7

Development of Heat Conduction Code THAP and its Application to the Fastener Jointed Component Analysis

Kunihko OHTAKE
Structural Mechanics Division
National Aerospae Laboratoty
Tokyo, Japan

Mitsuko KAMOHARA and Hiroko INOUE
Marine and River Technology
Tokyo, Japan

Abstract

We are developing the finite element heat conduction analysis code "THAP". The main purpose of this development is the numerical simulation of the hypersonic aeroframe heat conduction. Special attention is paid to the joint structure thermal conductivity. The code can treat transient nonlinear heat conduction with temperature dependent material property, radiative and convective thermal load and contact thermal resistance. Using the code, we have examined the parametric identification of a heat conduction model for a fastener jointed structural component.

1. Introduction:

Recently in Japan, technology development for a hypersonic transportation system has become a very important engineering subject. There, thermal protection of aeroframes from the extremely high heat flux caused by aerodynamic heating is one of the most important design problems. It is thought that computer simulation will play an important role in the up-to-date efficient design of a heat resistant aeroframe . For the heat conduction analysis of solid configuration, Nickell[1)] first developped the finite element formulation with the aid of Gurtin's variational principle[2)]. Afterwards Oden[3)] developped the general finite element formulation method based on the weak or Galerkin form variational principle. Thereafter it seems that, because of the development of the FEM code, heat conduction analysis is only a problem of computer time, even for very complex configuraton. But the f acts show that the situation is not so simple for aeroframe heat conduction simulation. There are mainly three trouble sources which disturb the required precise FEM heat conduction simulation. First is the uncertain material properties which is enevitable from a heat conductivity mechanism. Second is the uncertain heat input and output. The aeroframe s tructure is composed of thin plates and shells, which means that the surface dimension is much larger than the thickness dimension. Therefore heat flow through the surface has a great effect on the solid inside heat flow. In other words, the boundary condition is uncertain. Third is that the usual aeroframe contains many joints, which cause the uncertinty of the numerical simulation model. In order to overcome these difficulties and to supply sufficient data for design request, development of modelling or system identification technique for heat conduction simulation is important. In addition we need a compact and high performance FEM code which can treat the complex modelling requests.

2. The feature of our heat conduction analysis code THAP [4)]

We developed a new heat conduction code, THAP (Temperature and Heat conduction Analysis Program), in order to cope with the above situation. The emphasis is on the treatment of boundary heat transfer including contact thermal resistance. In addition THAP can treat coupling analysis with aerodynamic heating code FIVAD and radiative heat transfer code

RHT95 .

2.1 Program Organization

The program is devided into five phases. The total program structure is shown in Fig. 1

1) Input control information, mesh and element data time functions.

2) The system conductivity and heat capacity matrices are assembled in the compact skyline form [5]

3) Calculation of Applied heat flow vectors

According to the time dependent load history function data, the program calculates the nodal heat flow vector. Edge(2D) or face(3D) distributed loads are converted to the nodal load data.

4) Step by step solution

Central difference time integration and the Forward Euler method is available [8].

5) Nonlinear equiliblium iteration

If the material properties are temperature dependent, incremental equiliblium iteration can be available [4].

2.2 Element Liblary

1) 2 node 1D element
2) 3 and 4 node 2D element
3) 3 and 4 node axisymmetic element
4) 4 node nonplaner surface element used for 3D conection or fin element
5) 6 and 8 node 3D element
6) 2 node 3D contact thermal resistance element

Every element can be available for temperature dependent material property.

2.3 Boundary condition

1) Applied heat flux, concentrated and distributed
2) Convection boundary, disrtibuted
3) Radiative boundary, concentrated and distributed
4) Fixed temperature boundary

2.4 Contact thermal resistance element [4]

Our idea of a contact element is the extension of the 1-d heat conduction element. Let's define contact thermal resistance (inverse of conductance) R by $Q = A/R*(T^{+}-T^{-})$; where Q:total heat flux, A:contact surface area, T^{+} and T^{-} :contact surfacetemperature of each side, respectively [6]. Comparison of this equation to the Fourier's law of heat conduction leads us to the 2 x 2 heat conductance matrix of K = {kij}, where k11 and k22 = A/R, k12 and k21 = -A/R. This is regarded as a limit of the 1-d usual conductance element, where element length is zero. We add that the quantity R can be measured experimentaly. In Fig. 2 we demonstrate the contact element ability. Two rods, rod1 and rod2, are connected to each other. The length of rods are L1=0.1m, L2=0.05m. Thermal conductivity: k1=2000W/mK, k2=1000W/mK. Specific heat: c1=c2=500J/m^3K. Contact thermal resistance:1/R= 20,000W/m^2K. Contact area: A=0.01m^2. From one end heat flux q= 2000W/m^2 is given and the other end is kept to a constant initial temperature of 300K . After 100 seconds, temperature distribution reaches the steady state, as shown in the figure. The result coinsides with the theoretical value.

3. Model Optimization [7]

3.1 Parameter Identification method

Once the FEM model or heat conductance matrix system is formed, computer simulation is carried out and compared with some observed results. We set the measure of error as $E = \Sigma W*(\theta e - \theta c)**2$, where W is weight, θe is experimental temperature, θc is calculated temperature and summation is taken over all interested points. We want to minimize E by changing the matrix system and want to get better fitting between observation and simulation . The choice of the parameters or design variables is most important in the optimization procedure. Our general policy is to minimize the number of design variables. Usually the dimension of the FEM matrix system is large. We choose the physical thermal coefficients as the design variable base. Quite often coefficients are temperature dependent and it seems inevitable that the number of variables is of the same order as the number of elements (or even a multiple of of the element number). This is unfortunate. We don't use the coeficients directly as the design variables, but choose the multiplier of the temperature dependent coefficient function in such a case. Therefore we prepare only one variable for each material's particuler coefficient. In another case we set some special relation among the different material's coefficients and make it into one group. Then only one parameter is necessary to the related coefficients. In seeking the minimum of object functional E, we use gradient information of θ with respect to the chosen variables, and apply the information to the necessary condition of minima, which says that partial differential of E with respect to design variable should be zero. This leads us to the linear system, the solution of which tells us the minimum direction. The iteration procedure leads us to the mimina.

3.2 Example Demonstration of the Method

We apply the above method to a two piece fastener jointed structure. Fig. 3 shows the 2-d FEM model of this test specimen. Total height is 150mm, width is 280mm, contact surface length is 20mm, and the upper 50mm is electrically heated. Temperature measurement results, partially interpolated to coinside with the FEM node points, are compared with THAP simulation results. Then parameter identification iterations are carried out. Several different approaches are tested and convergence is obtained. Figs. 4 and 5 show a steady state example, where the thermal conductivity k, contact thermal resistance R and surface emisivity em1 and em2 is chosen as the variable base. Here em1 and em2 are related linearly. Therefore the number of design variables is three in this case. Three iterations are enough for this case. Initial and final optimised values of thermal properties, calculated from the design parameters, are tabulated in Table 1.

References:

1) Gurtin, M. E: Variational Principles for linear initial-value Problems, Q. Appl. Math. 22, pp. 252-256 (1964)
2) Wilson, E. l> and Nickell, R. T. : Application of the finite element method to the heat conduction analysis, Nucl. Engng. Design, 4 pp. 276-286 (1966)
3) Oden, J. T. and Reddy, J. N. : Variational Methods in Theoretical Mechanics, Springer-Verlag (1976)
4) Ohtake, k., Okumura, h., Kamohara, M. and Inoue, H: Heat Conduction Analysis Code THAP. V3 for Joint Structure, NAL J-94004, pp. 139-178, (1994)
5) Bathe, K. J. :Finite Element Proceduresin Engineering Analysis, Prentice-Hall (1982)
6) JSME Databook: Heat Thansfer 4th ed.
7) Fox, R. L: Optimization Methods for Engineering Design, Addison-Wesley (1971)
8) ADINA ENG. : ADINAT User's Manual, Report AE81-2 (1981)

```
main - deffile
     + echinp
     + second
     + error
     + timtable
     + input  - incord
     |        + inptbc  - functn
     |        |         + inload
     |        |         + inload1 - intpltn
     |        |         + inload2 - intpltn
     |        |         + inload3 - intpltn
     |        |         + inradi
     |        |         + inconv2
     |        |         + inconv3
     |        |         - intemp  - intpltn
     |        |
     |        |                 - toned + oned - colht
     |        - inptel  -elemnt + tquad - quad - colht
     |                          |              - funct2
     |                          + thexa - hexa - colht
     |                          |              - funct3
     |                          - tcontr- contr- colht
     + addres
     + assem  - elemnt------------------------- toned - oned - tempel
     |                                        |              + ptable    - serchpara
     |                                        |              + otdm
     |                                        |              - addban
     |                                        |
     |                                        + tquad - quad - tempel
     |                                        |              + ptable    - serchpara
     |                                        |              + qtdm      - consis
     |                                        |              - addban
     |                                        |
     +bload --- cload                         + thexa - hexa - tempel
     |        |                               |              + ptable    - serchpara
     |        + sload -- loadedg              |              + htdm
     |        |        + loadfce - functld    |              - addban
     |        |        |         + loadfsdm   |
     |        |        - addvec               - tcontr- contr- tempel
     |        |                                              + ptable    - serchpara
     |        + hload  - convedg - addban                    + ctdm
     |        |        + convfce - fnctld                    - addban
     |        |        |         + convsdm
     |        |        |         - addban
     |        |        - addvec
     |        - rload
     |
     + makels
     + fixbc
     |
     + colsol
     |
     + result
     |
     + equit  - elemnt -- *
     |        + bload  -- *
     |        + colsol
     |        + result
     |        - discrm - maxtt  - norm
     |
     - writd
```

Fig.1 THAP Program Structure

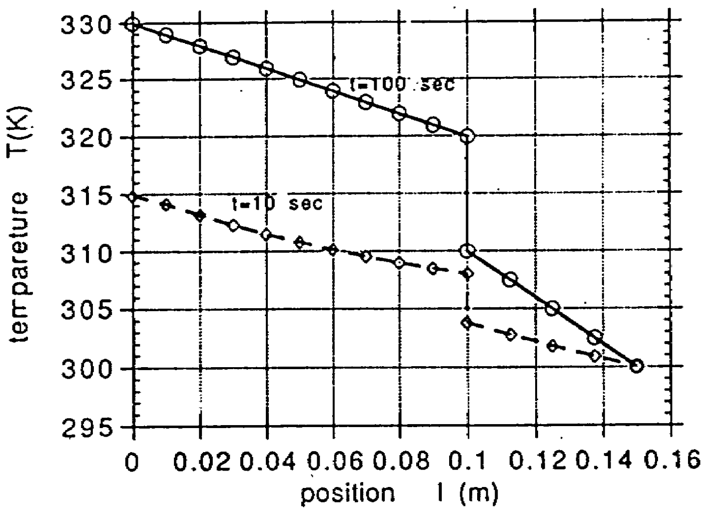

Fig.2 Contact Thermal Resistance Example Simulation

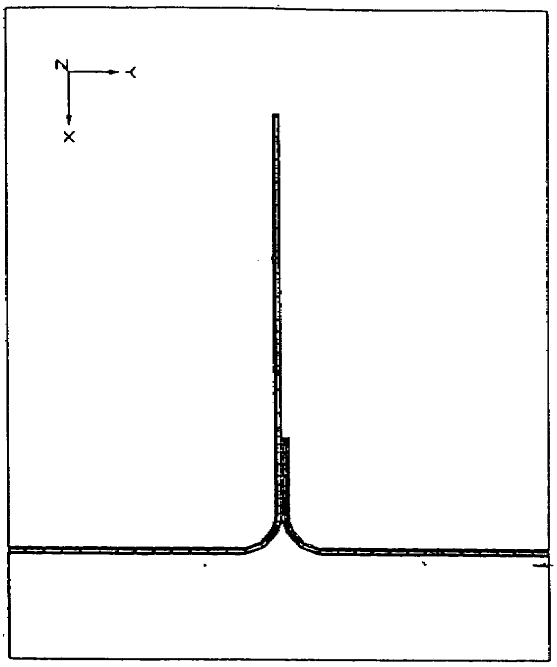

Fig.3 FEM Simulation Model

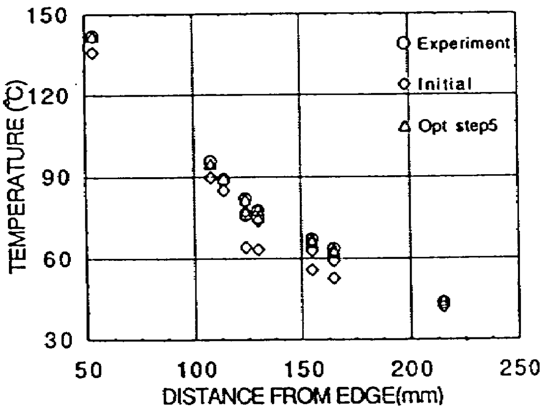

Fig.4 Parametric Identification Demonsrtation Results

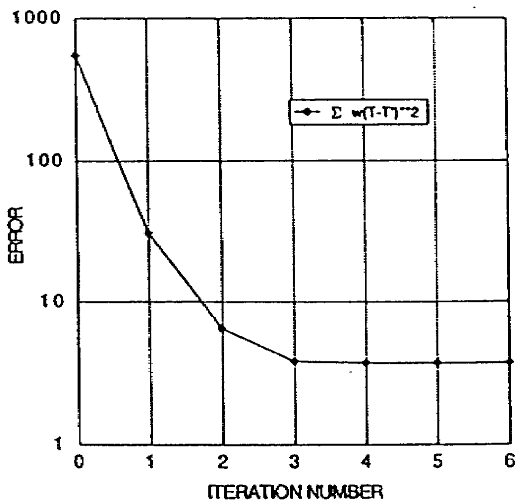

Fig.5 Minimization Error vs. Iteration Number