Conference Experience Report

<table>
<thead>
<tr>
<th>Participant</th>
<th>Aw Wee Earn</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conference Name</td>
<td>ExHFT 8- 8th World Congress on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics</td>
</tr>
<tr>
<td>Dates</td>
<td>16-20 June 2013</td>
</tr>
<tr>
<td>Location</td>
<td>Instituto Superior Tecnico, Lisbon, Portugal</td>
</tr>
</tbody>
</table>

Objective(s) for attending this conference

The objectives for attending this conference are listed as below.

1. Cultivate capability for carrying out experiment and study of microchannel heat transfer
2. Training the proper academic paper writing skills
3. Training academics oral presentation skills
4. Obtaining valuable experience by the exposure of different social events

Important invited speaker

2013 Nusselt- Reynolds Prize Winner- Prof Brian E. Launder, University of Manchester

Sessions attended

1. Nusselt- Reynolds Prize Winner Lecture
2. Keynote Lecture: 1, 2, 3, 4 and 7
3. Poster Session
4. Oral presentation session: 1c, 2a, 3e, 4e and 5e

Session highlights

Prof Brian who is the winner of Nu-Re Prize 2013 conducted an interesting lecture in the main auditorium of Instituto Superior Tecnico. He briefly introduced the family background of Osborne Reynolds and Horace Lamb. Both of them are the historical fluids giant from Manchester. Their success in study of fluid was contributed by their effect in conducting experiments.

Another interesting speaker is Prof. Kenneth David Kihm, University of Tennessee, U.S.A. Prof Kihm is working under the study of near-field thermo fluidic characterization by using Surface Plasmon Resonance (SPR). He listed out some general imaging methods and their pros and cons. By comparing with others, the currently imaging method of his research team was believed to be the latest one. Their goal is to achieve the figure of merit up to 3 ~ 5.
Social event highlights

The committee of conference organized some interesting social events during the conference period. Here, I highlighted some of them. During the first day of presentations, all of the participants were invited to take a boat tour travelling along the Tagus River, the longest river on the Iberian Peninsula. There are plenty of beautiful views along the river bank. The boat passed through the old town of Lisbon with historic buildings and following in front of us were modern architectures. Besides that, we visited a delightful town called Sintra. Owing to its 19th-century Romantic architecture and landscapes, it becomes the paradise of travelers. The welcome banquet was held in Restaurant Mar do Guincho, which is not far from the Western-most point of continental Europe.

Other notes

This year around 350 extended abstracts had been accepted and at the end 250 full papers were submitted. ExHFT is a worldwide conference in the field of heat transfer, fluid mechanics and thermodynamics and it emphasizes on the effort of experimental studies. The discussion section in each session provides me a lot of ideas about the way of thinking of other people. From the discussion during my presentation, I obtained very useful information which will improve my experiment study in the future. Besides that, during the social events, communicate with people from all over the world was an unforgettable experience.

Photo Gallery

Main entrance of Instituto Superior Tecnico, Lisbon, Portugal
Oral Presentation

Social event-Boat tour

Cost summary
As attached in separate document.
AN EXPERIMENTAL STUDY ON THE COOLING CAPACITY OF ENHANCED MICROSCALE HEAT TRANSFER IN MACRO GEOMETRY

WEE EARN AW, KIM TIOW OOI, KIAN SHING KONG
School of Mechanical and Aerospace Engineering
Nanyang Technological University
50 Nanyang Avenue
SINGAPORE
WEAE1@e.ntu.edu.sg

ABSTRACT

The accelerated heat dissipation issue has increased the demand of more effective cooling technology, including the actively-researched microchannel heat sink. The main objective of this paper is to study the effect of flow profile on the cooling performance of a microscale channel in macro geometry. The flow channel of interest has a nominal gap size of 300 μm, with single-phase water flowing at 2 L/min and subjected to two initial temperatures at 60 and 90°C. The highest cooling capacity, i.e. 1214 W, was recorded in Flow P1, which was 66.8% higher than that of the smooth case. This was resulted from the re-initialization of boundary layer and presence of turbulence in the flow channel. Besides, there was an average 128% increase in the cooling capacity when the initial temperature increased from 60 to 90°C. Larger temperature difference between the incoming flow and heated wall resulted in better heat transfer performance as well.

Keywords: Experiment, Enhanced, Microchannel, Cooling, Convective Heat Transfer

NOMENCLATURE

\[ \dot{Q} \]  Cooling capacity (W)
\[ c_p \]  Specific heat at constant pressure (J/K kg)
\[ m \]  Mass (kg)
\[ \Delta T \]  Temperature difference (K)
\[ t \]  Time (s)

1. INTRODUCTION

In recent decades, the technology is developing rapidly and the power requirement of the electrical devices has also increased significantly. With higher electric power input, the heat dissipated by the electric devices increases proportionally and this has become the key challenge for the conventional cooling methods. The microprocessor which is the leader in the field is predicted by the International Technology Roadmap for Semiconductors in 2010 to generate heat flux up to 160 W/cm² in the near future [1]. The available cooling techniques gradually become unsuitable for the latest devices. In order to tackle the heat issue, various techniques are being proposed and investigated. For example, microchannel heat sink, micropumps, electroosmotic pump, advanced thermoelectric and miniature refrigeration system [2]. Among these, microscale heat sink which is the subject of this paper was introduced by Tuckerman and Pease [3] to cool very-large-scale-integration circuits in 1981.

Since then, numerous research studies were carried out to investigate the flow field and heat transfer characteristic of microscale flow channel. Kandlikar [4] conducted a critical review which provided a historical perspective of the progress made in understanding the underlying mechanism in single-phase liquid flow and two-phase flow boiling processes. Rosa et al. [5] reviewed the experimental and numerical results of microchannel heat transfer. According to their review, heat transfer in microchannels can be suitably predicted by standard theories and correlations, but the scaling effects, for example, entrance effects, viscous heating, and others, which are often negligible in macro-channels may now have a significant effect. Similar conclusions were made by Mishan et al. [6] in their investigations which highlighted the importance to account for entrance effects that are often negligible for standard flows. On the other hand, Lin and Kandlikar [7] noticed the significance of axial heat conduction effects in some cases and they developed a theoretical model to analyze and account for the effects.

The geometry of channel is also one of the key affecting factors in microscale heat transfer. Lorenzini and Morini [8] investigated the laminar, fully developed flow in microchannels with trapezoidal and rectangular cross-sections and rounded corners. Different aspect ratios and non-dimensional radii of curvature of channels were investigated. Park and Punch [9] also carried out experiments using rectangular channel. Deviations were found between the experimental and theoretical heat transfer rates. Meanwhile, Celata et al. [10] studied circular microducts with diameter ranging from 120 to 528 μm and their results showed that Nusselt number decreased with decreasing diameter. Mohammed et al. [11] and Sui et al. [12] conducted numerical simulation of heat transfer enhancement in wavy microchannel heat sink. The former group reported that heat transfer performance of the wavy microchannels was better than straight channel, while the latter group claimed that secondary flow, i.e. Dean vortices could be generated when liquid flowed through the wavy microchannels.
Abouali and Baghemerzhad [13] studied numerically microchannel flows with enhancement using two different types of grooves. The heat removal flux of the enhanced channels increased by 61 and 72%, respectively compared to the smooth channels. Eliamsa-ar and Promvonge [14] found that the heat transfer in the turbulent channel flows over periodic grooves resulted in a considerable enhancement at about 158% over the smooth channel although higher pressure loss was noticed. Conder and Solovitz [15] carried out a comprehensive series of simulations to determine the optimal geometry and Reynolds numbers for thermal performance. Their findings indicated that moderately deep features could result in nearly 75% greater average convective heat transfer coefficient with approximately 35% of pressure penalty at turbulent conditions. In contrast to continuous fins, Lee et al. [16] applied sectional oblique fins in order to modulate the flow in microchannel heat sinks. The average Nusselt number for the heat sink which used water as working fluid was reported to increase as much as 103%, from 11.3 to 22.9.

Motivated by the previous studies and challenges met, the objective of this experiment is to mainly study the effect of channel geometry and flow profile on the heat transfer performance. It will be investigated by measuring the cooling capacity of four types of microscale flow in macro geometry which can be manufactured by conventional method. It is a continuation of the study conducted earlier by Kong and Ooi [17] to measure the heat transfer coefficient in such microscale flow. In the present investigation, a smooth annular flow and flows over three differently-configured grooves were studied. A copper pipe with inner diameter of 20 mm and an insert with mean diameter of 19.4 mm are placed concentrically in order to create flow paths which behave like microchannel. Four different types of insert were fabricated in order to create the four types of microscale flow. The experiments were conducted at a constant flow rate of 2 L/min subjected to two different initial temperatures, i.e. 60 and 90°C.

2. EXPERIMENTAL METHOD

2.1 Test Rig

The experimental test rig used in the previous study [17] is used in the current investigation. Its schematic diagram is presented in Figure 1. The system consists of four flow loops, namely the main loop, air vent loop, pressure relief loop and by-pass loop. The test module which comprises the flow channel of interest was located in the main loop. The air vent loop is used to remove the trapped air bubbles from the test rig. The pressure relief loop is used to prevent the pressure from surpassing the safety limit which is 10 bar gauge in the present setup. The by-pass loop is used to control the amount of water that flows through the test module. The measuring devices in the flow paths include four Type-T thermocouples, four pressure transducers and two volumetric flow meters. Their locations are as shown in Figure 1.

2.2 Test Module

The experimental test module is presented in Figure 2. It consists of various components, namely six heating disks, a copper pipe (hot zone), a stainless steel insert, two insert holders, two polyether ether ketone (PEEK) pipes, and a mica insulating cover. All the components of test module are shown in Figure 3. The length of the channel of interest is 30 mm. Heat is being transferred to the water through the channel of interest. The outer and inner diameters of copper pipe are 32 and 20 mm, respectively. The key dimensions of hot zone and heaters are summarized in Table 1 and 2. In the experiments, four types of insert were used, namely Insert Smooth, Profile 1, Profile 2 and Profile 3. The key features of each insert were summarized and presented in Table 3. The flow associated with the Insert Smooth, Profile 1, Profile 2 and Profile 3 will be denoted as Flow S, Flow P1, Flow P2 and Flow P3 accordingly. The geometrical details of all the inserts are illustrated in Figure 4.

Figure 1: Schematic diagram of the test rig [17].

Figure 2: The experimental test module[18].
Figure 3: Sectional view of test module.

Table 1: Key dimensions of hot zone.

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Volume</td>
<td>$2.017 \times 10^{-5}$ m$^3$</td>
</tr>
<tr>
<td>Outer diameter</td>
<td>32 mm</td>
</tr>
<tr>
<td>Inner diameter</td>
<td>20 mm</td>
</tr>
</tbody>
</table>

Table 2: Key dimensions of heaters.

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Volume (each)</td>
<td>$3.388 \times 10^{-5}$ m$^3$</td>
</tr>
<tr>
<td>Volume (total)</td>
<td>$2.033 \times 10^{-4}$ m$^3$</td>
</tr>
</tbody>
</table>

Table 3: Key features of each insert.

<table>
<thead>
<tr>
<th>Insert Type</th>
<th>Mean Diameter (mm)</th>
<th>Details of the Insert Profile</th>
</tr>
</thead>
<tbody>
<tr>
<td>Smooth</td>
<td>19.4</td>
<td>Smooth surface.</td>
</tr>
<tr>
<td>Profile 1</td>
<td>19.4</td>
<td>The grooves which are perpendicular to the flow direction. They are 2 mm wide and spaced out evenly at a pitch of 2 mm.</td>
</tr>
<tr>
<td>Profile 2</td>
<td>19.4</td>
<td>The grooves which are parallel to the flow direction. They are 1.7 mm wide and spaced out evenly at a pitch of 1.7 mm.</td>
</tr>
<tr>
<td>Profile 3</td>
<td>19.4</td>
<td>The grooves which are 45° to the flow direction. They are 1.7 mm wide and spaced out evenly at a pitch of 1.7 mm.</td>
</tr>
</tbody>
</table>

The measuring devices in the test module include six Type-J thermocouples located in the copper pipe and another four in the heating disks. The location of the thermocouples in the copper pipe is illustrated in Figure 5 and they are used to measure the wall temperature. The thermocouples in the latter component are used for monitoring purposes.

Figure 4: Insert Smooth (Top), Profile 1, Profile 2, and Profile 3 (Bottom).

3. DATA REDUCTION

The experiments were conducted at various settings as presented in Table 4. The copper hot zone was first heated up to the desired temperature values at 60 and 90°C. Once the temperature reached the desired value, the power supply of the heaters was cut off and water was allowed to flow through the test module. The cooling capacity of each experimental setting was deduced by Eq. (1).

$$
\dot{Q} = m_{\text{hot zone}} \cdot c_{p, \text{copper}} \cdot \left( \frac{\Delta T}{\Delta t} \right)_{\text{hot zone}} + m_{\text{heaters}} \cdot c_{p, \text{copper}} \cdot \left( \frac{\Delta T}{\Delta t} \right)_{\text{heaters}}
$$

(1)
The \( m_{\text{hot zone}} \) and \( m_{\text{heaters}} \) in Eq. (1) are obtained by calculating the product of their volume and density at 30°C, which is 8.960 kg/m\(^3\) [19]. The calculated masses are respectively 0.181 kg and 1.822 kg. For the specific heat \( c_p \), it has the value of 385 J/K kg [19]. The temperature gradient of the copper hot zone was taken at its highest value, whereas for the heaters, the temperature gradient at the same time period was used in the calculations.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature of Copper Hot Zone (°C)</td>
<td>60 and 90</td>
</tr>
<tr>
<td>Flow rate (L/ min)</td>
<td>2</td>
</tr>
<tr>
<td>Types of Insert</td>
<td>(1) Insert Smooth</td>
</tr>
<tr>
<td></td>
<td>(2) Insert Profile 1</td>
</tr>
<tr>
<td></td>
<td>(3) Insert Profile 2</td>
</tr>
<tr>
<td></td>
<td>(4) Insert Profile 3</td>
</tr>
<tr>
<td>Gap size (µm)</td>
<td>300</td>
</tr>
</tbody>
</table>

### Table 4: Experimental operating conditions.

#### 4. RESULTS AND DISCUSSION

The validation of the test bed was firstly conducted by Kong and Ooi [17]. From the validation experiments, the analytical and experimental Nusselt number had a discrepancy of 11%. All the experiments in this paper were carried out by utilizing the validated test bed. The time plot of the copper hot zone temperature for all the four types of flow were presented in Figures 6 and 7, respectively for the case of 60 and 90°C. The corresponding cooling capacities were calculated and presented in Figure 8. The results show that the cooling capacity was affected both by the geometry of the flow channel and the initial temperature.

#### 4.1 Effect of Channel Flow Profile

From the experiment results in Figure 3, it can be observed that the cooling capacity of Flow P1 was the highest at both initial conditions. In the case when the initial temperature was at 60°C, the cooling capacity of Flow P1 was 467 W, which was 7.7, 30.3 and 52.6% higher than Flow P3, Flow P2 and Flow S, respectively. Meanwhile when the initial temperature was at 90°C, the difference between the cooling capacities became bigger. The cooling capacity of Flow P1 was calculated to be
1214 W, which was 32.5, 65.2 and 66.8% higher than Flow P3, Flow P2 and Flow S, respectively.

The Reynolds number when the flow rate was at 2 L/min was calculated to be 1344 and this indicates that the flow fell under the laminar regime. In Flow P1, P2 and P3, the grooves and protrusions on the inserts, as shown previously in Figure 4, can be perceived as turbulators which initiate turbulent flow in the laminar regime and therefore resulting in the heat transfer enhancement. However, the grooves and protrusions were configured differently in the three channels and hence the flow fields and extent of heat transfer enhancement were not identical.

Among the three, Flow P1 had the best heat transfer improvement due to the re-initialization of the boundary layer and presence of highly turbulent flow. As shown in Figure 9, the abrupt geometrical change due to the presence of the grooves affected the growth of the boundary layer and caused it to re-initialize. Besides that, circulations occurred within the grooves resulted in better mixing of flow. The arrangement of the grooves in Flow P1 which are perpendicular to the flow direction also disturbed the flow by the greatest extent and induced the highest amount of turbulence as compared to P2 and P3.

As shown in Figure 10, the angle of the grooves in Flow P3 is 45° to the direction of incoming flow. The presence of the grooves induced the flow to deviate away from the incoming flow direction. The flow travelled a longer pathway as compared to Flow P1 and P2. The swirly motion of the flow created turbulence and resulted in better mixing of flow for heat transfer to occur. However, the extent of disturbance and turbulence in Flow P3 was lesser than Flow P1, therefore, the improvement of heat transfer was as well lesser than Flow P1.

The heat transfer improvement in Flow P2 was the least among the three. In Flow P2, the axial direction grooves did not affect the growth of the boundary layer and it continued growing from the beginning of the hot zone area. The grooves introduced very little disturbance and turbulence as compared to those in Flow P1 and P3.

4.2 Effect of Hot Zone Temperature

According to the results presented in Figure 8, the cooling capacity of channel flow at higher initial temperature was higher than at lower temperature. The cooling capacities of Flow S, Flow P1, Flow P2, and Flow P3 at initial temperature 90°C increased by 138, 160, 105, and 112%, respectively. Despite subjected to different hot zone temperature, the Reynolds number and Prandtl number of the water remained nearly unchanged, suggesting that the Nusselt number and heat transfer coefficient of each type of flow remained nearly constant. From Newton’s law of cooling, when the heat transfer coefficient and surface area are maintained, the higher the temperature difference between hot zone and flow water, the higher the heat transfer rate.

5. CONCLUSION

Experiments were performed to investigate the cooling capacity of single-phase flow in 300-μm channels which had different flow profiles and subjected to two different initial conditions at 60 and 90°C. Heat transfer enhancement was attained in Flow P1, P2 and P3. The highest cooling capacity recorded was 1214 W in Flow P1, which was about 66.8% higher than that of the smooth microscale flow. The heat transfer enhancement was resulted from the re-initialization of the boundary layer, presence of circulation and turbulence. On the other hand, the cooling capacity was also greatly affected by the initial temperature of the hot zone. When the temperature increased from 60 to 90°C, the cooling capacities increased by an average of 128% due to the presence of larger temperature difference.

6. REFERENCES


