NUMERICAL INVESTIGATION OF CONJUGATE NATURAL CONVECTION HEAT TRANSFER IN CLOSED ENCLOSURE USING ONE PHASE LIQUID

Farouq Ali Gdhaidh
E-mail: fagi1980@yahoo.com
Bradford University (Mechanical Engineering department)
Sponsored by: Al- Zawia University

Introduction

1. Computer cooling is required to remove the waste heat produced by computer components to keep components within permissible operating temperature limits.
2. Current computer cooling technology uses fans to dissipate heat and hence cool the various components of the system.
3. This has many disadvantages in terms of power needed to operate the fans and techniques for reducing noise and vibration generated from the operation of such fans. Also the fans is the main cause of accumulation of dust on heat sink.
4. In this project it is proposed that coupled natural convection and conduction could be used to cool the system without using fan for heat sink.
5. In the previous studies for coupled conduction and convection the isothermal condition technique were applied.

Fig (1) forced convection

Aims & Objectives

The main aim of this work is to investigate numerically the possibility of cooling electronic components by coupling between conduction and natural convection within a heated closed cavity.
The following objectives were identified:

1. Understand the effect of one phase liquid on the maximum temperature and heat transfer by simulating CPU of desktop computer.
2. Using closed enclosure filled with water attached to the heat source and find how much heat could be removed when one exhausted fan is installed at the back of the computer covering case.
3. Finding the optimum enclosure width respecting to available space through the computer chassis and also the maximum working temperature.

Research Methodology

1. Some assumptions have been applied to the model as following:
   - Laminar natural convection inside the enclosure and turbulent within the computer chassis.
   - Three dimensional analysis.
   - Steady state heat transfer.
   - The effects of radiation and roughness for the enclosure are considered to be negligible.
2. Using ANSYS Icepak to simulate the desktop computer with some important part inside the computer chassis which affect the flow.
3. Simulate the geometry for vast range of heat input 15-40 W.
4. Using realistic boundary condition for cooled plate.

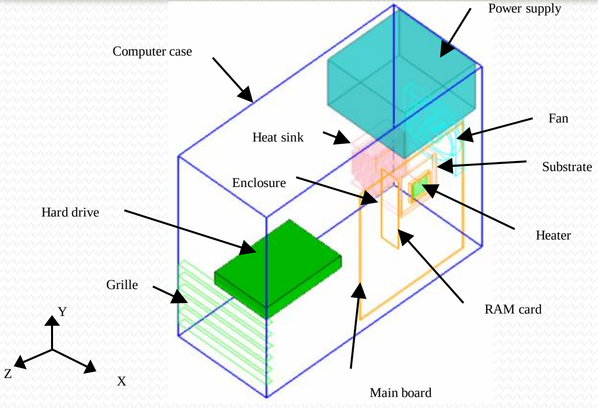

Fig (2)Schematic diagram of the physical model

Preliminary Results

The results showed that using enclosure filled with water reduced the maximum temperature of the system by 38% compared to case without enclosure figure 3. Moreover the temperature distribution at substrate/fluid interface displayed in figure 4 for the full range of the applied heat energy.
The present cooling system is able to keep the maximum temperature in the save level less than 85°C when the heat input is at 40W.

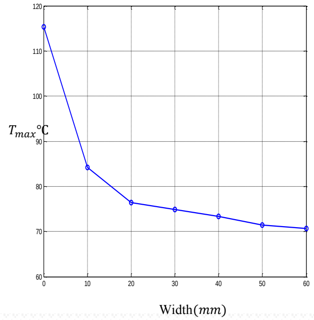

Fig (3) The enclosure width effects on the maximum heater temperature

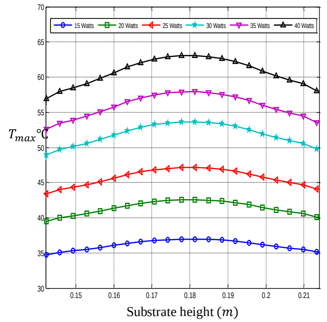

Fig (4) The local substrate/water interface temperature with different applied power

Conclusion and Further work

Conclusion:

1- The size of the enclosure has an effect on the value of the maximum temperature. However, it was also found that the temperature variation remains constant beyond certain enclosure size.

2- It was found that the addition of an enclosure with water had a greater effect on the value of the maximum temperature when compared to the case without enclosure. A reduction of 38% of maximum temperature was reached when the enclosure with water was added.

Future work:

1- The results will be obtained for two different fluids (air and water) in the cavity and they will be compared.

2- The numerical results will validate with experimental data and also the average heat transfer coefficient and Nusselt number will be obtained.

3- compare the proposed concept with other methods of cooling electronic chips.