Calculation of prearcing times using the Finite Element Method

H.Kürschner, A.Ehrhardt, G.Nutsch

Ilmenau University of Technology

Faculty of Electrical Engineering and Information Technique

I.Harrison, A.Boerner, T.Mickley

University of Nottingham

Department of Electrical & Electronic Engineering

Abstract

It is used a commercially available finite element program LUSAS to solve two different problems involving electrical fuses. The first one involves the transient behaviour of the fuse with the ultimate goal being to predict accurately the prearcing time and the I-t-curves of the fuse. The second concentrates on a problem of calculating the temperature of the fuse bodies in different surroundings. In the first case the discretization of the fuse body is very important because close to the fuse element one must use a small mesh size to obtain the temperature of the element accurately. By comparing experimental results with those obtained using the finite element model one can quantify how good the finite element model describes the physical problem. In addition, by minimising the differences between the experimental and theoretical results one can obtain information about the optimum mathematical conditions (i.e. element size and shape, and the time step). Since the aim is to develop a model which is valid for a wide variety of fuse designs the simulations and experiments will be performed over a wide variety of different fuse element designs and material properties.

The second section is concerned with the thermal interaction of the fuse body with its surrounding. This interaction may play a significant role in the operation of the fuse at low overload currents. The temperature of the fuse body will be predict from the simulations and compared with experimental results. These results are important in the commercial use of fuses since they are invariably used in surroundings which are different from those laid down in the appropriate international standards.

1. Introduction

Along with many other factors such as I^2t, a knowledge of the prearcing time of a fuse as a function of current is essential when selecting fuses to protect equipment. The accurate computation of the pre-arcing time would allow a fuse manufacturer to quickly distinguish the effects of geometry changes and reduce necessary tests. This, in turn, will reduce both development time and costs. Moreover, it is widely accepted that a designer of protection equipment must take into account the environment in which the fuse is operating. For example, if a fuse is to be used inside a distribution board along with other fuses then it's current rating must be derated. This derating is necessary because the fuse is operating at elevated temperatures caused by heat generated by the other fuses in close proximity to it. There are several mathematical approaches to modelling the operation of a fuse. These are finite difference [1], Transmission Line Modelling [2], and finite element[3]. Finite difference is relatively simple to program but does not easily lend itself to modelling complex 3 dimensional structures. Transmission Line Modelling is not widely used as a method of solving diffusion problems. This leaves finite elements. It is widely used to simulate complex structures and so was chosen for this work. Since the long term aim of this research program is to transfer the technology from the universities into industry, it was decided to chose a commercial finite element program which had a good graphical input and output interface. Therefore all simulations presented in this paper were performed using the commercial package LUSAS from FEA.

2. Fuse model

For the analysis of the prearcing time the fuse model developed by Schumann [6] was used. It allows the modelling of fuse elements with lengths up to 200 mm with and without restrictions. For the calculations used in this paper fuse elements with a length of 50 and 100 mm were used. A schematic diagram of the fuse is shown in **fig. 1**.

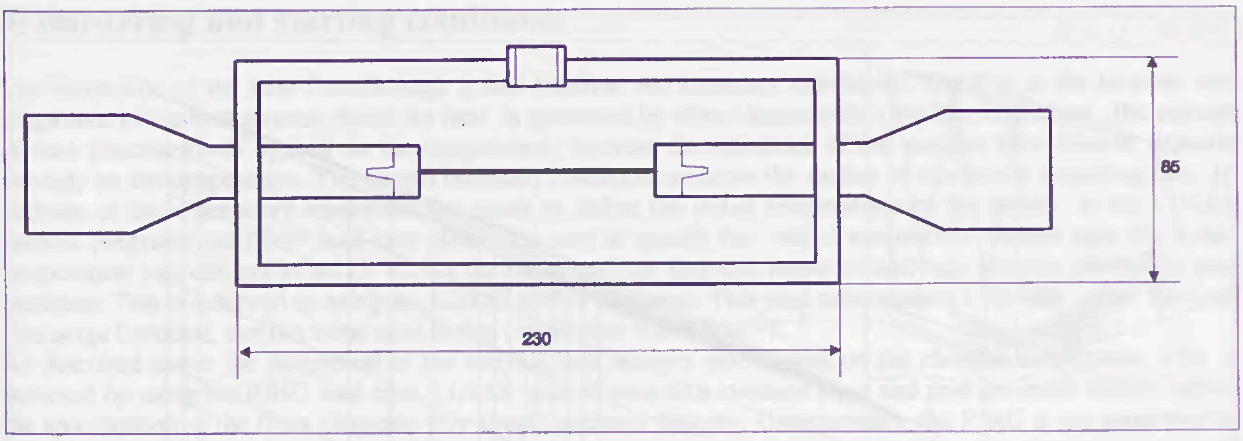

Fig.1: Scheme of the fuse used in the prediction of pre-arcing times.

To study the effects of the surroundings on the operation of the fuse element a different model of fuse was used. Since this is only a preliminary study, the exact modelling of the fuse element was not required and so an approximation to a real fuse was made. A schematic diagram of the fuse is shown in **fig. 2**.

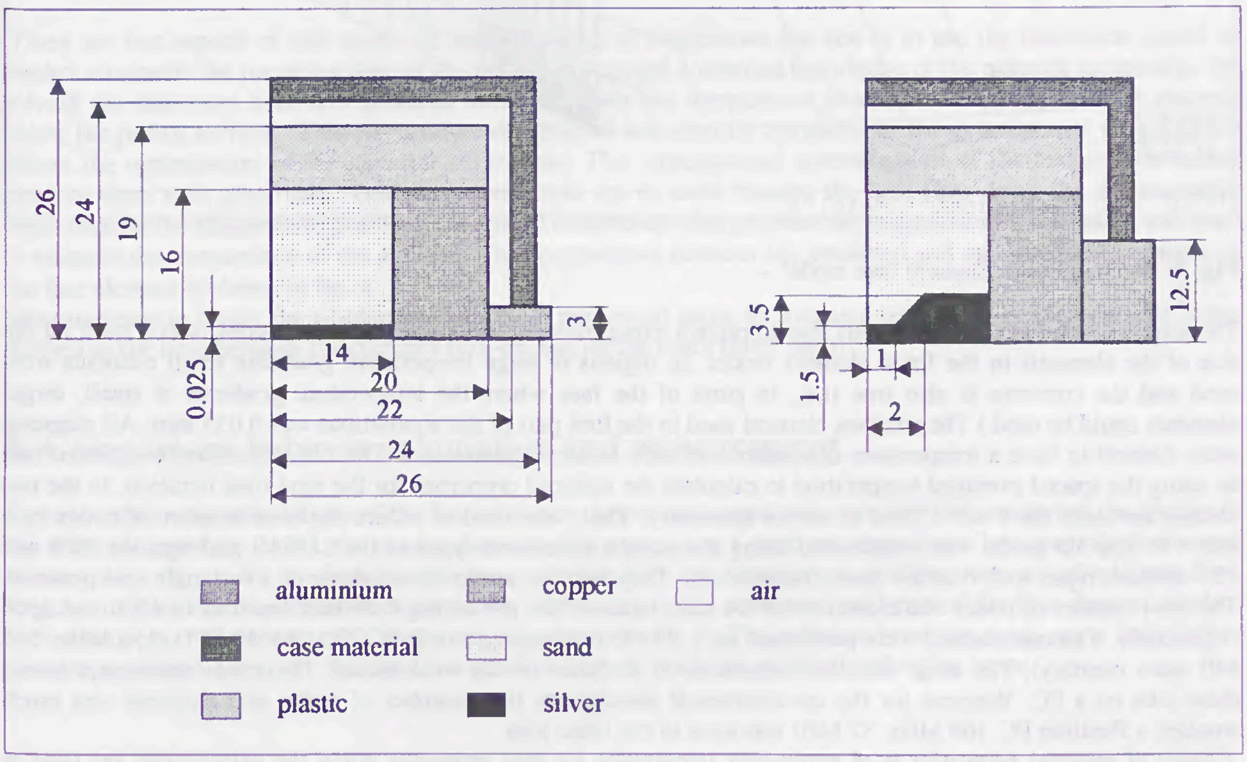

Fig. 2: A schematic diagram of the fuse model used in the study of the effects of the environment.

This difference between the two aspects of the work was also relevant in the design of the finite element mesh. For the pre-arcing time simulations the fuse was divided into segments of 5 mm in length. This allows the model to be quickly adapted to model fuses of different length by inserting additional elements. The number of elements could also be changed rapidly by modifying one of the 5 mm sections. The finite element model of the fuse is shown in fig. 3.

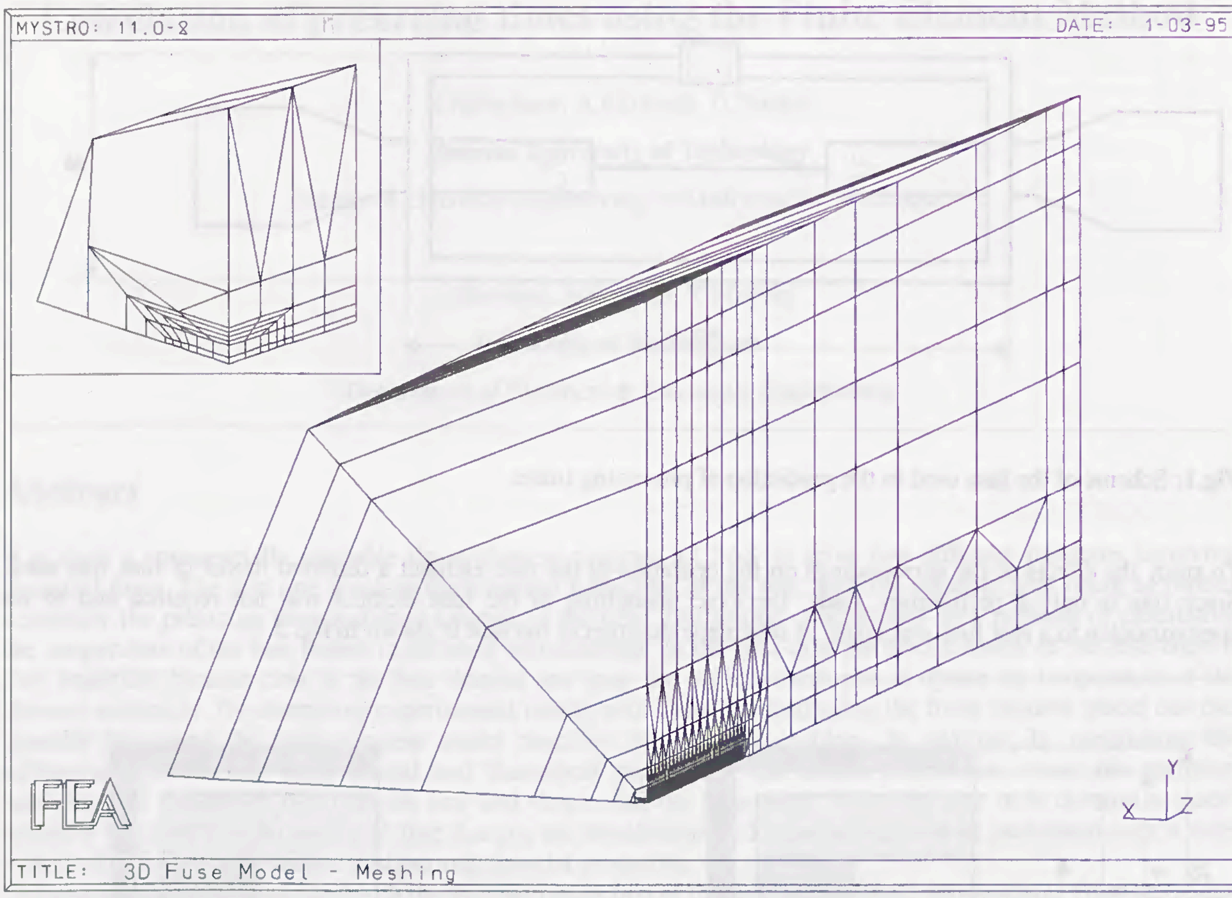

Fig. 3: Ilmenau Finite Element fuse model.

The requirement that the simulations should predict experimental results as closely as possible sets a limit on the size of the elements in the finite element model. In regions of large temperature gradients small elements were used and the converse is also true (i.e., in parts of the fuse where the temperature gradients is small, larger elements could be used.) The smallest element used in the first part of the simulations was 0.035 mm. All elements were allowed to have a temperature dependence of their material parameters. The LUSAS software achieved that by using the spaced averaged temperature to calculate the material properties for the next time iteration. In the two simulation cases there are 3 lines of mirror symmetry. That was used to reduce the total number of nodes by a factor 8. The 3D model was constructed using the standard elements types of the LUSAS package, the HF8 and PF6 element types with 8 and 6 nodes respectively. They had the approximate shape of a rectangle and pyramid. The total number of nodes and elements for the calculation of the pre-arcing time fuse amounts to 4500 and 3500 respectfully. The calculations were performed on a R4400 workstation (4x Ip19 CPU/ R4010 FPU /150 MHz /512 MB main memory). The large simulations look up to 12 hours on the workstation. There was no attempt to run these jobs on a PC. Whereas for the environmental simulations then number of nodes and elements was much smaller, a Pentium PC (60 MHz, 32 MB) was used to run these jobs.

Choice of material properties is of paramount importance for fuse especially when the calculations are used to predict the prearcing time. Data in the literature are not comprehensive and one often only gets values over a limited range or there is large discrepancies between the data derived by different authors. The material properties used in the current simulations are shown in **table 1**. There is no need to take into account the heats of vaporisation or melting since the simulations are stopped when the maximum element temperature reaches the melting point.

Material	(W/m/K)	(kg/m^3)	c_p (J/kg/K)	Source
Silver	444...347	10.5...9.3	235...311	[8] [11]
Brass	112	8.56...8.12	390	[8]
"Pertinax"	0.21	1300E-3	2500	[8]
Sand	0.44	1.7	605...1708	[8] [9] [10]

Table 1: Material properties.

3. Bordering and starting conditions

The simulation of the heat flow through a fuse requires two boundary conditions. The first is the location and magnitude of the heat sources. Since the heat is generated by ohmic losses within the fuse restriction , the amount of heat generated will depend on the temperature because the resistance of the metallic fuse element depends strongly on the temperature. The second boundary condition concerns the surface of the fuse or mounting case. In addition to these boundary conditions one needs to define the initial temperatures of the nodes. In the LUSAS suite of programs the PDSP load case allows the user to specify the initial temperature. In this case the initial temperature was defined to be 13 °C. At the surface of the fuse one needs to take into account convection and radiation. This is achieved by using the LUSAS ENVT load case. This load case requires a variable called Thermal Exchange Constant, and the value used in this experiment was 5 $Wm^{-2}K^{-1}$.
As described above the magnitude of the internal heat sources will depend on the element temperature. This is achieved by using the RIHG load case. LUSAS suite of programs contains a pre and post processor which makes the specification of the finite elements very simple and user friendly. Unfortunately, the RIHG is not supported by MYSTRO the LUSAS pre-processor. There are two methods of overcoming this problem. The first is to hand code the RIHG element into the LUSAS data file and the second one is to use an additional program [7] to modify the LUSAS data file. Both approaches were done. When large numbers of elements were used the later approach is recommend due to the time constraints of hand encoding.

4. Measurements

There are two aspects of this work. In the first series of simulations the aim is to use the theoretical model to predict accurately the prearcing time of the fuse. This requires a detailed knowledge of the material parameters. By solving the stationary heat flow problem, one can obtain the temperature distribution inside a fuse for currents below the fusing current. This temperature distribution can then be compared to the experimental values which allows the optimisation of the material parameters. This experimental determination of the temperature values must be done with great care. Thermocouples could not be used because the heat flow down the thermocouple wires distorts the temperature profile of the fuse. To overcome this problem the resistance of the element was used to estimate the temperature of the element. The comparisons between the predicted and measured temperatures in the fuse element is shown in **fig. 4**.
Measurements to obtain the t-I-characteristic were performed using an transient recorder. The melting time being defined as the time between the current's turn-on and the current's break down.

5. Comparison between calculation and measurement

Since the stationary calculations of the fuse elements temperature were only a preliminary set to test the quality of the finite element fuse model, the results shown in **fig. 4** represent a good correspondence between calculation and measurement. To get the t-I-characteristic for an electric fuse, it's necessary to perform transient calculations for a large set of currents. The transient analysis, for each current, was continued until part of the fuse element reached the melting point of the element. The results are shown in **fig. 5**. The calculated melting times are a little higher than the measured. However, this is not an problem since it produces an additional safety factor.

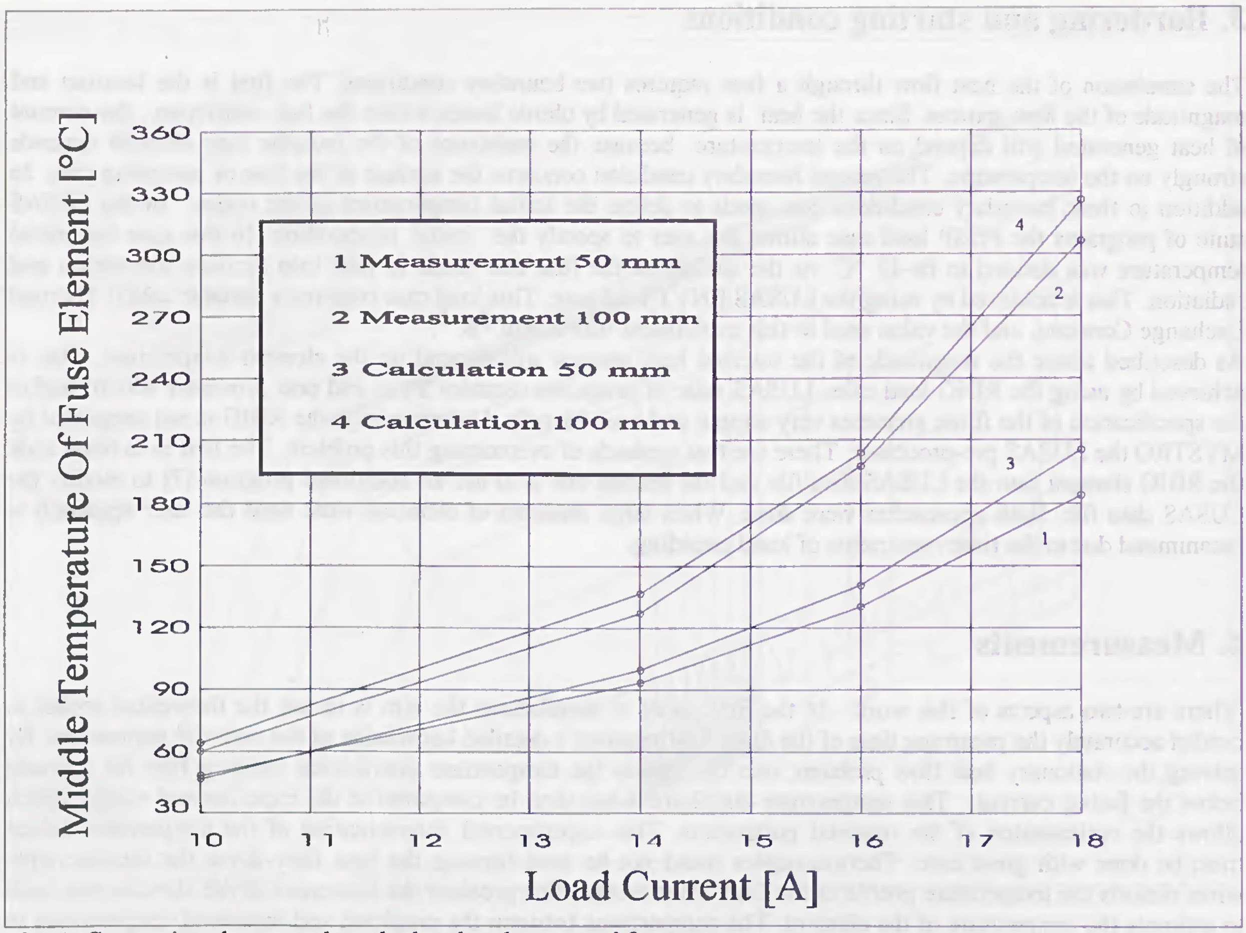

Fig. 4: Comparison between the calculated and measured fuse temperature.

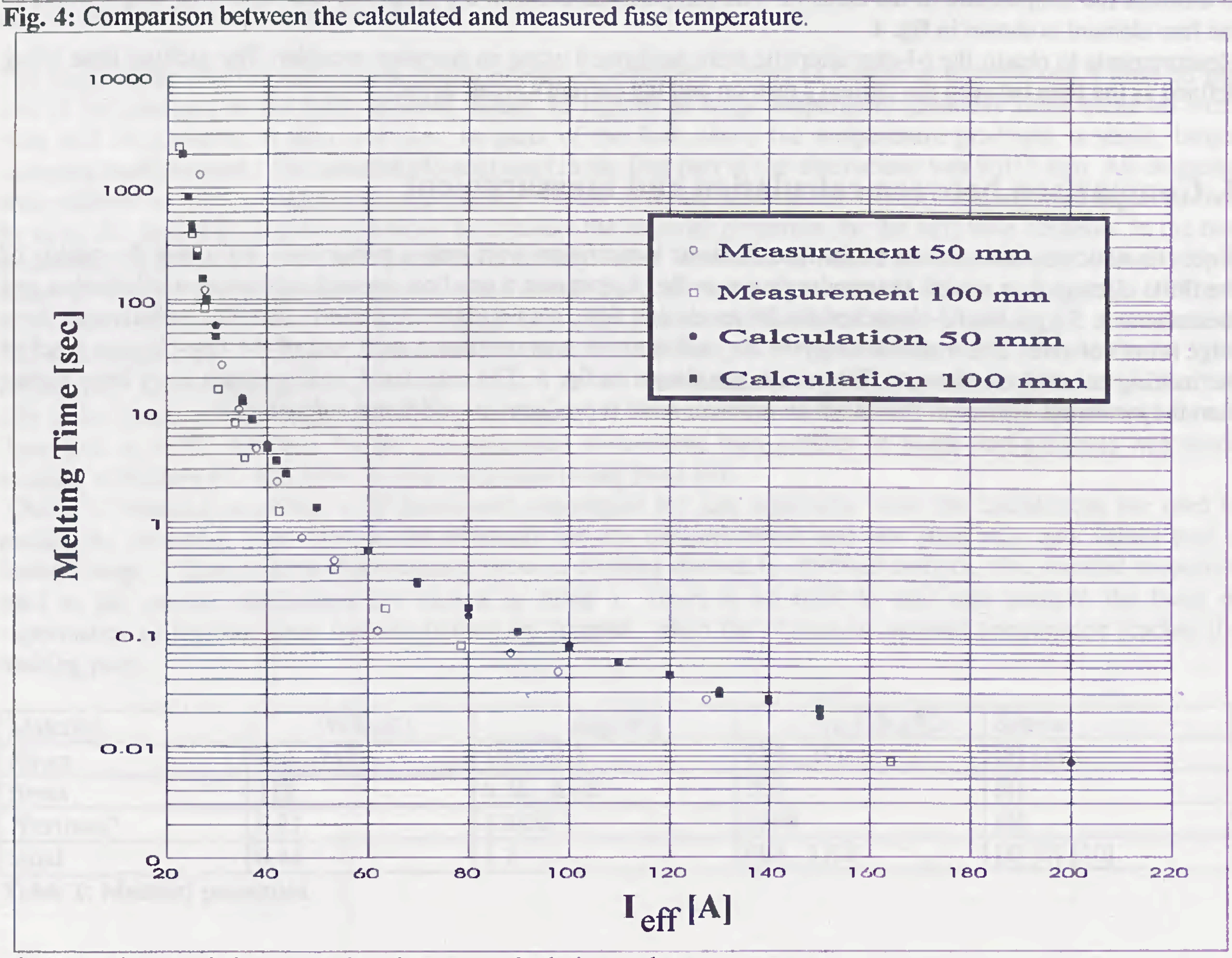

Fig.5: t-I-characteristic, comparison between calculation and measurement

The second part of the study concentrated on studying the effects of a fuse in close proximity to a case. Experiments performed at Bussmann (Cooper) UK have shown that the effects of placing a fuse in a case can be significant. The temperature of the outside body of the fuse can increase by up to 38K [5]. Presented in figures 7 and 8 are the temperature variations of the fuse element and the body temperature as a function of case thermal conductivity (k) receptivity. It can be seen that the temperatures of both the fuse element and fuse body are greater for the fuse in a case than without one. The size of the increase depends on the thermal conductivity of the case. For cases with low thermal conductivity the temperature rise is more significant than those with a higher thermal conductivity. The temperature of the case close to the point where the wires are fed through to the fuse is at a temperature higher than that of the central region of the box. This suggests that heat flow through the element and the connecting wires is extremely important in the cooling of a fuse element.

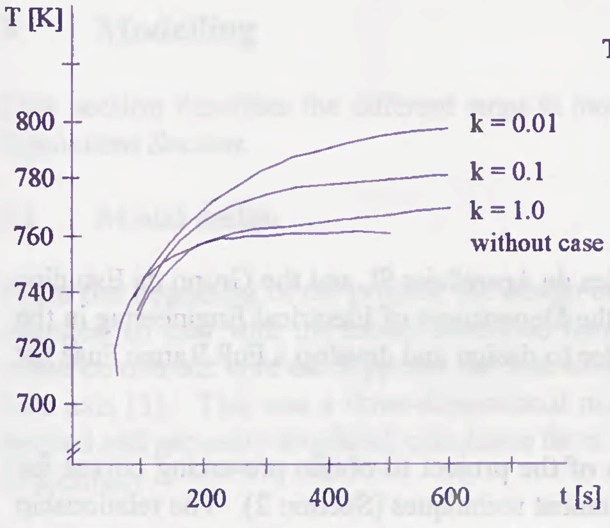

Fig.7: Element temperature

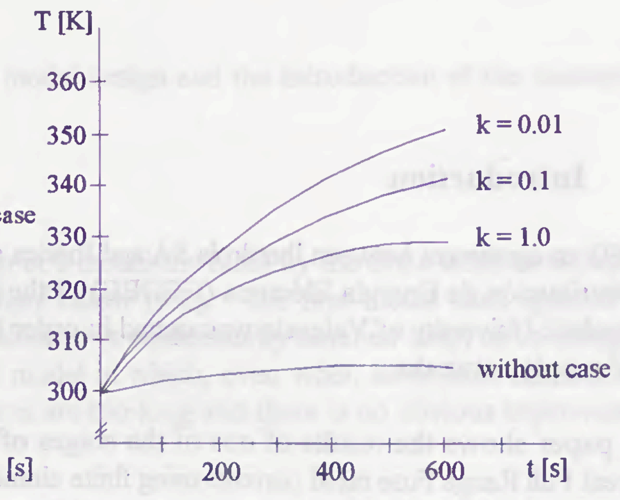

Fig.8 Body Temperature

6. Conclusions

It has been shown, by the results presented in this paper, that it is possible to calculate the t-I characteristics of fuses by finite elements. In addition, it has been shown that finite elements can be used to model the temperature profile of a fuse held in a cabinet.

References

[1] C.Garrido and J Cidras, "Analysis of pre-arcing time-current curves of fuse links" p 178 ICEFA 1991 (Nottingham).

[2] L.A.V Cheim and A.F. Howe, "Calculating fuse pre-arcing times by transmission line modelling (TLM) pp 162 ICEFA 1991 (Nottingham)

[3] M.Lindmayer, and M.Luther "Fusing and Short Circuit Interruption Behaviour of Metal Film Fuses" pp 107 ICEFA 1991 (Nottingham)

[4] LUSAS Finite element Analysis System (Theory Manual)

[5] A.Preidel and R.D.Harrison, Private communication

[6] Schumann, K.: Anlage zur Aufnahme von Wiederverfestigungskennlinien. 1992 Ilmenau: Techn. Hochsch., Fakultät für Elektrotechnik und Informationstechnik, Diplomarbeit 173-92D-03

[7] Programm RIHGGEN - eine Ergänzung für LUSAS. FEM-CAD Ingenieurbüro Ilmenau 1995.

[8] Mende / Simon: Physik. Gleichungen und Tabellen. 9. Auflage VEB Fachbuchverlag Leipzig 1986.

[9] Weissgerber,W: Untersuchungen über die Temperaturfelder von Hochspannungs-Hochleistungs-Sicherungen. Dissertation A, Technische Universität Hannover (1971).

[10] Hofmann,M.: Experimentelle und rechnerische Untersuchung von Ansprechkennlinien und Alterungsvorgängen bei Sicherungsschmelzleitern. Dissertation A , Technische Universität Carola-Wilhelmina Braunschweig (1987).

[11] Thermodata System.