order to save calculation time, in the heating line direction, non-uniform finite element meshes are adopted. Considering the symmetry, half of the model is chosen.

In the temperature field analysis, a 3-D element with 8 nodes is adopted. The finite element model has 4,320 elements and 5,453 nodes. Experimental and analytical conditions are as follows:

Size of the plate: 2m×2m
Thickness of the plate: 20mm
Power of the heating source: 37kw
Efficiency: 50%
Diameter of inductive loops: 80mm
The material used in this study is mild carbon steel whose thermal physical properties are shown in **Fig. 3**.

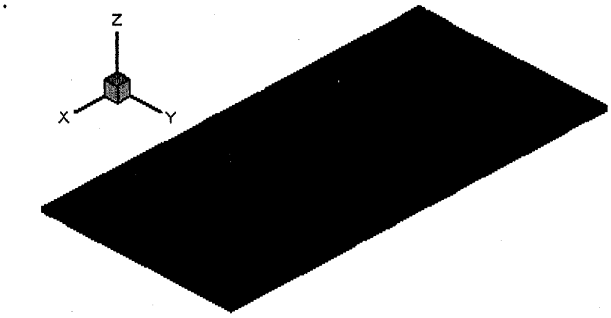


Fig. 2 Simulation finite element model.

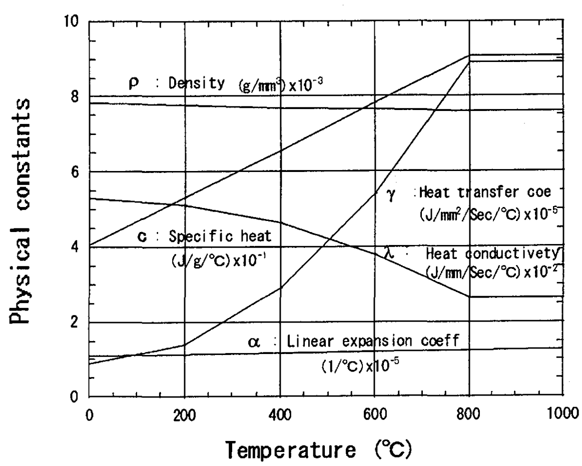
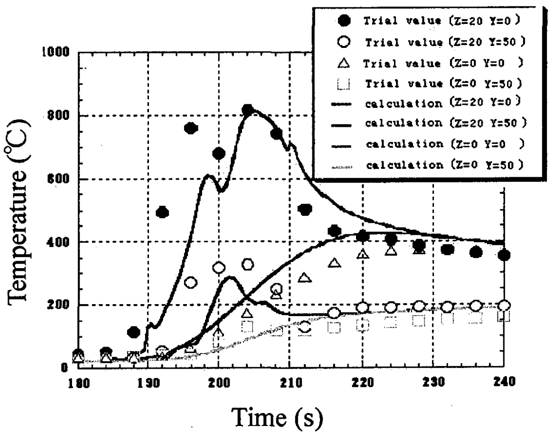


Fig. 3 Thermal physical properties of mild carbon steel.

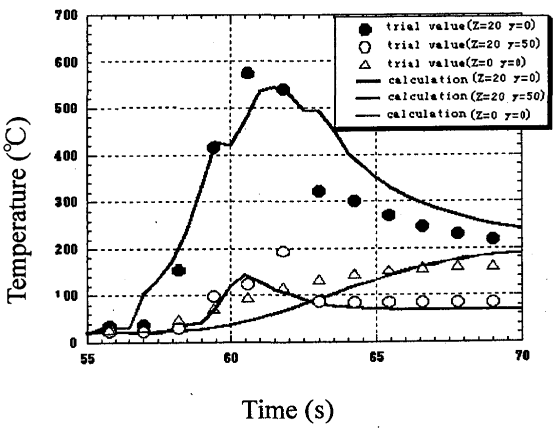
The parameters of the heat source are assumed as follows: R0=7.5mm, R1=12.5mm, R2=42.5mm, R3=50.0mm, R4=52.5mm, Qmax1=QM, Qmax2=0.6QM

Two cases in which the moving speed of the heat source are 300mm/min and 1000mm/min were simulated.

Thermal cycle curves at different point of the plate are shown as Fig4. In Fig.4(a) and Fig.4(b), the comparison between experiments and computations with the different speeds of 300mm/min and 1000mm/min are presented The results show that experimental values basically accord with calculations. Therefore, the assumed parameters can be considered to be valid.



a) Moving speed of heat source:300mm/min



b) Moving speed of heat source:1000mm/min

Fig. 4 Thermal cycle curves at different point of the plate.

4. Finite Element Simulation of Swing Heating

The temperature field of a typical swing heating process is analyzed by adopting the parameters of the specific heat source model.

4.1Analytical Conditions

The research model is shown as **Fig. 5**. The calculation is performed through ANSYS SOLID 70 which is composed of 8 nodes, and 5280 elements, 6601 nodes. Analytical conditions are as follows:

Dimensions of the plate : 1m×1m
Thickness of the plate: 20mm
Power of the heat source: 37kw
Efficiency: 50%
Diameter of inductive loops: 80mm
Material is mild carbon steel. The thermal physical properties are shown in Fig. 3.
As an example, the parameters of the swing heating source are selected as follows:
Moving speed of heat source: 300mm/min;
Swing half-amplitude of heat source: DYD=30mm
Distance marching forward along axis-X within one period:
DXD=100mm
Heat input: Q=10.45J/mm

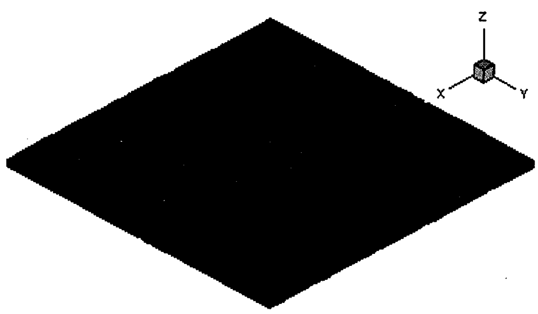
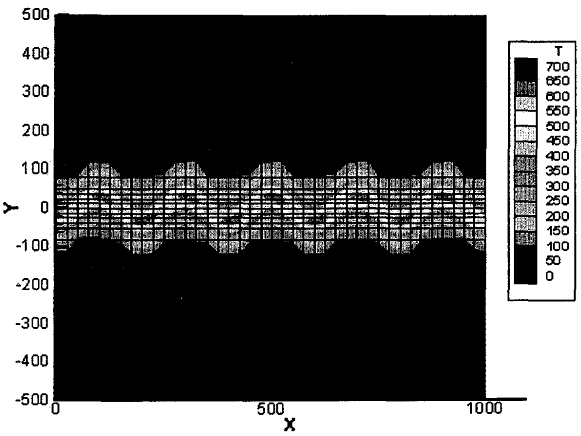
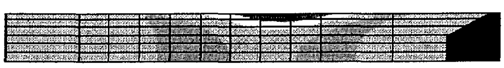


Fig. 5 Simulation finite element model for swing heating



a) Surface



b) Center of cross section

Fig. 6 Distribution of the maximum temperature of the plate.

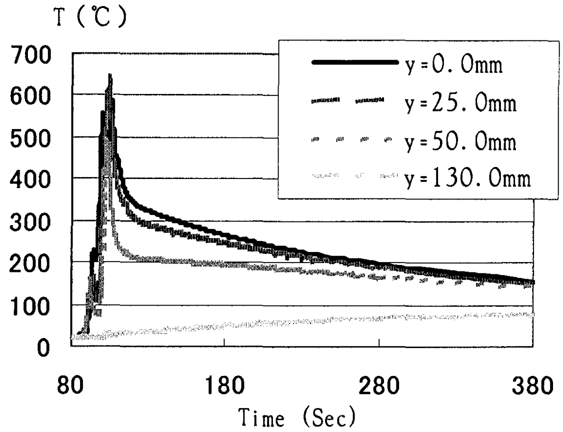


Fig. 7 Thermal cycle curves of plate surface points.

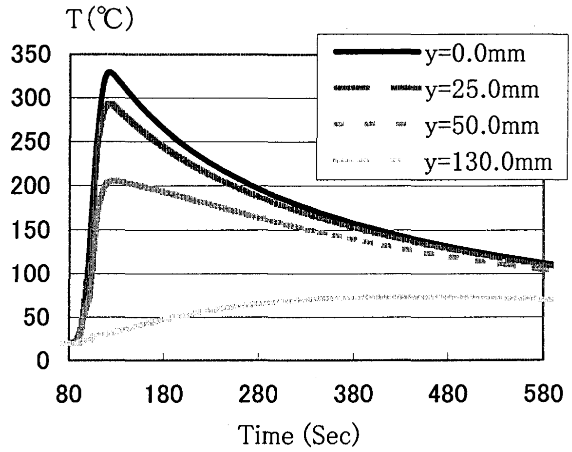


Fig. 8 Thermal cycle curves of plate inner points.

4.2 Analytical results

The temperature field of the swing heating is achieved based on FEM calculations. The maximum temperature at each point on the plate surface and in the middle cross section of the plate is shown in **Fig. 6**. Thermal cycle curves on the surface and the inner points of the plate are shown as **Fig. 7** and **Fig. 8**.

5. Calculation of Inherent Strain

Considering the formulae for inherent strain produced in welding process [6], the formulae for inherent strain produced in the high frequency induction heating process are proposed as following:

$T_{1x} > T_{max}$

$$\varepsilon_x = 0 \qquad (1)$$

$$\varepsilon_y = 0$$

$T_{1x} < T_{max} < T_{2x}$

$$\varepsilon_x = -(B\varepsilon_Y/\beta) * (T_{max} - T_{1x})/(T_{2x} - T_{1x}) \qquad (2)$$

$$\varepsilon_y = 0$$

$T_{max} > T_{2x}$

$$\varepsilon_x = -B\varepsilon_Y/\beta \qquad (3)$$

$$\varepsilon_y=-A\alpha(T_{max}-T_{2x}) \quad (4)$$

where,

T_{max}: the highest temperature during heating
$\varepsilon_Y=\sigma_Y/E$
$T1x=\sigma_Y/\beta\alpha E$
$T2x=2\sigma_Y/\beta\alpha E$
β: constraint coefficient in x direction
α: coefficient of linear expansion
E: Young's Modulus
B: the parameter relating to ε_x
A: the parameter relating to ε_y

Based on a large amount of T.E.P FEM calculations, the relationship between parameter A and heat input parameter (Q/h^2) was obtained as shown in **Fig. 9**. The relationship between the parameter B and Q/h^2 is shown in **Fig. 10**.

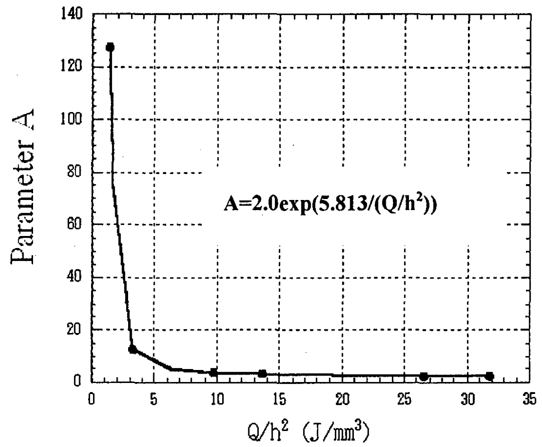


Fig. 9 Relationship between parameter A and Q/h^2.

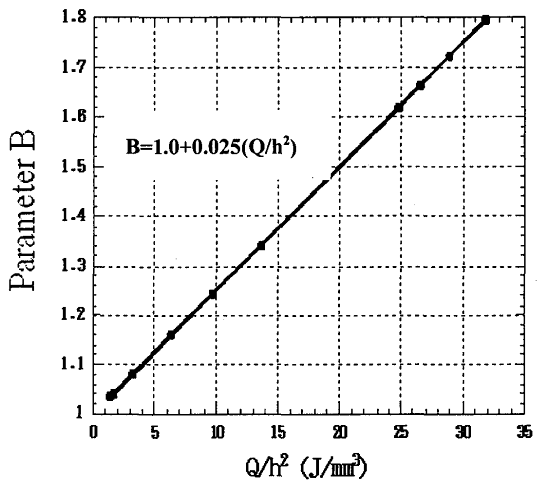


Fig. 10 Relationship between parameter B and Q/h^2.

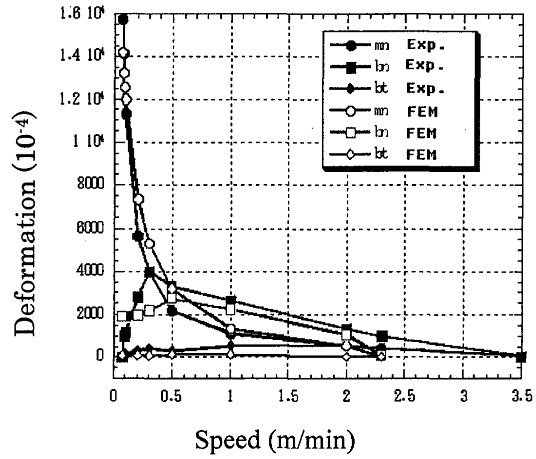


Fig. 11 Comparisons of simulated results and experiments.

6. Simulation Results of Elastic FEM

In this study, based on the inherent strain concept, an elastic finite element method was developed to predict the deformation due to high frequency induction heating. When the parameters A and B were determined, T_{max} could be used to determine the inherent strains. The obtained inherent strains are introduced into the elastic FEM, and the final deformation can be calculated.

For the model in which the length, the width and the thickness of the model are 1000mm, 1000mm and 20mm respectively, the simulated results and the experimental results are compared in **Fig. 11**. In this figure, the point marked "mn" represent the shrinkages in the heating direction (X-direction), "bn" bending deformation in the vertical direction (Y-direction), and "bt" bending deformation in the heating direction.

In this figure, it can be seen that the calculated results are in good agreement with the experiments. By comparison with the experimental results, we can conclude that the elastic FEM based on the inherent strain concept is an effective method to predict the deformation due to high frequency induction heating and bending.

7. Conclusions

1) The features of a high frequency induction heating source are as follows: heat current density is lower beneath the round inductive loops; and it increases gradually with increasing radius far from the heating center. When the density reaches a maximum, it maintains its for a distance and then begins to decrease. At the same time, a triangle wave crest of heat current density appears in the direction of the radius.

2) The specific parameters of the high frequency induction heating source model are determined by the combination of experimental results and computation.

3) The relationships between the inherent deformations and heat input (Q) or parameter (Q/h^2) have been calculated. These relationships can be used as a database for the elastic FEM.

4) Based on the inherent strain concept, an elastic FEM has been developed to predict the final deformation due to high frequency induction heating and bending. The predicted results are in good agreement with the experimental measurements. Therefore, the elastic FEM is a very effective method for predicting deformation.

Acknowledgement

The present work was sponsored by the Natural Science Foundation of China. (Project No. 50175973).

References

1) S. Imatani et al. Experiment and Simulation for Thick-Plate Bending By High Frequency Inductor. Acta Metallurgical Sinica (English Letter) 1998(6): 449-455

2) Jerzy Zgraja et al. Induction heating of large steel disks: Coupled high frequency, thermal and mechanical simulation. Int. J. of Applied High frequencies and Mechanics 1999 (10): 303-313

3)Jong Gye Shin et al. A numerical Thermoplastic Analysis of Line Heating Processes for Saddle-type Shells with the Application of Artificial Neural Network. Journal of Ship Production 1999(1):10-20

4) Yukio Ueda, Hidekazu Murakawa et al. Development of Computer Aided Process Planning System for Plate Bending by Line-Heating (Report 1)—Relation between the Final Form of Plate and the Inherent Strain. Transactions of JWRI 1991, (2): 129-139

5) H. Murakawa, Y. Luo and Y. Ueda, "Inherent Strain as an Interface between Computational Welding Mechanics and Its Industrial Application", Mathematical Modelling of Weld Phenomena, Vol.4 (1998), pp.597-619.

6)Y. Luo, H. Murakawa and Y. Ueda,"Prediction of welding deformation and residual stress by Elastic FEM based on inherent strain (report I)", Trans. JWRI., 1997, 26(2), 49-57.