FINITE ELEMENT THERMAL - STRUCTURAL ANALYSIS OF HEATED PRODUCTS

Pramote Dechaumphai and Wiroj Lim

Chulalongkorn University
Bangkok 10330 , Thailand

Abstract

Finite element analysis procedures for predicting temperature response and associated deformation including thermal stresses of heated products are presented. Finite element computer programs that can be used on standard personal computers have been developed. The capabilities of the finite element method and the computer programs are evaluated by the examples of: (1) heat transfer in amplifier fins, and (2) thermal stress in an engine piston. Results from these examples demonstrate the efficiency of the method for the analysis of heated products that have complex geometries.

1. Introduction

The manufacturing industry in Thailand has grown up rapidly during the past few years. The industry has the capability to manufacture more complex advanced products that could not be done a decade ago. These include new styles of sanitary products, large size plastic products, automotive engines and accessories, aluminum alloy wheels with complex shapes for sport cars, etc. The styles and shapes for almost all of these products, however, have been designed abroad. Several Thai manufacturers have attempted to design new products with different shapes suitable for the needs of the Thai community and the taste of Thai people. But several new designs have failed due to the lack of an analysis capability for predicting and assuring the designers that the new designs will work. For example, sanitary products and aluminum alloy wheels that have complex geometries may crack during the cooling process or fail during operation. Such failure wastes a large amount of investment, time, and manpower. More importantly, the failure creates fears for the designers which impedes further development of new products with different design shapes.

To avoid such failures and reduce the cost of production, the behavior that occurs in the products during manufacturing or while operating must be predicted in advance. These behaviors include the stresses from external loads as well as the thermal stresses from the temperature difference in the products. The purpose of this paper is to present a finite element method that could be used to analyze heat transfer problems that have complex geometries for the temperature distribution. The predicted temperature combined with the applied external loads are then used to compute the deformation and thermal stresses of the products.

The heat transfer analysis using the finite element method will be presented first. The thermal stress analysis and the corresponding finite element formulation are then described. Finite element computer programs for the two analyses, that can be used on standard personal computers, have been developed. The finite element formulations and these computer programs are evaluated by solving the problems of: (1) heat transfer in amplifier fins, and (2) heat transfer and thermal stress in an engine piston.

2. Heat Transfer Analysis

2.1 Differential Equation

Transient heat conduction in three-dimensional body is governed by the differential equation in the form [1]

$$-\left(\frac{\partial q_x}{\partial x}+\frac{\partial q_y}{\partial y}+\frac{\partial q_z}{\partial z}\right)+Q = \rho c \frac{\partial T}{\partial t} \quad (1)$$

where q_x, q_y, q_z are the conduction heat fluxes in the x, y, z coordinate directions, respectively; Q is the internal heat generation rate per unit volume; ρ is the mass density; c is the specific heat; and T is the temperature that varies with the coordinates as well as the time t. The conduction heat fluxes can be written in the form of temperature using Fourier's law. For isotropic materials, the relations are

$$q_x = -k\frac{\partial T}{\partial x}$$
$$q_y = -k\frac{\partial T}{\partial y} \quad (2)$$
$$q_z = -k\frac{\partial T}{\partial z}$$

where k is the material thermal conductivity.

2.2 Boundary and Initial Conditions

Heat transfer boundary conditions consist of several heat transfer modes that can be written in different forms. The boundary conditions frequently encountered are as follows :

(a) Specified temperature on surface S_1,

$$T_s = T_1(x, y, z, t) \quad (3a)$$

(b) Specified heat flux on surface S_2,

$$q_x n_x + q_y n_y + q_z n_z = -q_s \quad (3b)$$

(c) Convection heat transfer on surface S_3,

$$q_x n_x + q_y n_y + q_z n_z = h(T_s - T_\infty) \quad (3c)$$

where T_1 is the specified surface temperature; q_s is the specified surface heat flux (positive into surface); n_x, n_y and n_z are the direction cosines of the outward normal vector to the surface; h is the convective heat transfer coefficient; T_s is the unknown surface temperature, and T_∞ is the convective exchange temperature.

The initial condition associated with the differential equation (1) is given by

$$T(x, y, z, 0) = T_i(x, y, z) \quad (4)$$

2.3 Finite Element Formulation

Finite element equations corresponding to the governing differential equation (1) and the boundary conditions, Eqs. (3a)-(3c), can be derived using the method of weighted residuals [2]. The element temperature distribution is first written in the form

$$T(x, y, z, t) = [N(x, y, z)]\{T(t)\} \quad (5)$$

where $[N]$ is the element interpolation function matrix and $\{T\}$ is the element nodal temperatures. The vector of the temperature gradients can then be written in the form

$$\begin{Bmatrix} \partial T/\partial x \\ \partial T/\partial y \\ \partial T/\partial z \end{Bmatrix} = [B(x, y, z)]\{T(t)\} \quad (6)$$

where $[B]$ is the temperature gradient interpolation matrix.

The application of the method of weighted residuals on the governing differential equation (1) using the boundary conditions, Eqs. (3a)-(3 c), leads to the finite element equations in the form

$$[C]\{\dot{T}\} + \left[[K_c] + [K_h]\right]\{T\}$$
$$= \{Q_c\} + \{Q_Q\} + \{Q_q\} + \{Q_h\} \quad (7)$$

where the element matrices are defined by the integrals over the element volume V or surface area A as follows:

$$[C] = \int_V \rho c \{N\}[N]\, dV \quad (8a)$$

$$[K_c] = \int_V [B]^T k [B]\, dV \quad (8b)$$

$$[K_h] = \int_A h\{N\}[N]\, dA \tag{8c}$$

$$\{Q_c\} = -\int_A (\bar{q} \cdot \hat{n})\{N\}\, dA \tag{8d}$$

$$\{Q_Q\} = \int_V Q\{N\}\, dV \tag{8e}$$

$$\{Q_q\} = \int_A q_s\{N\}\, dA \tag{8f}$$

$$\{Q_h\} = \int_A h\, T_\infty\{N\}\, dA \tag{8g}$$

where $[C]$ is the capacitance matrix, $[K_c]$ is the conduction matrix, $[K_h]$ is the surface convection matrix, $\{Q_c\}$ is the conduction load vector, $\bar{q}$ is the conduction heat flux vector, n is the unit normal vector to the surface, $\{Q_Q\}$ is the load vector associated with internal heat generation, $\{Q_q\}$ is the load vector from specified surface heating, and $\{Q_h\}$ is the load vector associated with surface convection. These element matrices can be evaluated in closed-form for some element types. As an example, for the four-node tetrahedral element as shown in Fig. 1(a), the capacitance matrix is

$$[C] = \frac{\rho c V}{20}\begin{bmatrix} 2 & 1 & 1 & 1 \\ 1 & 2 & 1 & 1 \\ 1 & 1 & 2 & 1 \\ 1 & 1 & 1 & 2 \end{bmatrix} \tag{9}$$

and the convection matrix is

$$[K_h] = \frac{hA}{12}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 2 & 1 & 1 \\ 0 & 1 & 2 & 1 \\ 0 & 1 & 1 & 2 \end{bmatrix} \tag{10}$$

where A is the surface area of the plane that consists of node numbers 2, 3 and 4.

3. Structural Analysis

3.1 Differential Equations

The equilibrium condition for a three-dimensional elastic body is governed by the differential equations in the form [3]

$$\begin{aligned} \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + F_x &= 0 \\ \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + F_y &= 0 \\ \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + F_z &= 0 \end{aligned} \tag{11}$$

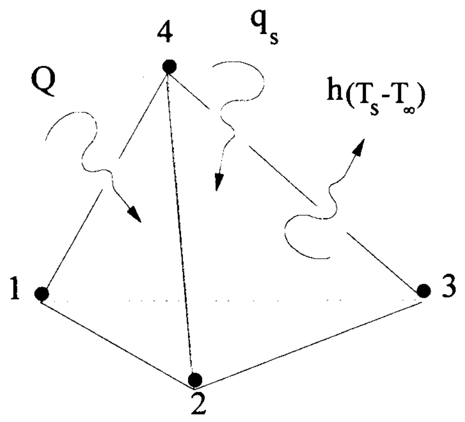
where σ_x, σ_y, σ_z are the normal stresses in the x, y, z coordinate directions, respectively; τ_{xy}, τ_{xz}, τ_{yz} are the shearing stresses; and F_x, F_y, F_z are the body forces in the x, y, z coordinate directions, respectively. The stress components can be written in form of the strain components using the generalized Hooke's Law as

$$\{\sigma\} = [D]\{\varepsilon - \varepsilon_0\} \tag{12}$$

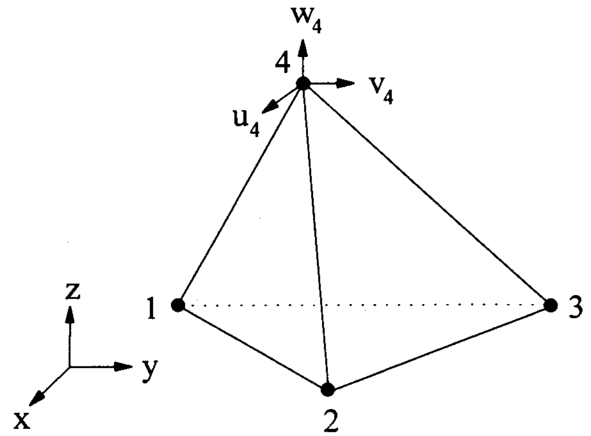
where $[D]$ is the elasticity matrix [3] and

$$\{\sigma\}^T = [\sigma_x\ \sigma_y\ \sigma_z\ \tau_{xy}\ \tau_{yz}\ \tau_{xz}] \tag{13}$$

$$\{\varepsilon\}^T = [\varepsilon_x\ \varepsilon_y\ \varepsilon_z\ \gamma_{xy}\ \gamma_{yz}\ \gamma_{xz}] \tag{14}$$



(a) Thermal analysis



(b) Structural analysis

Figure 1. Four-node tetrahedral finite element for thermal and structural analyses.

$$\{F_B\} = \int_V [N]^T \{F\} \, dV \qquad (23c)$$

$$\{F_t\} = \int_A [N]^T \{S\} \, dA \qquad (23d)$$

where $[K]$ is the stiffness matrix, $\{F_0\}$ is the load vector associated with the temperature change, $\{F_B\}$ is the load vector from the body forces, and $\{F_t\}$ is the load vector from the specified surface tractions.

4. Applications

Two applications are presented to demonstrate the capability of the finite element formulation and the computer programs developed. The first application is the thermal analysis for predicting temperature distribution in three-dimensional amplifier fins. The second application is the thermal and structural analyses for predicting temperature and thermal stress distributions in an engine piston. These results were obtained from the computer programs that have been developed and operated on standard personal computers. The computer programs have been verified by solving several academic-type problems that have exact solutions before being applied to solve these problems.

4.1 Heat Transfer in Amplifier Fins

To demonstrate the capability of the finite element thermal analysis program developed, a sample problem of a three-dimensional amplifier fin made of aluminum as shown in Fig. 2 was selected. The fins are used to dissipate heat generated by transistors located beneath the amplifier fins with the position shown in the figure to the surrounding air. For the purpose of demonstrating the analysis capability, the temperature at this position is assumed to be 125^{o}C. Along the surfaces of all the fins, convection heat transfer occurs to the surrounding air that has a temperature of 30^{o}C.

With the geometry and dimensions of the amplifier fins shown in the figure, a finite element model consisting of 2,017 tetrahedral elements and 664 nodes was created as shown in Fig. 3. With the boundary conditions of specified temperature at the position mentioned and convection heat transfer on all fin surfaces, the finite element program was used to solve for the temperature solution. The predicted temperature distribution is shown in Fig. 4 using contour lines. The figure shows high temperature at the base of the model near the transistor location. Away from this location, the temperature decreases because of convection heat transfer from the fins.

The result of the temperature distribution obtained from this example not only provides a solution to the problem but also suggests ways to improve the amplifier model. For example, less material may be used or some sections far away from the base of the model can be cut out because they provide little convection heat transfer. With the help of such analysis capability, designers can create a model that can provide the highest efficiency before manufacturing, and at the same time, avoid the trial-and error process that is commonly used in the manufacturing industry in Thailand today.

4.2 Thermal Stress in an Engine Piston

Thermal-structural analysis of an engine piston is another example that was selected to demonstrate the analysis capability because the piston has complex geometry and may be subjected to complex boundary conditions. The engine piston geometry and boundary conditions are shown in Fig. 5. The piston is subjected to heating as well as pressure from combustion on the top surface. Both the heating and pressure are assumed to be uniform as shown in Fig. 5(a). Along the inner surface, convection heat transfer occurs as shown in Fig. 5(b). All the data used in the analysis were taken from Ref. [4], however, the main objective of the analysis still concentrates on demonstrating the capability of the thermal-structural analysis programs developed.

Due to symmetry of the problem, only a quarter of the engine piston was used in the analysis. A finite element model was created as shown in Fig. 6. The model consists of 3,066 tetrahedral elements and 945 nodes. For the thermal analysis, each node has only one unknown that of temperature. Thus, there are 945 unknowns to be solved in the thermal analysis. In the structural analysis, however,

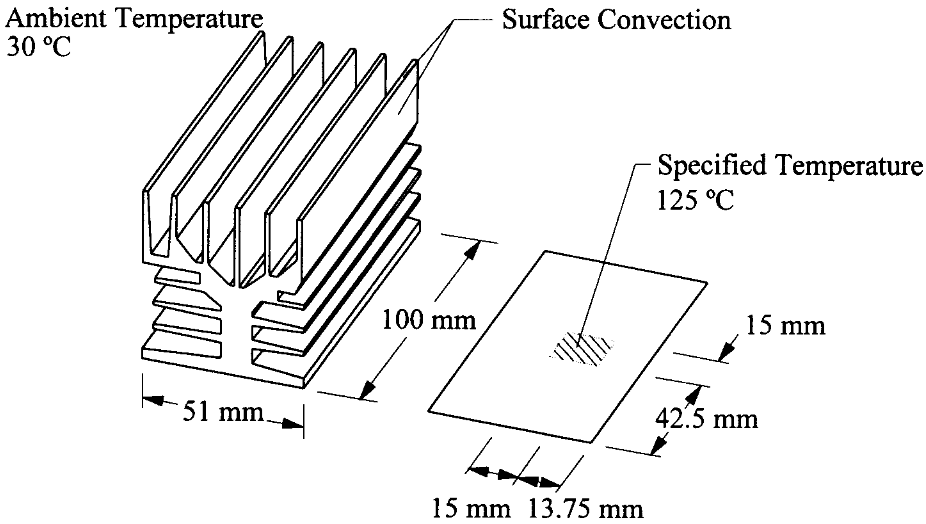


Figure 2. Geometry and boundary conditions of amplifier fins.

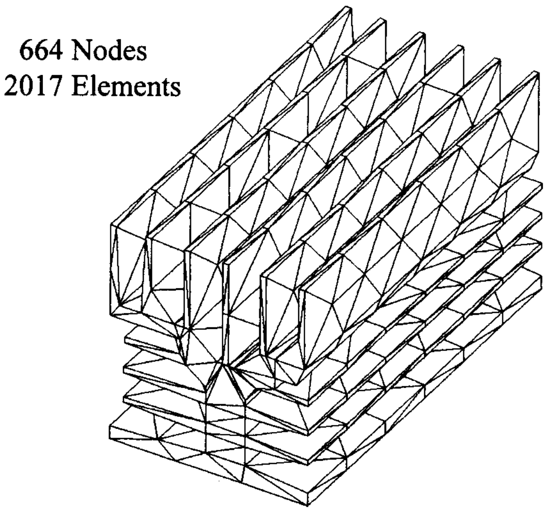


Figure 3. Finite element model of the amplifier fins.

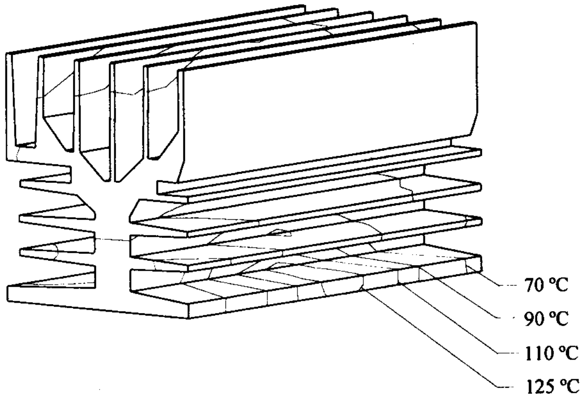


Figure 4. Predicted temperature distribution of the amplifier fins.

each node has three unknowns which are the displacement components. Thus there are 2,835 unknowns to be solved in the structural analysis

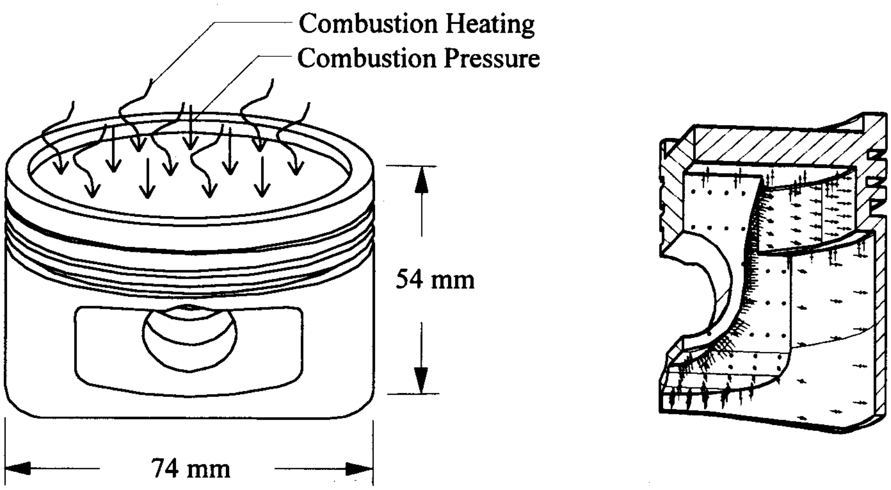
The predicted temperature distribution in the form of contour lines is shown in Figure 7. The solution indicates high temperature and temperature gradient near the top surface of the piston due to large solid section in that region. The predicted temperature distribution from the thermal analysis was then used as the input to the structural analysis with the inclusion of the specified combustion pressure. The same finite element model was used in the structural analysis. Thus the nodal temperatures obtained from the thermal analysis can be transferred directly to the structural analysis. The structural analysis was then performed. The predicted Von Mises stress distribution in the form of contour lines superimposed on the deformed shape is shown in Figure 8. The figure indicates that high stresses occur in the region of high temperature gradient and near the center region of the top surface due to the combustion pressure

For problems with complex geometry and boundary conditions such as this example, additional analyses with refined finite element models should be performed to insure the solution behavior and convergence. The predicted solutions presented herein, however, demonstrate the finite element thermal-structural analysis capability for problems with complex geometries. These results, in addition, are obtained using computer programs developed that can be used on standard personal computers.

5. Conclusion Remarks

Thermal and structural analyses of heated products using the finite element method were presented. The analyses predict temperature distribution and deformation including thermal stress for complex three-dimensional bodies. The finite element formulation was described and the associated finite element equations were derived for both analyses. The corresponding finite element computer programs have been developed. These computer programs can be used on standard personal computers. The programs have been verified by a number of academic-type problems that have analytical solutions prior applying to solve more complex problems.

To demonstrate the capability of the finite element method and the computer programs developed, two example problems were analyzed. The first example problem represents



(a) Heating and pressure loads (b) Convection surface

Figure 5. Geometry and boundary conditions of the engine piston.

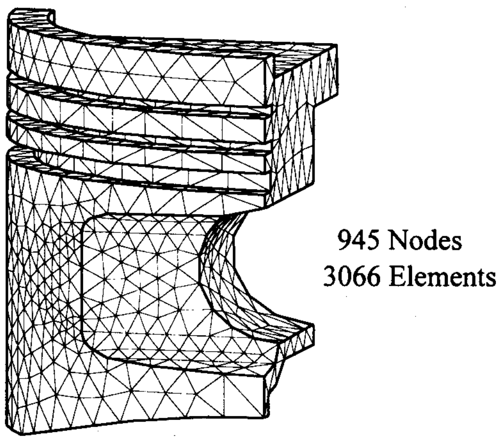


Figure 6. Finite element model of the engine piston.

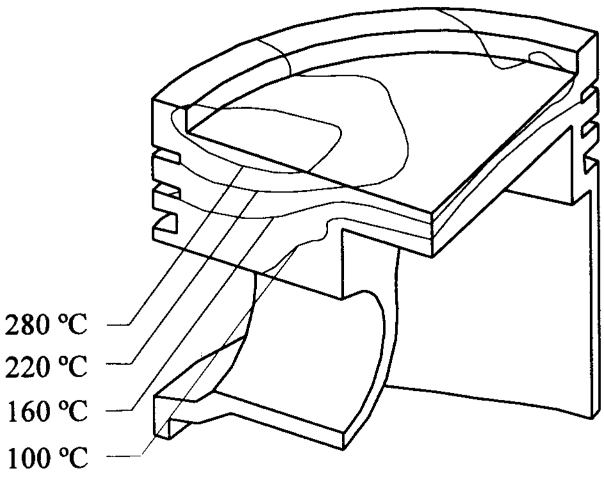


Figure 7. Predicted temperature distribution of the engine piston.

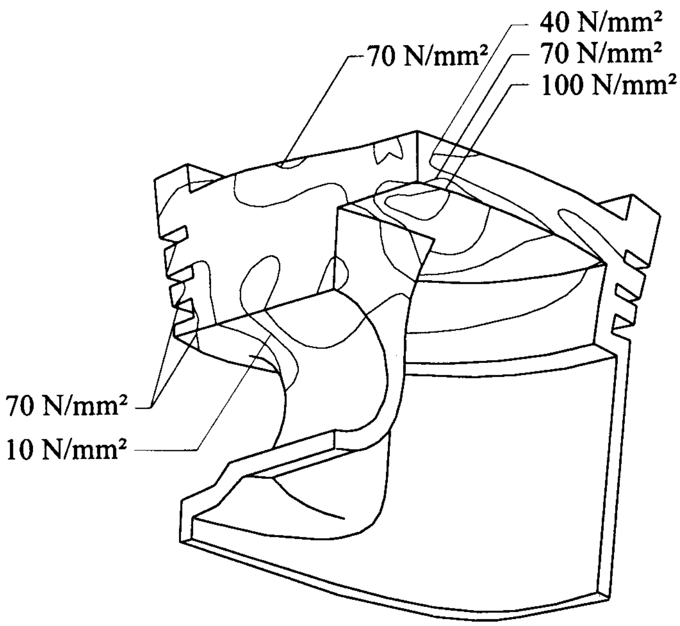


Figure 8. Predicted Von Mises stress distribution superimposed on the deformed geometry of the engine piston.

the determination of temperature distribution in amplifier fins that dissipate heat generated by amplifier transistors to the surrounding air by convection heat transfer along the fin surfaces. The computed temperature solution can aid designers to arrive at a model that has the highest efficiency in dissipating heat as well as using less material. The second example problem represents the thermal and structural analyses of an engine piston. In addition to the solutions of the temperature and thermal stress response obtained, this second example highlights the ease of using a single finite element method with common model discretization to carry out both analyses. Such procedure is useful especially for thermal stress problems that have complex geometries.

The finite element analysis procedures and the corresponding computer programs developed can generate information that helps decision making and reduce the trial and error processes commonly used in manufacturing new products in Thailand.

6. Acknowledgement

The authors are pleased to acknowledge the National Metal and Materials Technology Center of the National Science and Technology Development Agency under the Ministry of Science Technology and Environment, Bangkok, Thailand, for supporting this research work.

7. References

[1] Holman, J. P. 1992. Heat Transfer. 7 ed. McGraw-Hill, New York.

[2] Dechaumphai, P. 1994. Finite Element Method in Engineering. Chulalongkorn University Press, Bangkok.

[3] Ugural, A. C. and Fenster, S. K. 1975. Advanced Strength and Applied Elasticity. Elsevier North-Holland, New York.

[4] Crouse, W. H. and Anglin, D. L. 1995. Automotive Engines. 8 ed. McGraw-Hill, Singapore.