Journal of Achievements in Materials and Manufacturing Engineering

VOLUME 17
ISSUE 1-2
July-August
2006

Thermo-viscoelastic-plastic deformation of huge products in thermal process

C. Cho [a,*], S.-Y. Kwahk [b], W.-J. Kim [a], J.-K. Choi [b]
[a] Department of Mechanical Engineering, Inha University,
253 Yonghyun Dong, Nam Ku, Incheon, 402-751, S. Korea
[b] Advanced Material R&D Center, Korea Institute of Industrial Technology,
Dongchun Dong, Yeonsu-Ku, Incheon, S. Korea
* Corresponding author: E-mail address: cdcho@inha.ac.kr

Received 15.03.2006; accepted in revised form 30.04.2006

Analysis and modelling

ABSTRACT

Purpose: This paper is to numerically predict thermo-elastic-plastic deformation during thermal process. Decreasing material strength in thermal processes causes severer deformation at elevated temperature even under self-weight of huge and heavy products.
Design/methodology/approach: A hybrid method is proposed and applied, in this study, to analyse thermomechanically coupled problems such as heat treatment. The finite difference method (FDM) and the finite element method (FEM) are prefered. In general, FDM is favored for heat/fluid flow, FEM for structure analysis.
Findings: The solution of heat treatment processing is conducted by using the proposed hybrid method that we developed the numerical program for calculating the deflection induced due to its own weight and the creep. The code is verified by the analytic solution of a simple plate model.
Research limitations/implications: There have been developed peculiar computational methods fitted in each single field problem. Recent problems necessitate total solutions not only in a major relating science but also in adjacent engineering parts. To keep the efficiency of respective methods even in coupled field problems, it is very desirable to combine advantages of respective methods as hybrid technique. This study suggests and applies a hybrid method of FEM and FDM for simulating heat treatments. Further improvement to convert different types of computational models to each other is one of important issues.
Practical implications: In the past half century, there have been developed numerous computational techniques in various fields in separate ways. Hybrid method of combining existing computational techniques rather than further extending the techniques may have significant implication whenever practical problems necessitate total solutions coupled over multiple physics.
Originality/value: The concept of a hybrid technique between FEM and FDM was implemented in this study and applied to simulate a heat treatment case as a multiphysical problem.
Keywords: Computational mechanics; Thermaldeformation; Finite Element Method; Finite Difference Method

1. Introduction

In heat treatment process as well as casting, although computing thermal stress[1,2] has been major interests in engineering, various difficulties in conducting computation are still unsolved mainly due to coupling characteristics between temperature, micro-structrure, stress, and chemical-physical properties.[3-6] Heat treatment process is done to obtain desired mechanical properties. This process requires a tedious and expensive experimentation to find the process parameters before setting final process steps. Development of reliable and efficient computation softwares can control process parameters and save time cost to achieve the desired microstructure and mechanical properties.

In a general heat treatment process, the work-piece is austenized by heating and immersing in a coolant such as water. Thermal gradients due to the rapid temperature change induce the thermal stress in products. Heat treatment has products deformed due to their own weight as well as temperature variation. Besides, creep induces large deformation in products when set in the solute state at elevated temperatures for a long time.

In the past two decades, many studies have been reported to focus on cooling steps. [7,8] Only a few studies on the change of microstructure at the heating step have been made [9] and little knowledge is reported about deformation at high temperature environment of heat treatment process. To precisely analyze thermoelastoplastically coupled problems while heat treated, the understanding of initial process on heating products is important.

As computational skills over coupled problems, we propose and appy a hybid method between the finite element method (FEM) and finite difference method (FDM) to heat treatment problems while few studies by Grill et. al [10] and Hou et. al [11] are found in casting process.

2. Theoretical background

2.1. Weight

The product weight is the gravity force over whole volume and is considered a body force in the mechanical applications. The potential energy of a system is expressed by the follow equation.

$$\Pi = \frac{1}{2}\int_v \varepsilon^T D_e \, \varepsilon \; dv - \int_v u^T b \; dv - \int_A u^T T \, dA - \sum_i u_i^T P_i \tag{1}$$

where the first term is deformation energy of element , b, T, and P are body force, surface force , and point force, respectively. The body force is expressed in the finite element analysis by

$$F_b = \sum^N \int_{v_e} u^T b dv = d^T \int_{v_e} [N]^T bdv \tag{2}$$

where, N is number of elements and [N] is shape function. The detailed finite element equation is discussed in the next paragraphs.

2.2. Creep

Generally, solids deform only in response to the applied loads but they can also get deformed under certain conditions, especially when the solid is subject to a high temperature environment. Progressive deformation of solids can be observed even at a constant load or stress under this circumstance. Such behavior is called creep deformation. Creep deformation is regarded as a time-dependent inelastic deformation of solids.

Creep damage for most materials becomes significant at a temperature above $0.4T_m$, where T_m is the melting temperature of the material, e.g. about 260°C for aluminum alloys; 520°C for steels. General form of creep law is

$$\varepsilon_c = f(\sigma, t, T) = f_1(\sigma) f_2(t) f_3(T) \tag{3}$$

where σ , t, and T are applied stress, time, and temperature, respectively . For most engineering applications, the following Norton's equation is used

$$\varepsilon_c = \alpha \sigma^{\beta} t^{\gamma} \tag{4}$$

Table 1 shows the experimental creep constants for some materials [12].

Table 1.
Creep constants of some materials [12]

Material	Temp ^{0}C	E GPa	α for MPa, hours	β	γ
SAE 1035 Steel	524	161	1.58×10^{-11}	4.15	0.40
Copper Alloy 360	371	85.5	4.26×10^{-9}	4.05	0.87
Pure Nickel	700	150	2.42×10^{-6}	2.50	0.28
7075-T6 Aluminum	316	36.5	1.35×10^{-13}	7.00	0.33
Cr-Mo-V Steel	538	152	1.15×10^{-9}	2.35	0.34

3. Numerical model

3.1. Computational procedure

In this study we use a hybrid numerical analysis technique: FDM for the heat flow and FEM for the stress. A temperature field data conversion scheme in a three-dimensional space is required in order to employ the FDM/FEM hybrid technique and then the residual stress analysis based on FEM used converted temperature field data. So that, an efficient data conversion procedure based on linear interpolation has been developed. The detail procedure about the FDM/FEM hybrid method and data conversion procedure can be refer to the reference [13].

3.2. Finite element formulation

In our thermo-elasto-plastic model, the total incremental strain components may be assumed to consist of following components [14]

$$\{d\varepsilon\} = \{d\varepsilon_e\} + \{d\varepsilon_p\} + \{d\varepsilon_T\} + \{d\varepsilon_c\} \tag{5}$$

where $d\varepsilon_e$, $d\varepsilon_p$, $d\varepsilon_T$, $d\varepsilon_c$ are elastic, plastic, thermal, and creep incremental strains respectively. Incremental strain-stress relation can be expressed as

$$\{d\sigma\}=[D_e]\{\{d\varepsilon\}-\{d\varepsilon_T\}-\{d\varepsilon_p\}-\{d\varepsilon_c\}\} \tag{6}$$

where $[D_e]$ is elasticity matrix. For elasto-plastic analysis, we define a yield function, a hardening rule, and a flow rule. Plastic potential function, f, is adopted as a yield function which is treated a function of not only stress but also work hardening and temperature.

$$f=f(\{\sigma\},W_p,T) \tag{7}$$

where the parameter W_p is associated with a hardening rule and the function f represents the isotropic hardening.

In the finite element formulation, after discretization of the problem domain into finite elements, the displacement components of a point in an element $\{\Delta u\}$ and the incremental strain $\{\Delta\varepsilon\}$ are interpolated by incremental nodal displacement $\{\Delta d\}$

$$\{\Delta u\}=[N]\{\Delta d\} \quad and \quad \{\Delta\varepsilon\}=[B]\{\Delta d\} \tag{8}$$

where *[N]* and *[B]* are the appropriate functions and matrices.

On the other hand, the incremental potential energy is defined by $\Delta\pi = \Delta U+\Delta W$. However, in pure thermal- elasto-plastic analysis with no external work, ΔW is vanished. By applying the minimum potential theorem for $\Delta\pi = \Delta U$, the following must be met for equilibrium.

$$\frac{\partial(\Delta\pi)}{\partial(\Delta d)}=\left(\int_v [B]^T[D_{ep}][B]dV\right)\{\Delta d\}-\int_v [B]^T[D_{ep}]\left(\begin{array}{l}\{\alpha\}dT+\{d\varepsilon_c\}\\ +\dfrac{[D_{ep}]^{-1}[D_e]\{\sigma\}}{S}\dfrac{\partial f}{\partial T}\Delta T\end{array}\right)dV=0 \tag{9}$$

Eq.(9) may be rearranged into the finite element equations at an element level as

$$[K_e]\{\Delta d\}=\{\Delta F\} \tag{10}$$

where

$$[K_e]=\int_v [B]^T[D_{ep}][B]dV \tag{11}$$

$$\{\Delta F\}=\int_v [B]^T[D_{ep}]\left\{\begin{array}{l}\{\alpha\}dT+\{d\varepsilon_c\}\\ +\dfrac{[D_{ep}]^{-1}[D_e]\{\sigma\}}{S}\dfrac{\partial f}{\partial T}\Delta T\end{array}\right\}dV + \int_{v_e}[N]^T b dv \tag{12}$$

The equilibrium equations for a whole domain are obtained by assembling over *N* elements.

$$[K]\{u\}=\{\Delta R\} \tag{13}$$

where *[K]* is the global stiffness matrix and $\{\Delta R\}$ the global incremental thermal force vector as

$$[K]=\sum^{N}[K_e] \tag{14}$$

$$\{\Delta R\}=\sum^{N}[\Delta F] \tag{15}$$

4. Verification and application

4.1. Verification

In FDM for temperature analysis in this study, Austenite to Pearlite transformation through the diffusive type was considered by Avrami [15]. To verify our numerical procedure, simulation of a simple model, shown in Fig. 1, is carried out. And the result of our analysis is compared with that obtained from the analytic and commercial package ANSYS.

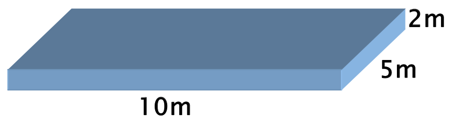

Fig. 1 Model and dimension

The dimensions of the Al-alloy plate used in the simulation is 10 X 5 X 2 m . The initial temperature of the plate is 500 °C and the temperature of the surrounding is also 500 °C. The mechanical properties of the Al-alloy is shown in Table 2.

Table 2.
Mechanical Properties of Al-alloy

Material		Aluminum
Young's modulus	20 °C	70 GPa
	500 °C	20 GPa
Density	20 °C	2600 kg/m3
	500 °C	2300 kg/m3

The type of element is hexahedron and uses various number of elements for examination of the accuracy. Figure 2 shows the deformed figure and Table 3 is the results of several test case.

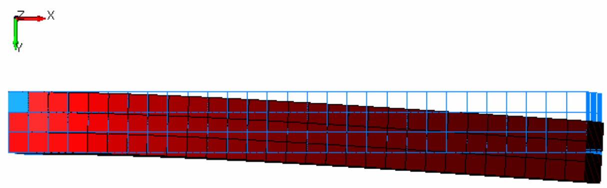

Fig. 2 Deformed shape of the plate model (500°C)

Table 3.
Results of the plate model, unit [mm]

100 Elements	1218 Elements	ANSYS	Analytic
13.2	15.3	15.0	16.9

Figure 3 is the model for industrial application. The model is 1mm long , 56mm wide and 235 kg in weight. The mechanical properties are same with the above plate model. The half part is considered due to geometric symmetry. The boundary condition is simply supported condition.

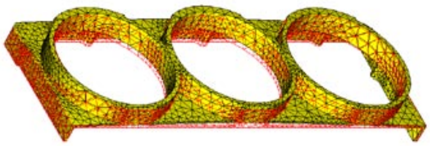

Fig. 3 Finite element mesh model

Figure 4 is the deformed shape of the above model. Point P indicates the maximum deflection zone and the maximum deflection is 2.7mm.

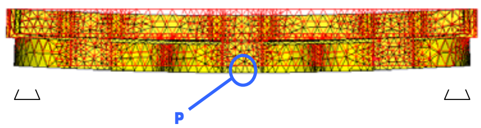

Fig. 4 Estimated deformation at 500 .

We considered the effect of weight at high temperatures. But high temperature which is faced in real process, especially solution treatment, induces creep deflection also. The creep equation and constants are used Eq. (4) and aluminum 7075 in Table 1. And the sustain time is assumed to be 5 hours. The results of with creep and without creep are summarized in Table 4.

Table 4.
Summary of results of deflection considering creep phenomena

	Weight	Weight + Creep
Plate model	1.53×10^{-2}	2.18×10^{-2}
Real product	2.72×10^{-2}	4.01×10^{-2}

5. Conclusions

Although the thermal gradient is small but the heating induces large deflection in heat treatment process due to the low rigidity and creep phenomena at high temperature. In this study, we developed the numerical program calculating the deflection induced due to its own weight and the creep. The code is verified by comparing with the analytic solution of a simple plate model. Also, we examined the deformation appearance for real product. The calculation of the initial deformation at heating step induces the exact prediction on the succession process for example, quenching. Finally, the above procedure is applied to annealing and/or tempering process.

Acknowledgements

This work was supported by Incheon Automotive Components Industry RIS.

References

[1] T. Inoue, D. Y. Ju, K. Arimoto, 1992, “Metallo-Mechanical Simulation of Quenching Process-Theory and Implementation of Computer Code ‘Hearts’ ”, Proc. of 1st Int. Conf. on Quenching & control of Distortion.,pp. 205-212.

[2] J. M. Berghheur , J. B. Leblond, 1990, “Coupling Between Heat Flow,Metallurgy and Stress-Strain Computations in Steels: The Aproach Developed in the Computer Code SYSWELD for Welding or Quenching,” Proc. Of the Fifth Int. Conf. On Modeling of Casting,Welding and Advanced Solidification Processes-V, pp. 203-209.

[3] B. G. Thomas, 1993, “Stress Modeling of Casting Processses: An Overview”, International Conference on Modeling of Casting & Solidification Process –VI, pp. 519-534.

[4] O.-S. Kim, B.-K. Gu, 1994, “FEA of Elasto-Plastic Thermal Stress in Quenching with Considering Phase Transformation (I)”, Korean Soc. of Heat Treatment Trans., Vol. 7, No. 4, pp.233-243.

[5] O.-S. Kim, K.-H. Song, B.-K. Gu, 1995, “FEA of Elasto-Plastic Thermal Stress in Quenching with Considering Phase Transformation (II)”, Korean Soc. of Heat Treatment Trans., Vol. 8, No.1, pp.12-23.

[6] T. Inoue, S. Nakaki, T. Kishino, M. Monkawa, 1981, “Description of Transformation Kinetics, Heat Conduction and Elastic-Plastic Stress in the Course of Quenching and Temperaturing of Some Steels,” Ingenieur-Archive, Vol. 50,pp.315-327.

[7] S. Denis, S. Sjostrom, and A. Simon, 1987, “Coupled Temperature, Stress, Phase Transformation Calculation Model Numerical Illustration of the Internal Stresses Evolution during Cooling of a Eutectoid Carbon Steel Cylinder,” Metall. Trans. A, Vol.18A, pp. 1203-212.

[8] J. Rohde, A. Jeppsson, 2000, “Literature Review of Heat Treatment Simulations with Respect to Phase Transformation, Residual Stresses and Distortion,” Scandinavian Journal of Metallurgy, Vol. 29(2), pp. 47-62.

[9] T. Reti, M. Gergely, and P. Tardy, 1987, “Computing method for non-isothermal heat treatments,” Materials Science and Technology, Vol. 3, pp 365-371.

[10] Grill. A., Brimacombe, J. K. and Weinberg, F., 1976, "Mathematical Analysis of Stresses in Continuous Casting of Steel ," Ironmaking and Steelmaking, No. 1, pp. 38-47.

[11] T. X. Hou , R. D. Pehike, J. O. Wilkes, 1990, “Computer Simulation of Casting Solidification Using a Combination of the Finite Element and Boundary Element Methods”, Int. Conf. on Modeling of Casting & Solidification Process –V, pp.15-22.

[12] N. E. Dowling, Mechanical Behavior of Materials, Prentice Hall, 1999.

[13] H. M. Si, C. Cho, and S. Y. Kwak, 2003, “A Hybrid Method for Casting Process Simulation by Combining FDM and FEM with an Efficient Data Conversion Algorithm,” JMPT, Vol. 133, pp.311-321.

[14] T. R. HSU, 1986, The Finite Element Method in Thermo-mechanics, Allen&Unwin, Inc.

[15] M. Avrami, 1940, “Kinectics of Phase Change II”, J.Chem. Phys., Vol. 8, p.212-218.