THERMAL MODELLING OF A HEAT BLOCK AND HEAT SINK CONNECTED BY HEAT PIPES USING FINITE ELEMENT METHODS

A Groenewald & RT Dobson

Department of Mechanical Engineering, University of Stellenbosch,

Private Bag X1, Matieland, 7602, South Africa

Tel 27 17 634 2129

E-mail: abrahamg@megchem.co.za

ABSTRACT

This article considers the heat transfer between a simple heat block and a heat sink, using heat pipes as the energy transport medium. Both the heat block and heat sink are made from carbon steel. The heat pipes have copper containers and uses water as the working fluid, with a phosphor bronze mesh wick. The source, sink and heat pipe combination were modelled using finite elements. The program used was Nastran 2.0 for Windows, although the finite element application would be universal. The heat pipe was assigned constant properties, namely density, specific heat capacity and also conductivity. The purpose of the research was to investigate the merits of using constant properties for heat pipes in a finite element simulation. Using constant properties is a significant simplification of the complex internal thermo-fluid processes of a heat pipe, and will result in much faster calculation times. The results indicated though that this type of simplification may lead to inaccuracies. It would be beneficial if a heat pipe property can be developed and assigned to a three dimensional finite element. Another (and maybe more realistic) option would be to rather do modelling of heat pipes with a finite element based program that is able to incorporate non-linear boundary conditions, with which the heat pipe's boiling and condensation processes can be modelled.

1. INTRODUCTION

An investigation into the use of heat pipes for the thermal management of casting moulds were undertaken. By being able to control the heating and cooling rates of casting moulds, production rate and quality can be significantly improved. [1]

Although methods like finite elements and finite differences can readily be used to calculate temperature distributions throughout a mould, and many standard computer packages are available for these analyses, there is not a standard element that can easily be used to model a heat pipe. Therefore a simple heat pipe element (having constant properties) was developed to include in a FEM package. (The heat pipe properties could also be used in a finite difference or finite volume based approach) The theoretical calculations were compared to experimental results, so as to compare the accuracy and applicability of this type of analysis. The finite element approach was chosen instead of other types of analysis methods (such as finite volume or finite difference methods), since in many engineering applications one of the output requests could be to determine thermal stresses caused by the resulting temperature field.

2. EXPERIMENTS PERFORMED

The following experiments were performed:

1. Determination of the heat pipe properties
2. Verification of the heat source and heat sink material and surface properties
3. Heat transfer between a heat source and heat sink using heat pipes

The experimental set-up and procedures of experiment 1 will not be discussed, but only the results are displayed, in section 5.1. For experimental details refer to [1].

Experiments 2 and 3 are discussed in more detail in sections 3 and 4, and their results are located in sections 5.2 and 5.3 respectively.

3. EXPERIMENTAL SET-UP

The heat pipes used for the experiments have copper containers and are filled with water as the working liquid. A double layer of phosphor bronze wire mesh (150 mesh) is lined against the inside of the container and acts as the wick structure. The heat pipe is shown schematically in Figure 1. Figure 2 shows the heat source and heat sink blocks connected to each other by the heat pipes. Figure 3 shows a diagram of the blocks and heat pipes and shows the relevant measurements that were used in this study.

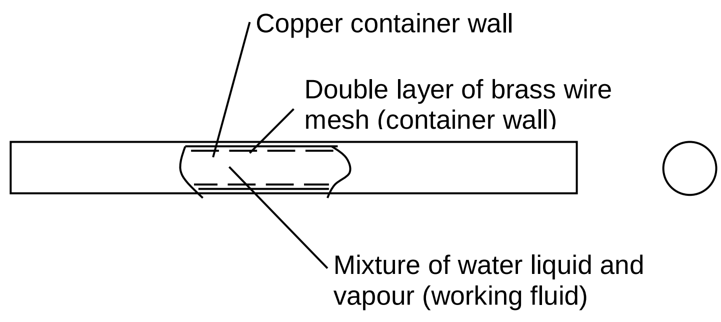

Figure 1 Schematic of heat pipe

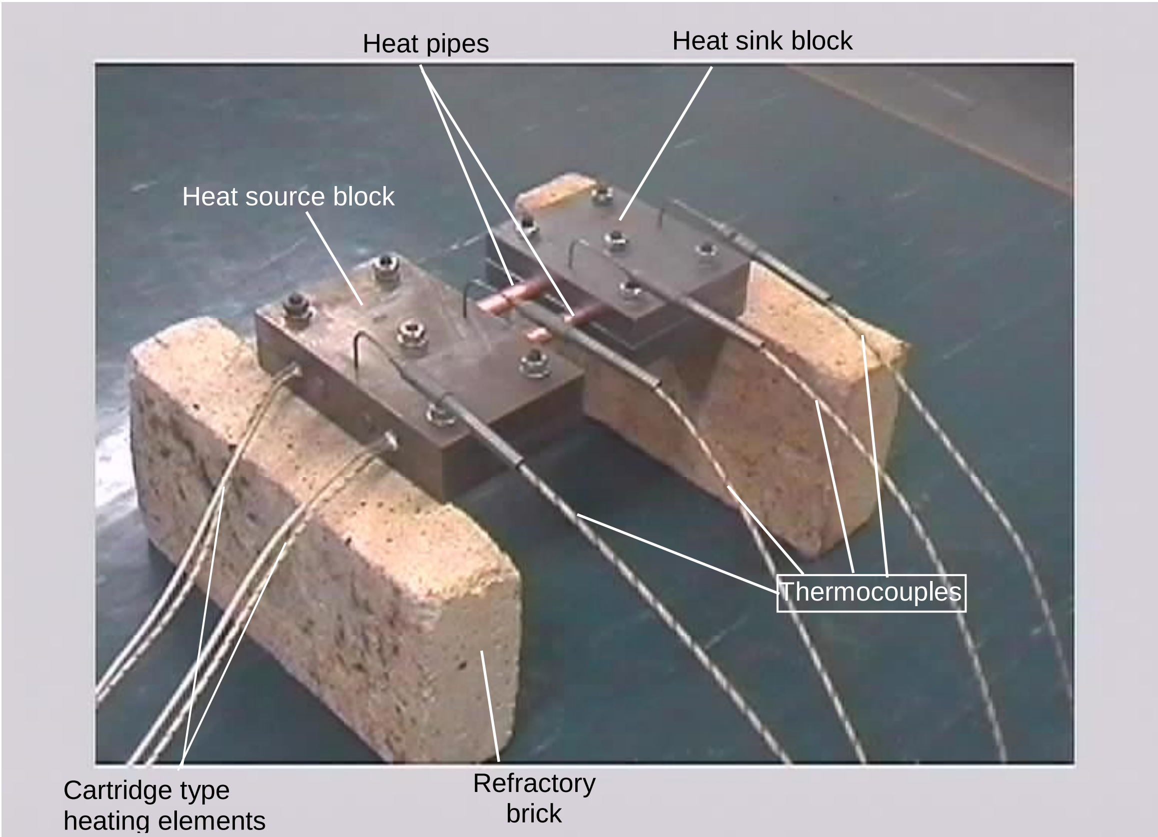
Heat pipes
Heat sink block
Heat source block
Thermocouples
Cartridge type
heating elements
Refractory
brick

Figure 2 Heat source and heat sink blocks connected via heat pipes

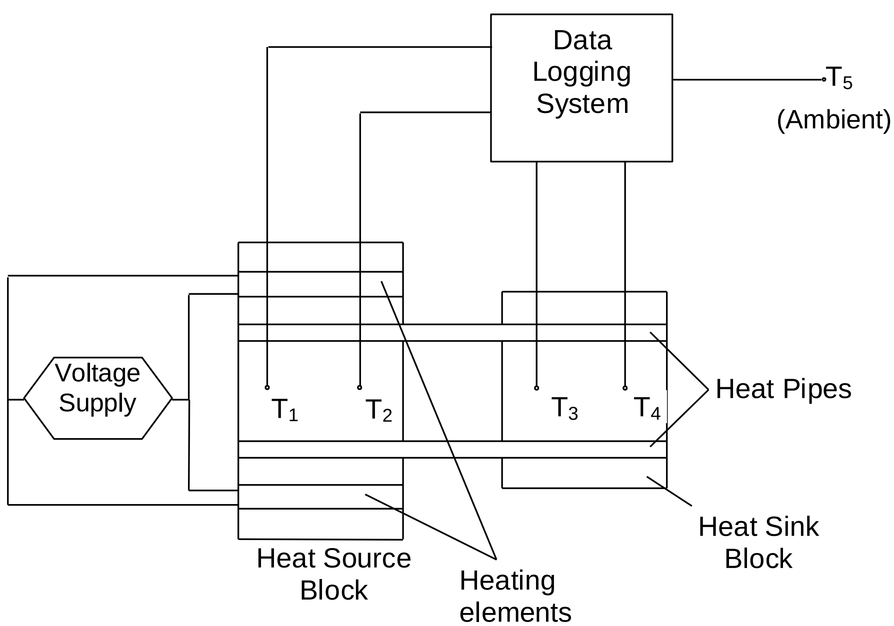

Figure 3 Schematic experimental lay-out. The heat source block has dimensions of 30x80x130 mm and the heat sink block 30x80x100 mm. The length of a heat pipe is 203.2 mm (8 inches) and the outside diameter is 9.525 mm (3/8 inches)

4. MODELLING

4.1 Material properties

Figure 4 shows the solid round bar used to model the heat pipe's thermal characteristics.

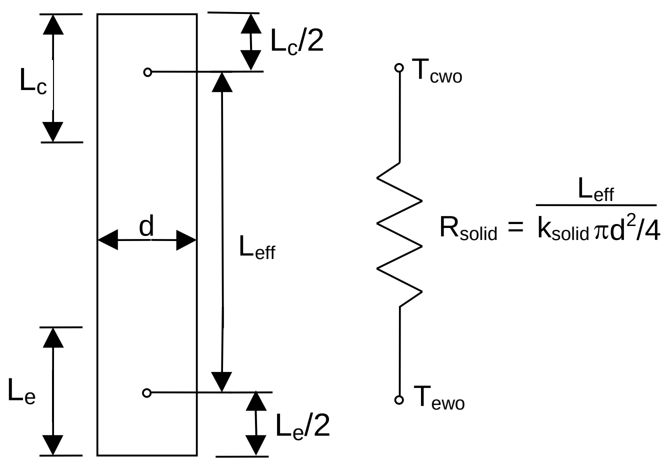

Figure 4 Axial thermal resistance of a heat pipe, modelled as a solid round bar

For the bar in Figure 4, the dimensions are the following:

$L_e = L_c = 0.08m$; $L_{eff} = 0.1232m$; $d = 0.009525m$

Conductivity

The total thermal resistance over the heat pipe is the sum of the evaporator and condenser resistances. These two external areas are the same and the heat transfer coefficient for both is taken as 2000 W/m^{2}K (refer to section 5.1). The total thermal resistance (which should then be equivalent to R_{solid}), is given by:

$$R_{solid} = R_c + R_e = 2\left(\frac{1}{(2000)\pi(0.009525)(0.08)}\right) = 0.41773\ °C/W \quad (1)$$

It follows that

$$k_{solid} = \frac{L_{eff}}{R_{solid}\pi d^2/4} = \frac{0.1232}{(0.41773)\pi(0.009525)^2/4} = 4139\,W/mK \quad (2)$$

Density

The mass of the heat pipe is 45 grams, and it's volume is 14.48x10^{-6} m^3, thus the effective solid density is calculated as:

$$\rho_{solid} = \frac{0.045}{14.48\times10^{-6}} = 3107.9\ kg/m^3 \quad (3)$$

Specific heat capacity

The copper container (and brass wick) is assumed to be the only masses that have heat capacities. The specific heat capacity of copper is therefore given to the solid element, so that $C_{p(solid)} = 385\,W/mK$.

The other properties needed for the performance of the numerical simulation include those of the steel blocks, the natural convection and radiation coefficients, as well as the properties of the heating elements, which consist of a steel filament (used as the heating wire) that is spun around a MgO_2 core. The properties used are summarised in Table 1. (The verification of the material properties is discussed in section 5.2)

Table 1 Material properties and other coefficients used for the analysis

APPLICABLE MATERIAL	PROPERTY	VALUE
Steel	Convection coefficient	h_{NC} = 5 W/m^{2}K
	Radiation emmisivity	ε_{rad} = 0.5
	Radiation absorbtivity	α_{rad} = 0.5
	Density	ρ_{steel} = 7840 kg/m^3
	Conductivity	k_{steel} = 50 W/mK
	Specific heat capacity	$C_{p(steel)}$ = 450 J/kgK
Heating filament (Inconel)	Density	$\rho_{filament}$ = 8000 kg/m^3
	Conductivity	$k_{filament}$ = 50 W/mK
	Specific heat capacity	$C_{p(filament)}$ = 600 J/kgK
Heating element isolation material (magnesium oxide)	Density	$\rho_{(magnesium\ oxide)}$ = 4000 kg/m^3
	Conductivity	$k_{(magnesium\ oxide)}$ = 15 W/mK
	Specific heat capacity	$C_{p(magnesium\ oxide)}$ = 1000 J/kgK
Heat pipe	Density	$\rho_{(heat\ pipe)}$ = 4000 kg/m^3
	Conductivity	$k_{(heat\ pipe)}$ = 15 W/mK
	Specific heat capacity	$C_{p(heat\ pipe)}$ = 1000 J/kgK

4.2 Geometric modelling and finite element discretisation

The model is of such a geometry that it is symmetrical around the plane defined by the y and z axes (the y-z plane), as well as around the x-z plane. This allows for the modelling of only a quarter of the geometry, as is seen in Figure 5. The finite element discretisation of the model is shown in Figure 6 to Figure 8. Linear brick elements (thus a solid element, having 8 nodes) are used.

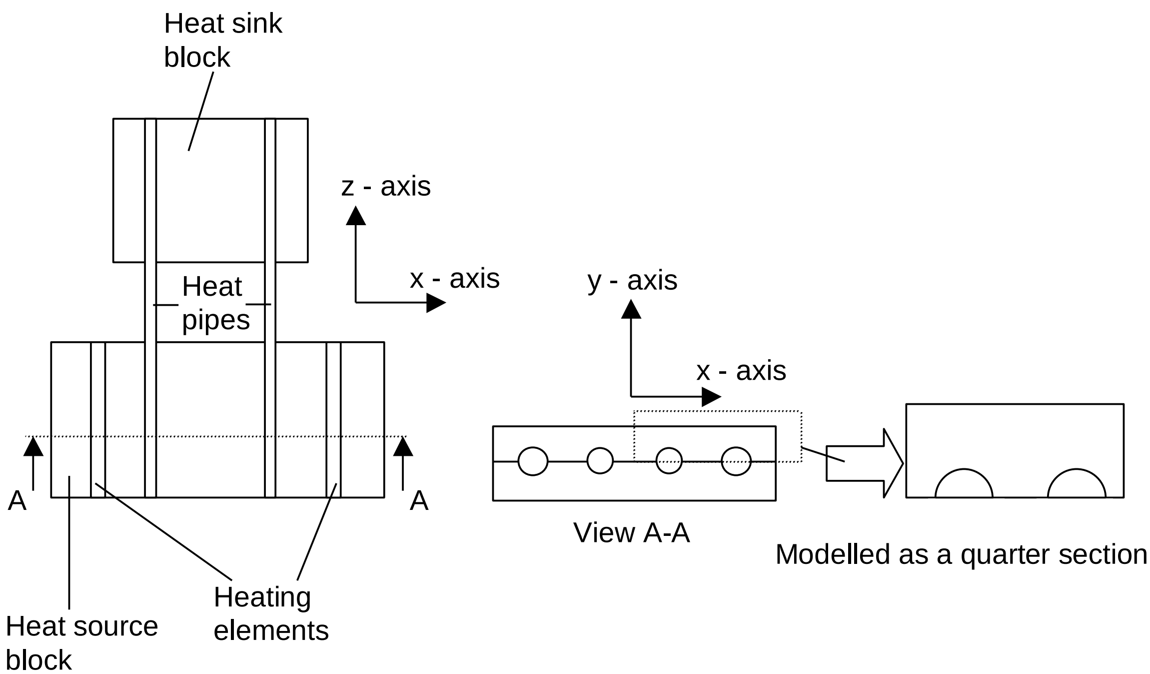

Figure 5 Schematic geometric model of the set-up, showing the reduction to a quarter-model

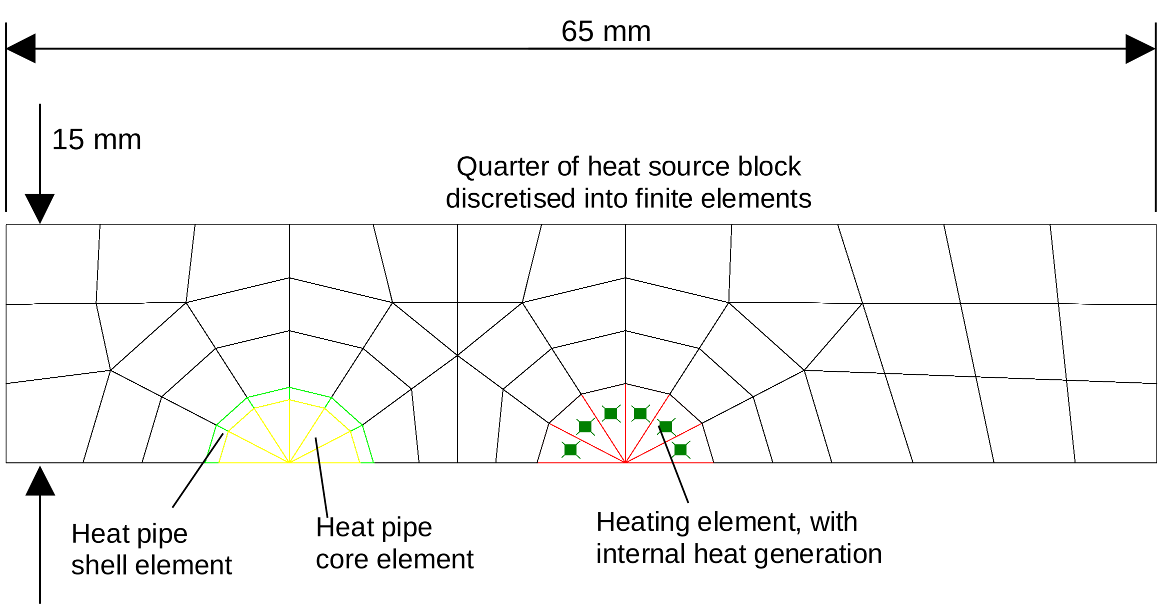

Figure 6 Side view of the discretised finite element model

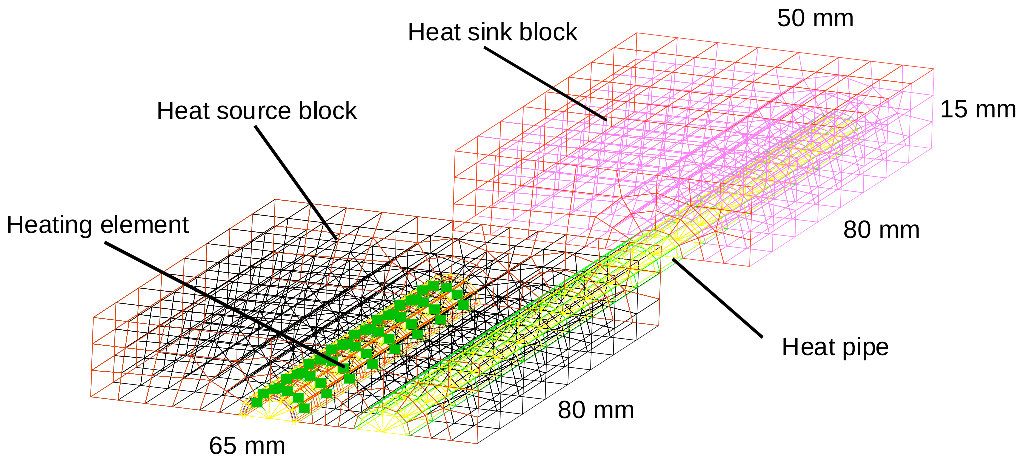

Figure 7 Three dimensional view of the discretised finite element model

4.3 Loads and boundary conditions

The thermal load is applied by assigning a volumetric heat generation to the resistance wire in the heating element. A 250 W heat load was produced inside each heating element during the experiments, and the volumetric heat generation in the resistance wire was calculated accordingly. The convection and radiation boundary conditions are shown in Figure 8.

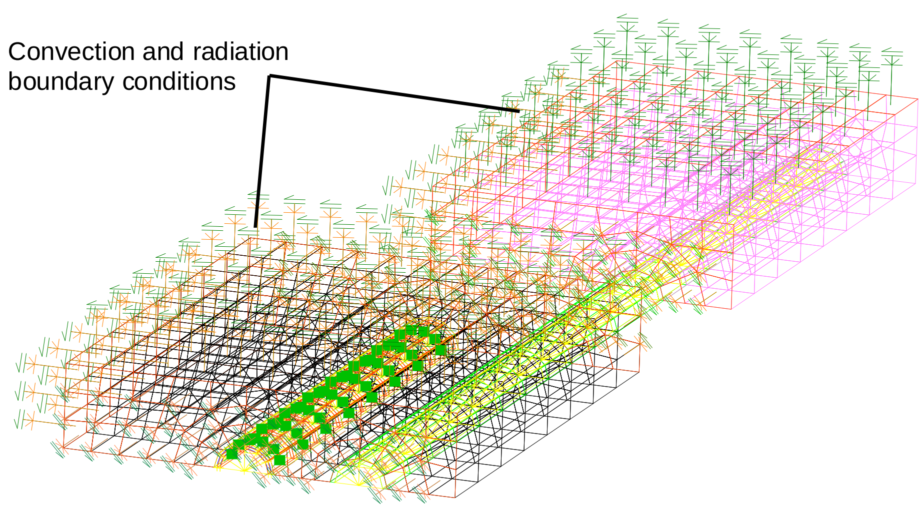

Figure 8 Three dimensional finite element model, showing convection and radiation boundary conditions on the applicable surface elements

5. RESULTS

5.1 Heat pipe properties

From the experimental data obtained [1], the heat transfer coefficients of both the evaporator and condenser ends were experimentally established as functions of the heat transfer rate, the temperature difference over the heat pipe as well as the internal temperature of the heat pipe. For this finite element analysis, though, the heat pipe properties have to be constant, and a method has to be found to do this. Figure 9 shows an averaged heat transfer rate versus temperature difference between the outside wall temperatures of the evaporator and condenser ends of the heat pipe. There are two lines present on the graph, one for a standard heat pipe, and one for a 'special' heat pipe. The experiments required that a 'special' heat pipe, having two thermocouple holes had to be manufactured. This special heat pipe is shown in Figure 10. The heat transfer coefficients were calculated using the average of the results for the normal and special heat pipes. Figure 9 shows that there is a slight difference between the results for the specialand normal heat pipes. Whether this effect is due to the changed geometry, or in fact experimental error, is not known, but the heat transfer coefficient is calculated using the average of the results for the normal and special heat pipes.

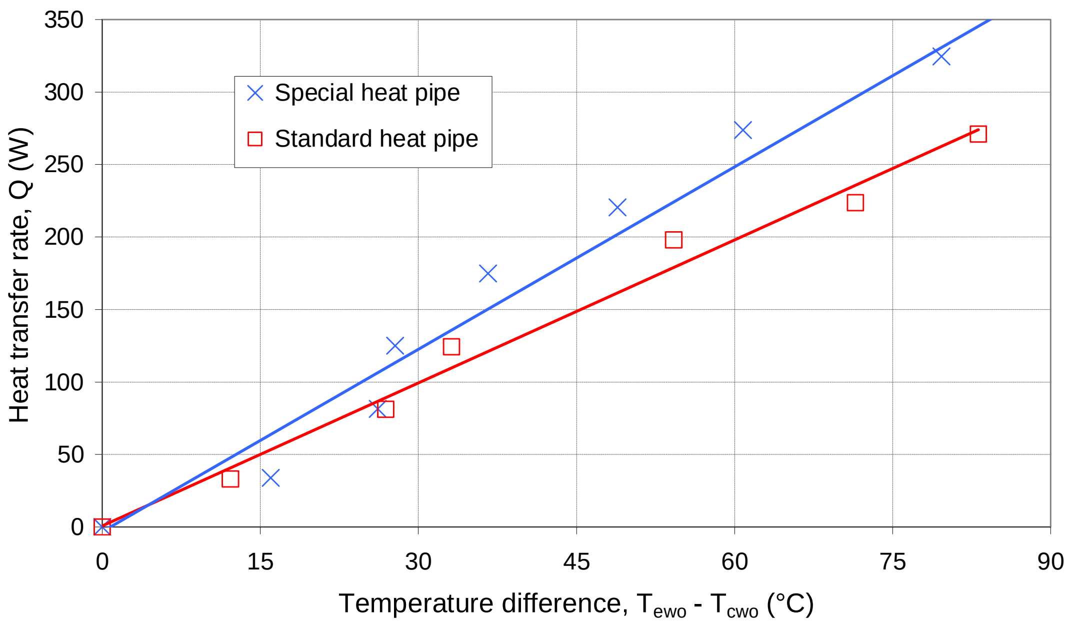

Figure 9 Comparison between heat transfer rate versus temperature difference for a normal and special heat pipe

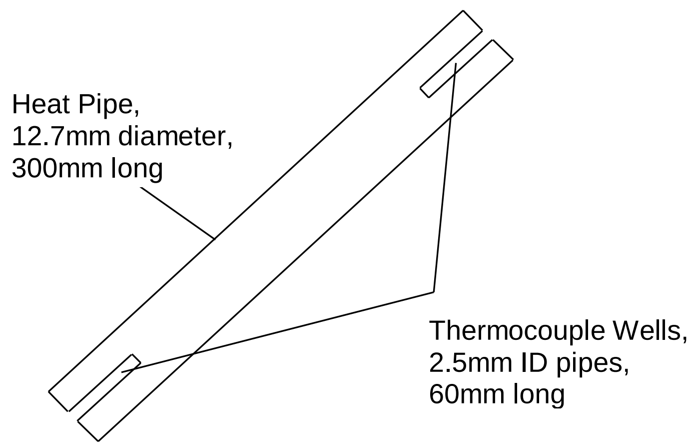

Figure 10 Schematic of the special heat pipe

From Figure 9, the heat transfer coefficient of the heat pipe can be calculated. For simplicity, it is assumed that the condenser and evaporator end heat transfer coefficients are the same. The dimensions of the heat pipe in the experiment [1] was the following: L_e = 0.08 m, L_c = 0.11 m and d = 0.0127 m. Thus:

$$R_{t(normal)} = \frac{\Delta T}{\dot{Q}} = \frac{45}{150} = 0.3\,°C/W \quad \text{(normal heat pipe)} \tag{4}$$

$$R_{t(special)} = \frac{\Delta T}{\dot{Q}} = \frac{60}{250} = 0.24\,°C/W \tag{5}$$

$$R_{t\_av} = 0.27\,°C/W \tag{6}$$

$$R_{total} = R_c + R_e = \frac{1}{h_c A_c} + \frac{1}{h_e A_e} = \frac{1}{h_{hp}}\left(\frac{1}{\pi(0.0127)(0.08)} + \frac{1}{\pi(0.0127)(0.11)}\right) \tag{7}$$

thus

$$h_{hp} = \frac{\frac{1}{\pi(0.0127)(0.08)} + \frac{1}{\pi(0.0127)(0.11)}}{0.27} = 2004.3\,W/m^2K \tag{8}$$

Therefore, the heat transfer coefficient is taken as h_{hp} = 2000 W/m^{2}K. (As used in section 4.1)

5.2 Verification of heat source and sink material and surface properties

For this experiment, the heat source block was not connected to the heat pipes, but it was heated on it's own. After a while the heat supply was switched off, and the block was left to cool. The comparison between the theoretical and experimental temperatures in the block is shown in Figure 11. It can be seen that four temperature profiles are represented on the graph – an experimental temperature and three temperatures calculated by using finite element methods. The difference between T(FEM 1), T(FEM 2) and T(FEM 3) is that different natural convection coefficients and surface radiation properties were assumed for each simulation. These different values used are shown in Table 2.

Table 2 Convection and radiation values used for the results of Figure 11

PARAMETER	FEM 1	FEM 2	FEM 3
Natural convection coefficient	5 W/m^2K	10 W/m^2K	3 W/m^2K
Radiative emmisivity	0.5	1	0.2
Radiative absorbtivity	0.5	1	0.2

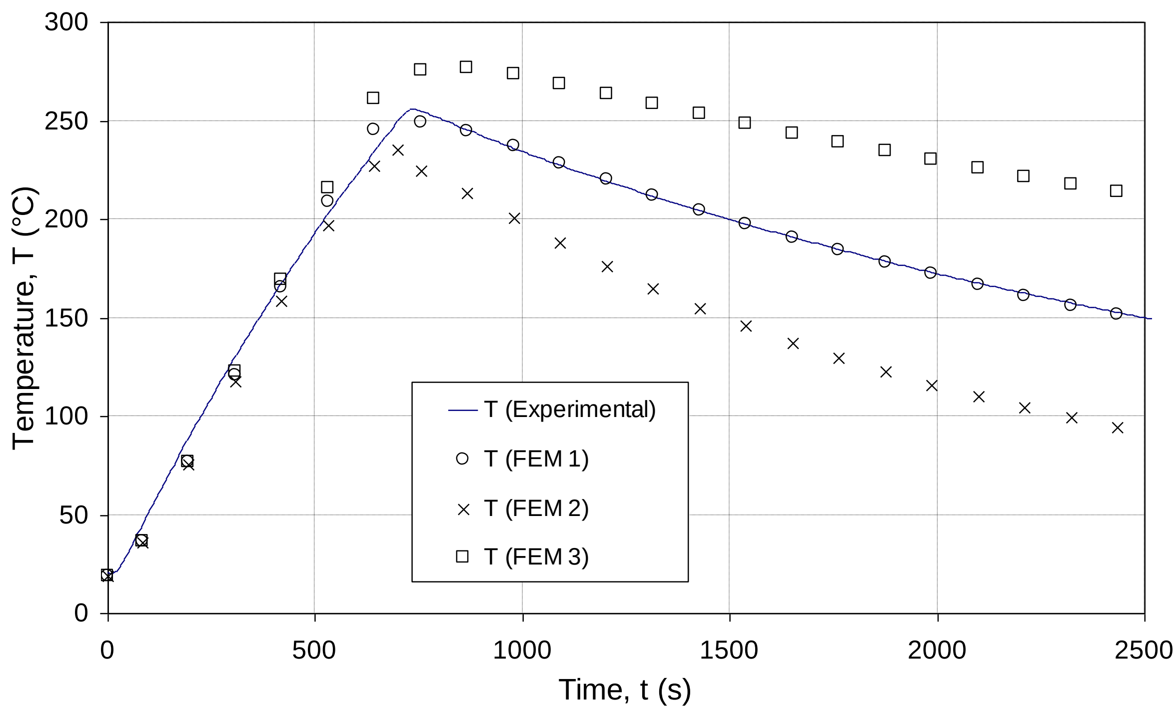

Figure 11 Experimental and theoretical (using FEM) temperatures when the heat source block is heated on its own, for different convection and radiation surface coefficients assumed

From Figure 11 it can be seen that the analysis using the properties associated with FEM 1 gave the most accurate answers. As a matter of fact, the theoretical and experimental temperatures are in very good agreement for this case. These type of sensitivity analyses were performed to determine the influence of the other parameters on the accuracy of the simulation. Other parameters that were investigated were the heating element material properties and the steel properties. The end result was that the properties and values listed in Table 1 produced simulation results that were closest to the experimental results.

5.3 Heat transfer between the heat source and heat sink using heat pipes

After the properties and surface boundary conditions of the heating block was tested and verified, the main experiment was performed, which is the energy transfer between the heat source and heat sink by means of heat pipes. The finite element model of the set-up was analysed, and the properties and values shown in Table 1 were utilised.

Figure 12 shows the comparison between the experimental and theoretical temperatures of the heat sink and heat source blocks. For the geometric locations of T_1 to T_4, refer to Figure 3.

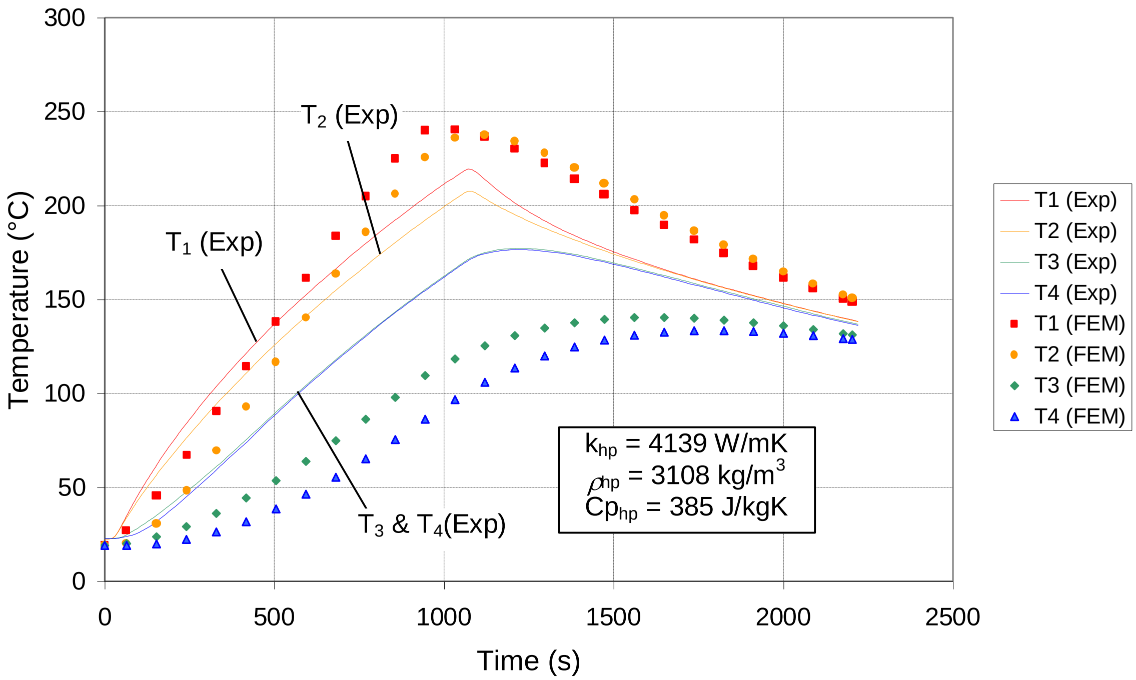

Figure 12 Experimental and theoretical (using FEM) temperatures of the heat source block and the heat sink block (connected by heat pipes), using the originally calculated properties for the heat pipes

From Figure 12 it can be seen that there is an unacceptably large difference between the theoretical and experimental temperatures. The question arises what the influence of the equivalent heat pipe properties, calculated in section 4.1, will be on the accuracy of the theoretical model. To assess this influence, different values for the conductivity, density and specific heat capacity of the heat pipe can be used and the results can be evaluated. Figure 13 shows theoretical and experimental temperatures when the finite element simulation was performed using the properties specified in Table 1, except for the thermal conductivity of the heat pipe, which was taken as k_{HP} = 25 000 W/mK. Through inspection, it can be seen that the theoretically predicted temperatures are much closer to the experimentally measured temperatures when this is done.

A way of determining the 'average error' between the theoretical and experimental temperatures is to consider the difference between the temperatures and defining the error E in the following way:

$$E = \sum_{n=1}^{N} \sum_{i=1}^{4} \left| T_{i\,(experimental)}^{n} - T_{i\,(theoretical)}^{n} \right| \Big/ N \tag{9}$$

where n is the number of the time step and N is the total number of time steps considered.

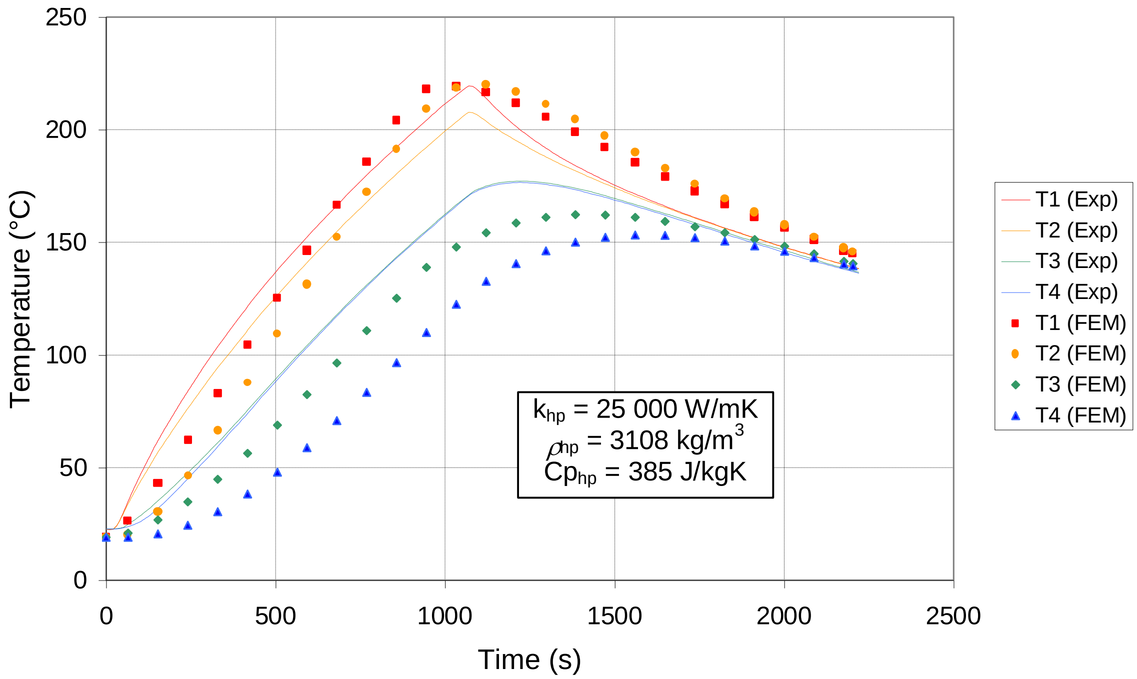

Figure 13 Experimental and theoretical (using FEM) temperatures for the heat source and heat sink block (connected by heat pipes), using a value of k_{HP} = 25 000 W/mK and the other originally calculated properties for the heat pipes

A total of 15 simulations similar to those shown in Figure 12 and Figure 13 were performed, using different values for the properties of the heat pipes. Equation 4 is used to evaluate the ‘accuracy’ of the theoretical results compared to the experimental temperatures.

The results (error values) of these 15 simulations are shown in Figure 14.

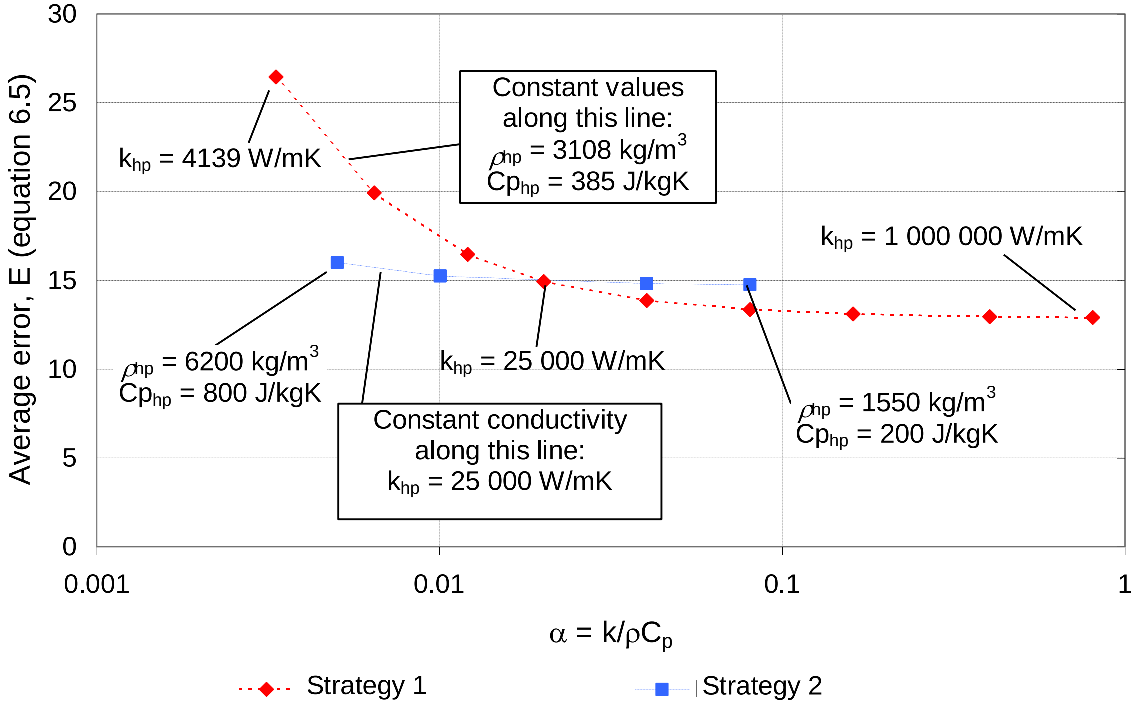

Figure 14 Average error (using equation 9) versus the thermal diffusivity for different properties of the heat pipe used in the FEM simulation

Two strategies were followed to obtain the results shown in Figure 14. Strategy 1 is to keep the values for the specific heat capacity and density of the heat pipes constant, while changing the thermal conductivity to obtain different thermal diffusivities. For strategy 1, the average error starts at around 27 for a conductivity value of 4139 W/mK, drops down to around 15 for k_{HP} = 25 000 W/mK and seem to converge at around 13 for k_{HP} = 1 000 000 W/mK. Since the choice of k_{HP} = 25 000 W/mK seems to give reasonable results, it was chosen as the constant value for strategy 2, where the relative importance of the density and specific heat capacity was investigated. As can be seen from the strategy 2 line, the influence of the density and heat capacity on the average error is much less significant than that of the thermal conductivity.

However, it seems that no matter how high the conductivity, the temperatures are never simulated completely correct.

6. DISCUSSION AND CONCLUSIONS

It appeared that no matter how high the conductivity value that was assigned to the heat pipe was chosen, the temperature results for the theoretical simulation never followed that of the experiments exactly. It seemed that some type of convergence was achieved, and the average error could not get any smaller, beyond a certain conductivity value. The fundamental difference between the mechanisms of heat transfer, which is latent heat of evaporation in an actual heat pipe, versus the conduction that takes place during the finite element analysis, is expected to be the main reason for the difference, since the material and surface properties of the steel and heating elements are expected to be reasonably accurate (refer to Figure 11).

It would be beneficial if a heat pipe element can be developed for use in a finite element analysis. This will require that the heat pipe element be derived from the phase-change behaviour that is inherent to the thermo-fluid processes in the heat pipe. Phase change can be handled in finite element analysis, and it is for instance currently used in certain casting simulation packages. The problem with heat pipes though, is that an energy balance needs to be achieved over the entire heat pipe, which will complicate matters significantly, and increase the processing time. A more realistic method would probably be to use a finite element program that can handle non-linear boundary conditions, so that correlations for the boiling and condensation processes can be used in the analysis.

One important parameter that was not changed during the analysis, was the coarseness of the finite element grid. It could be that a finer grid may produce more accurate results, and that the average error converges to a lower value.

7. RECOMMENDATIONS

It is suggested to investigate the possibility of developing a finite element for a heat pipe.

If a finite element for a heat pipe cannot be developed, it is recommended to use a finite element program that can accommodate non linear boundary conditions, and to use correlations for the boiling and condensation inside the heat pipe.

For this specific analysis, it is recommended that a finite element analysis is performed again, but using different grid sizes, to determine the effect on the accuracy of the simulation.

The methods used to determine the heat transfer coefficients of the heat pipes, were not very sophisticated. It is suggested that more accurate methods are used to do the experimental measurements on, in order to eliminate the possibility of experimental errors as far as is possible.