Heat transfer enhancement in drag-reducing channel flow

Yeo, Keng Hoo

2005

Yeo, K. H. (2005). Heat transfer enhancement in drag-reducing channel flow. Master's thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/5471>

<https://doi.org/10.32657/10356/5471>

Nanyang Technological University

Downloaded on 26 Apr 2023 19:06:40 SGT

Heat Transfer Enhancement in Drag-Reducing Channel Flow

Yeo Keng Hoo

School of Mechanical & Aerospace Engineering

A thesis submitted to the Nanyang Technological University
in fulfillment of the requirements for the degree of
Master of Engineering

2005

ABSTRACT

Since the discovery of the Toms effect (Toms, 1948), polymers as drag-reducing additives have been widely used to reduce the undesired drag that occurs over long-distance transportation of liquids. However, less attention has been focused on this area of research, especially in the case of re-circulating flow systems. This is mainly due to the fact that the polymer additive's capability as a drag reducer can be permanently crippled when subjected to high shear rates or exposed to prolonged period of turbulent flow and repeated heating and cooling. In contrast to the polymer additives, mechanical degradation of surfactant additives when exposed to regions of high shear stress or repeated heating and cooling is only temporary.

In the present study, a two-dimensional water tunnel was designed and fabricated to investigate drag reduction by surfactant additives. Flow development check experiments were performed with different concentrations of surfactant additives to justify the fully developed flow conditions at the test section. The results indicate that the hydrodynamic entry length for water flow without surfactants is very short. For surfactant concentrations of 20, 60, 120 and 160 ppm, the entry lengths were about 300, 500, 1200 and 1400 mm respectively. The entry length for the surfactant concentration of 200 ppm was consistent with that of 160 ppm.

Dilute surfactant solutions at different concentration were used to examine the influence of surfactant additives on the skin friction drag and heat transfer. The maximum achievable drag reductions with the four different surfactant concentrations, namely 30, 70, 80 and 90 ppm, are

about 7, 30, 50 and 54% respectively. The present results showed no heat transfer reduction when 30 ppm of surfactant is added into the flow. With the increase of surfactant concentration to 90 ppm, heat transfer was reduced by about 60%. The critical Reynolds numbers for heat transfer reduction were noted to be higher with higher surfactant concentration (e.g. at 12000 and 14000 respectively for surfactant concentrations of 70 and 80 ppm) in the flow.

To enhance heat transfer rate in the drag reducing fluid, a row of low-profile mixers was installed in the spanwise direction on the flat heating surface. These mixers were used to create an array of near-wall longitudinal spiraling vortices that help to disrupt the orientation of the orderly aligned micelle structures of the surfactant in the boundary layer and also to promote large-scale bulk motion of fluid between the main stream and the near-wall region. The effect of the low-profile mixer on heat transfer enhancement was examined for surfactant concentrations of 90 ppm at three different Reynolds numbers, namely, 7000, 12000 and 16200. The result for $Re = 7000, 12000$ and 16200 show that the local Nusselt number is about 190%, 200% and 190% higher than that in 90 ppm without the use of the vortex generators, respectively. The increase in heat transfer rate may be related with the longitudinal vortex generated by the low-profile-mixers. Deterioration of heat transfer enhancement by the longitudinal vortices was also noted. It was found that the enhancement of the low-profile-mixers to the heat transfer rate is not applicable for all magnitudes of x/H . When $x/H > 7$, the heat transfer enhancement started to decline. The location of the onset of this decline shifts further downstream with an increase in Reynolds number.

ACKNOWLEDGEMENT

I would like to thank both my supervisors Dr. Zhou Tongming and Dr. Leong Kai Choong for their guidance and utmost care that they have provided me to complete my M.Eng research.

I would also like to extend my appreciation to Dr Nguyen Nam Trung, Dr Yeo Joon Hock and the technicians in the Fluid Mechanics Laboratory of the School of MAE, NTU for support in the construction of the experimental facility.

LIST OF FIGURES

Figure 1.1 – Globular micelles	3
Figure 1.2 – Schematic illustration of rod-like micelles network (Yang, 2002)	4
Figure 1.3 – Instantaneous velocity fluctuation vectors in water at $Re = 2\,1000$ (Kawaguchi <i>et al.</i> , 2002)	6
Figure 1.4 – Instantaneous velocity fluctuation vectors in drag-reducing flow at $Re = 2\,1000$ (Kawaguchi <i>et al.</i> , 2002)	7
Figure 1.5 – Flow diagram of a district cooling system in Singapore (Tien, 2001)	10
Figure 1.6 – Side view of the wire mesh installation position (Li <i>et al.</i> , 2001)	13
Figure 1.7 – Isometric view of vortex generator orientation (Eschenbacher <i>et al.</i> , 2001)	14
Figure 1.8 – Cross sectional view of the single-pipe countercurrent heat exchanger (Fossa and Tagliafico, 1995)	15
Figure 1.9 – Diagram of spirally fluted tube (Qi <i>et al.</i> , 2001)	16
Figure 2.1 – Photographs of the constructed water tunnel facility	20
Figure 2.2 – Schematic of experimental facility	22
Figure 2.3 – Overall external dimensions of the honeycomb rectifier	23
Figure 2.4 – Rendering view of the entrance section	23
Figure 2.5 – Isometric view of designed rectangular duct	25
Figure 2.6 – Top view of the 6-m-long water tunnel	26

Figure 2.7 – Embodiment of the pressure-tap-point	26
Figure 2.8 – End view of the rectangular duct	27
Figure 2.9 –A sample of the entrance length approximation	27
Figure 2.10 – Rendering view of the exit section	29
Figure 2.1 1 – Rendering view of the reservoir tank	30
Figure 3.1 – Flow development diagram	39
Figure 3.2– Schematic of differential pressure measurement setup	43
Figure 3.3 – Schematic of the drag experimental setup	44
Figure 3.4 – Schematic of the test section	46
Figure 3.5 – Thermocouple distribution across the heating surface	49
Figure 3.6 – Thermocouple embodiment diagram	50
Figure 3.7 – Side view of the heater insulator	50
Figure 3.8 – Local convective heat transfer coefficient over the heating surface	52
Figure 3.9 - Equivalent thermal circuit of the heating plate	54
Figure 3. 10 – Local bulk temperature over the heating surface	56
Figure 3.1 1 – A row of low-profile mixers	57
Figure 3.12– Dimensions of the low-profile mixers	59
Figure 3.13– Rotational direction of vortices viewed from the end of test section	59
Figure 4.1 – Pressure drop along the water tunnel with 0 ppm of surfactant. The straight line represents the least-square linear regression to the experimental data.	61

Figure 4.2 – Pressure drop in the streamwise direction with 20 ppm of surfactant.	62
The straight line represents the least-square linear regression to the experimental data.	
Figure 4.3 – Pressure drop in the streamwise direction with 60 ppm of surfactant.	62
The straight line represents the least-square linear regression to the experimental data.	
Figure 4.4 – Pressure drop in the streamwise direction with 120 ppm of surfactant.	63
The straight line represents the least-square linear regression to the experimental data.	
Figure 4.5 – Pressure drop in the streamwise direction with 160 ppm of surfactant.	64
The straight line represents the least-square linear regression to the experimental data.	
Figure 4.6– Pressure drop in the streamwise direction with 200 ppm of surfactant.	65
The straight line represents the least-square linear regression to the experimental data.	
Figure 4.7 – Fanning friction factor at various Reynolds numbers	69
Figure 4.8 – Percentage of drag reduction at various Reynolds numbers	70
Figure 4.9 – Averaged Nusselt number at various Reynolds numbers in tap water	74
Figure 4.10 – Instantaneous velocity measurement in water using PIV at $Re = 21000$ (Kawaguchi <i>et al.</i> , 2002). (a) Instantaneous velocity fluctuation vectors;	
(b) Contour map of instantaneous velocity distribution.	75
Figure 4.11 – Streamwise distribution of local Nusselt number in water and surfactant solutions at $Re = 7000$	78

Figure 4.12 – Streamwise distribution of local Nusselt number in water and surfactant solutions at $Re = 12000$	80
Figure 4.13 – Streamwise distribution of local Nusselt number in water and surfactant solutions at $Re = 16200$	81
Figure 4.14 – Streamwise distribution of local Nusselt number in water and surfactant solutions at $Re = 18500$	82
Figure 4.15 – Photographs of the longitudinal vortices rotating in counter clockwise direction at time intervals $t = 0$ and 20 seconds.	86
Figure 4.16 – Photograph of longitudinal vortices rotating in counter-clockwise direction	88
Figure 4.17 – Effectiveness of the low-profile-mixers at low Reynolds number	89
Figure 4.18 - Streamwise distribution of local Nusselt number with low-profile mixers ($Re = 7000$)	91
Figure 4.19 - Streamwise distribution of local Nusselt number with low-profile mixers ($Re = 2000$)	92
Figure 4.20 - Streamwise distribution of local Nusselt number with low-profile mixers ($Re = 6200$)	93
Figure 5.1 – Conceptual design of the micro-actuators	102
Figure 5.2 – Precision pressure regulating system for micro-actuators	102
Figure 5.3 – Three-dimensional view of the proposed water tunnel	104
Figure 5.4 - External dimensions of the proposed water tunnel	107

LIST OF TABLES

Table 2.1 – Computation of minor losses in the re-circulating loop	34
Table 2.2 – Summary of pressure drop in the re-circulating loop	35
Table 4.1 – Contributions from various variables to the uncertainty of C_f	72
Table 4.2 – Contributions from various variables to the uncertainty of Nu	84
Table 5.1 – Summary of the main feature of each component	106
Table 5.2 – Comparison of the proposed water tunnel facility with those used by other researchers	108

NOMENCLATURE

C_f	Fanning friction factor
H	height of the water tunnel (m)
h_x	local heat transfer coefficient ($\text{W.m}^{-2}.\text{K}^{-1}$)
$\bar{h}$	average heat transfer coefficient ($\text{W.m}^{-2}.\text{K}^{-1}$)
I	heater input current (A)
k	thermal conductivity of water ($\text{W.m}^{-1}.\text{K}^{-1}$)
L	overall length of the heating surface (m)
Nu_x	local Nusselt number
ppm	part per million mass
Pr	Prandtl number
q_s	local heat flux (W.m^{-2})
Re	Reynolds number based on tunnel height = $\frac{U_b \cdot H}{\nu}$
Re_{cri}	critical Reynolds number for drag and heat transfer reduction
U_b	bulk velocity (m.s^{-1})
T_{bx}	local bulk temperature of the flow above the heating wall (K)
T_{inlet}	inlet bulk temperature (K)
T_{outlet}	outlet bulk temperature (K)
T_{sx}	local surface temperature of the heating wall (K)
V	heater input voltage (V)
W	width of the rectangular duct (m)
x	distance from the leading edge of the heating surface (m)

Greek Symbols

ν Kinematic viscosity ($\text{m}^2.\text{s}^{-1}$)

μ Dynamic viscosity (N.s.m^{-2})

ρ density of water (kg.m^{-3})

τ_w shear stress on the wall surface of the rectangular duct (N.m^{-2})

ΔC_f uncertainty associated with C_f

ΔNu uncertainty associated with Nu

Δp pressure drop in water tunnel (Pa)

Δx distance measured between pressure tapping points for drag measurement (m)

CONTENTS

ABSTRACT		i
ACKNOWLEDGEMENT		iii
LIST OF FIGURES		iv
LIST OF TABLES		viii
NOMENCLATURE		ix
CONTENTS		xi
CHAPTER ONE	Introduction	1
1.1	Background	1
1.2	Application of drag-reducing flow	8
1.3	Heat transfer enhancement in drag-reducing flow	11
1.4	Possible passive heat transfer enhancement in drag-reducing flow	17
1.5	Objectives and Scope	17
1.6	Layout of the report	18
CHAPTER TWO	Development of Experimental Facility	20
2.1	Overview of the water tunnel facility	20
2.2	Entrance section	22
2.3	Long rectangular duct	24
2.4	Exit section	28
2.5	Reservoir tank	29
2.6	Water pumping system i	30

CHAPTER THREE	Experimental Overview	36
3.1	Outline	36
3.2	Preparation of the surfactant solutions	38
3.3	Data reduction	38
3.4	Experimental procedure and set-up for flow development check	39
3.5	Experimental set-up and definition of drag reduction	43
3.6	Experimental set-up and definition of heat transfer reduction	47
3.7	Experimental procedure and set-up for heat transfer enhancement	56
CHAPTER FOUR	Results and Discussion	60
4.1	Flow development checks for various surfactant concentrations	60
4.2	Effect of surfactant on drag	66
4.3	Uncertainty analysis of Fanning friction factor	71
4.4	Heat transfer without surfactant	73
4.5	Heat transfer with surfactant	76
4.6	Uncertainty analysis of Nusselt number	83
4.7	Heat transfer enhancement	85
CHAPTER FIVE	Conclusions and Recommendations for Future Work	96
5.1	Conclusions	96
5.2	Recommendations for future work	98

5.2.1	Design of the micro-actuators for active heat transfer enhancement	100
5.2.2	Proposed experiment for the micro-actuators	103
5.2.3	Proposal for new water tunnel facility	103
REFERENCES		109
Appendix A – Dimensions of contracting nozzle		120
Appendix B – Dimensions of reservoir		121
Appendix C – Water pump performance graph		122
Appendix D – Detail calculation of heater losses		123
Appendix E – Lamination clamping device		125
Appendix F – Publications arising from this thesis		126

CHAPTER ONE: Introduction

1.1 Background

The reduction of frictional wall drag by adding small amounts of polymer additives in turbulent pipe flow is widely recognised by researchers. It had been proven that the introduction of a small quantity of soluble polymers with long chains and relatively linear structure and unique properties, significantly increased the flow rate under a constant driving pressure drop or decreased the pressure drop under constant flow conditions. This effect is commonly known as “Toms effect” (Toms, 1948).

Recently the use of polymers as drag-reducing additives has received less attention, especially for re-circulating flow systems. This is due mainly to the fact that the polymer’s capability as a drag reducer can be permanently crippled when subjected to high shear rates or when it is exposed to prolonged periods of turbulent flow and repeated heating and cooling regions. Polymers also do not have the ability to recover from repeated mechanical and thermal degradation cycles.

In contrast to polymers, the mechanical degradation of surfactant additives in regions of high shear stress is only temporary. Surfactants have the ability to restructure its rod-like microstructures and thus resume its own drag-reducing capability when the level of shear stress decreases. The orientation of the orderly large-scale rod-like micelle structures, which promote the drag reduction phenomenon, is recoverable in the order of seconds even after being disrupted.

Another characteristic of this drag-reducing agent is that when the surfactant concentration in the flow is high enough, the drag reduction will reach an asymptotic maximum. No further drag reduction is attainable with further increase of surfactant concentration. Its micelle microstructure becomes more robust to turbulent disturbance. As a result, a higher shear stress is required to degrade its drag-reducing capability. Thus, a system with a high concentration of surfactant is able to maintain its drag and heat transfer reducing ability at a larger Reynolds number.

Many varieties of surfactant additives have been investigated experimentally (Myska and Mik, 2003; Gasljevic and Matthys, 1993). The results have shown a common feature - with a small mass concentration of additives (e.g. a few milli Molarity) dissolved into a systems' working media, a dramatic decrease of drag and heat transfer is achievable.

According to Candau and Oda (1998) and Menger and Persypkin (2001), surfactants that are able to form rod-like micelles can be classified into three types, namely, cationic, zwitterionic and anionic. The drag-reducing effectiveness of the three different types of surfactant was studied previously. According to Yang (2002), a narrow temperature range for effective drag reduction limits the usefulness of the non-ionic surfactants; precipitation caused by bivalent ion (Ca, Mg) in hard water limits the application of the anionic soap surfactants; a wide temperature range and no sensitivity to the bivalent ion make the cationic surfactants more useful as compared to the other two types.

Chou *et al.* (1989) also found that by increasing the length of the alkyl chain in a cationic surfactant, the upper temperature limit for effective drag reduction would increase. Thus, the

most widely studied surfactant system is the cationic surfactant combined with counter-ions. This combination is well accepted by many researchers (Candau and Oda, 2001; Imai and Shikata, 2001; Kim and Yang, 2000; Magid *et al.*, 2000; Lin *et al.*, 2001; Raghavan and Kaler, 2001 ;Raghavan *et al.*, 2002) as a model rod-like micelle in practical applications. Typically, a surfactant molecule dissolved in water consists of a combination of a water-soluble (hydrophilic) and a water-insoluble (hydrophobic) part. The hydrophobic part is a long chain alkyl rest, whereas the hydrophilic part is ionizable, polar, polarizable or suitable for forming hydrogen bridges. When sufficient amounts of surfactant molecules are presented in pure water, tens or even hundreds of these molecules will be aggregated together by the attractive force between the alkyl chains, to form a globular micelle structure as shown in Figure 1.1.

The inner part of a globular micelle can be thought of as being a liquid droplet consisting of hydrocarbon chains that fill the void of the sphere. At the outer surface of the micelle sphere, the hydrophilic head groups are in contact with the solvent water responsible for the solubility of the micelles. As shown in Figure 1.1, the radius of the globular micelle structure approximately corresponds to the length of an alkyl chain in a monomer surfactant.

Figure 1.1 – Globular micelles

With an increase of surfactant concentration in water, the globular micelle structure will grow in size. The micelle size will grow to an extent when the volume of the micelle is fully occupied by the hydrocarbon chain. From then on, the size will not change. Instead, the number of micelles will increase with additional surfactant molecules in water. However, by adding some salt as counter-ions into the surfactant solution, the electrolytic repulsion forces of the head groups can be suppressed. As a result, the surfactant molecules can be packed more closely and will start to form rod-like micelles as shown in Figure 1.2.

Figure 1.2 – Schematic illustration of rod-like micelles network (Yang, 2002)

With the presence of counter-ions and with the increase of surfactant concentration, the length of the rod-like micelle will increase to a geometry that is similar to a medium-sized flexible fibre. Once the rod-like micelle is developed to this stage, its mechanical properties are similar to medium-sized flexible fibres.

In addition to the drag-reducing effect, another noticeable effect, which arises with the introduction of surfactant additives, is the heat transfer reduction that occurs between the solid boundary and the flowing fluid. According to previous studies by Gasljevic and Matthys (1993) and Roethig and Matthys (1988), the introduction of certain drag-reducing additives can result in a heat transfer reduction that is much larger than its drag reduction. Gasljevic and Matthys (1993) showed that in a fully developed pipe flow, the asymptotic drag reduction was about 80% while the heat transfer reduction was about 90%. This dramatic heat transfer reduction would be useful in the distribution pipeline of a district cooling system because it minimises heat losses and hence lower the requirement on insulation along the pipeline. However, this phenomenon is undesirable in some cases, such as in the heat exchanger and the chiller.

In order to understand the phenomenon of heat transfer reduction in drag-reducing flow, it is necessary to first understand the fundamentals of convection heat transfer. This mode of heat transfer between a solid surface and a flowing fluid is comprised of two mechanisms, namely, the random motion of fluid molecules and the bulk motion of fluid particles from the low-speed region close to the wall into high-speed region of the main stream. Although the total heat transfer is taken as superposition of energy transferred by the two mechanisms, the main contribution in the case of pure water is by the bulk motion of fluid (mixing) in the channel.

Kawaguchi *et al.* (2002) performed an experimental study in a channel with tap water and surfactant solution using a double-pulse particle image velocimetry (PIV) system. Pictures of instantaneous velocity field taken in water flow and drag-reducing flow are shown in Figures. 1.3 and 1.4, respectively. In these figures, the direction of the main flow is from left to right

and the height of the figure corresponds to the height of the channel. The bulk mean velocity in both figures is subtracted from all the vectors. As a result, vortices moving with the bulk mean velocity can be seen in Figure 1.3. However, in Figure 1.4 there are no vortices when surfactant is introduced. The only mechanism available to transfer the heat energy from the solid boundary into the flowing media is the random motion of the fluid molecules near the heating surface. Based on this observation, Kawaguchi *et al.* (2002) concluded that a reduction in bulk motion between layers will result from an introduction of surfactant additives in the flow.

Figure 1.3 – Instantaneous velocity fluctuation vectors in water at $Re = 21000$ (Kawaguchi *et al.*, 2002)

Another hypothesis about this heat transfer reduction is that the thickening of the thermal boundary layer near the wall may increase the thermal resistance between the wall and the flowing fluid. Kawaguchi *et al.* (1997) demonstrated the difference in the thermal boundary layer thickness between the dilute surfactant solution and its solvent, with the use of a fine

wire K-type thermocouple probes of 12.5 μm -diameter. They concluded that the thickening of the thermal boundary layer indeed happened when surfactants are introduced.

Figure 1.4 – Instantaneous velocity fluctuation vectors in drag-reducing flow at $\text{Re} = 21000$

(Kawaguchi *et al.*, 2002)

The effectiveness of the surfactant solution at a particular concentration is limited to certain range of Reynolds numbers. When Reynolds number increases up to a critical value, due to the shearing effect of the flow, the rod-like micelles start to degrade through the reduction of its length. As a result, drag reduction starts to decline. At this point, if the Reynolds number is increased further, the rod-like micelle will be completely destroyed by the shear stress of the flow, leading to a total breakdown of drag reduction capability. However, the loss of aggregation is only temporary. By decreasing the Reynolds number below that critical value, the drag reduction effect will be recovered immediately. The decrease of Reynolds number corresponds to the reduction of the shear stress that promotes the “self-repairing” capability of the micelle.

1.2 Application of drag-reducing flow

Feasibility studies were conducted by Gasljevic and Matthys (1993) and Li *et al.* (2000) to support the implementation of the drag-reducing additive and to optimise the design of a thermal distribution system by reducing its electrical power consumption. This can be achieved by introducing a small amount of the drag-reducing additive into the circulating flow to alter the fluid dynamic characteristics. With drag-reducing additives in the flow, both drag and heat transfer along the thermal distribution system's transmission pipe are reduced significantly.

A district cooling system is a means of distributing cooling energy from a centralised production plant to end-users. The cooling energy is delivered as chilled water via a network of underground pipes to offices, industrial or residential buildings to cool the indoor air of the buildings. In a district cooling system, the output of a centralised cooling plant is enough to meet the cooling-energy demand by dozens of buildings. It is an excellent candidate for application of drag reducing surfactant additives.

The advantages of the district cooling system include better utility of system capacity, better management of heat rejection, giving end-users convenience and comfort with round-the-clock thermal management, and eliminating noise and vibration that are produced by conventional compressors. In addition, space that are occupied either by the cooling tower or by the condensing units of conventional cooling systems can be freed for other purposes.

The use of this technique for the purpose of air conditioning is not new. Large district cooling systems were built as early as 1930s in Rockefeller Center and the United States Capitol

Complex. Currently, this technique to condition indoor air of buildings is widely used in downtown business districts and institutional settings such as college campuses in the United States (e.g. Century City, CA; Bunker Hill, downtown Los Angeles, CA; Boeing, Huntington Beach, CA; Disneyland Hotel, Anaheim, CA; College Park, Fullerton, CA; Dreamworks, Glendale, CA; Santa Fe International, Alhambra, CA and San Antonio, Texas and etc.). In Singapore, one of the business parks (Changi Business Park) located in the eastern part of the island has also adopted this technology to conserve space and energy in its tender premises (refer to Figure 1.5).

The popularity of the district cooling system will lead to issues such as the overall performance of the system. Chilled water in the district cooling system is transported from a centralised cooling plant to end users through a network of distribution pipelines that stretched for several kilometres (e.g. 3.5 km, according to report by Tien, 2001). The distribution pipeline network in the district cooling system is relatively large in scale. Thus, during the transportation of the chilled water, drag that exists in the piping systems is significantly large. The maximum distance or scale that the centralised cooling plant can supply or reach effectively is restricted by the drag in the flow. Presently, there are two approaches to overcome this problem. The first approach is by designing the main distribution pipeline with a bigger pipe diameter. The other approach is by using a larger pumping system to overcome the flow resistance. However, both methods are not cost effective. The use of larger pipes will eventually lead to the use of thicker insulation to reduce thermal losses along the transportation pipeline. A larger distribution pumping system implies a deployment of higher capacity water pumps. These not only incur a higher initial cost, but also incur a higher maintenance cost. The electrical power consumption by the

pumps also needs to be taken into consideration. A larger pump means that more pumping energy is required to overcome the drag along the distribution pipeline in order to deliver water to end users. As a result, higher initial investment cost and higher electrical cost will surely discourage potential stakeholders.

Figure 1.5 – Flow diagram of a district cooling system in Singapore (Tien, 2001)

Although the implementation of the drag-reducing additives in a thermal distribution system is expected by many researchers to be relatively simple and inexpensive, it is still not widely accepted. This is mainly due to the issue of heat transfer reduction effect, which occurs in the heat exchanger, undesirably promoted by the introduction of drag-reducing additive. Unless a strategy to manipulate heat transfer at the desired location is established at low cost, it can

only be used in applications where heat transfer performance is not so important. Hence, in order to fully implement this technique to reduce the pressure losses and to control the heat transfer phenomenon at desired locations in the district cooling system, a method to enhance the heat transfer effect in the drag-reducing flow has to be developed.

1.3 Heat transfer enhancement in drag-reducing flows

Heat transfer reduction coupled with drag reduction is desired at certain locations of a district cooling system, especially in the water transportation system. This effect is desired to minimise temperature increase along the transmitting pipeline.

In contrast, heat transfer reduction is undesired at certain locations such as the chiller and heat exchanger, where heat transfer is encouraged to increase the overall performance of the system. Thus, a technique to manipulate heat transfer at desired locations is required before the drag-reducing technique can be implemented.

Typically, passive and active approaches are used to enhance heat transfer between a solid boundary and the fluid. The passive approach does not require additional energy, while the active approach involves additional input of energy.

The enhancement of heat transfer between a solid boundary and flowing fluid has received extensive interest in the previous decades (Bergles, 1981; Gupta, 1985). Most of the researchers focused on water as the working fluid through the use of passive enhancement techniques. Some of the well known passive techniques by Bergles (1981), Gupta (1985) and Webb (1995) are used to increase the heat transfer surface area and/or the convective heat

transfer coefficient. The extension of the heat transfer surface area is achieved by using fins or grooves. The increase of the convective heat transfer coefficient is attained either by mixing the main flow with the flow in the wall region, reducing the flow boundary layer thickness, creating a rotation or raising the turbulence intensity.

In the aspect of drag-reducing flow, the heat transfer enhancement technique proposed previously for water may not be sufficient to increase the heat transfer since the flow characteristics in a non-Newtonian fluid (surfactant solution) are totally different from a Newtonian fluid (e.g. pure water). As a result, the heat transfer enhancement mechanism that transports momentum in the case of water may not be the same as the transport of heat in the case of drag-reducing flow (Gasljevic and Matthys, 1993). Thus, typical enhancing devices in drag reducing flow may perform differently (Matthys, 1997).

In recent years, many researchers (Li *et al.*, 2001; Eschenbacher *et al.*, 2001; Fossa and Tagliafico, 1995; Qi *et al.*, 2001) have begun to realise this issue and paid much attention on the investigation of an enhancement technique for the heat transfer in drag-reducing flows. A passive heat transfer enhancement technique was investigated by Li *et al.* (2001). The investigation involved the use of wire mesh and was conducted in a 6-m long water tunnel with a rectangular cross-section. At the bottom wall of the test section, a thin heating plate supplied a constant and uniform heating flux. The wire mesh was installed up-stream of the heating section as shown in Figure 1.6. The wire mesh served as a high shear stress inducer to destroy the rod-like micelle structures in the dilute surfactant solution. Two types of meshes, namely type-A and type-B, were used in their tests. Both types have the same wire diameter of 0.19 mm with different opening rates. The opening rates for type-A and type-B, are 7.6

wires per centimetre and 19.2 wires per centimetre, respectively. They found that this method for enhancing heat transfer is not effective in either high concentration surfactant solution or at low Reynolds number flows. According to their study, the relationship between pressure drop and the heat transfer improvement is not linear. It is therefore possible that the design of the wire mesh can be optimised to achieve better heat transfer enhancement.

Figure 1.6 – Side view of the wire mesh installation position (Li *et al.*, 2001)

Eschenbacher *et al.* (2001) used a single wing-type vortex generator to generate a longitudinal vortex in the drag-reducing flow. The vortex generator with a delta-shaped winglet was made of brass and attached to the bottom surface of the test section of a rectangular duct. Its angle of attack against the main flow was fixed at 30° as shown in Figure 1.7. The height of the trailing edge was $0.5H$ and the base length was set to $2H$, where H refers to the height of the water tunnel. The longitudinal vortex that was created by this generator was found to be able to recover the reduced wall heat transfer coefficient of the surfactant solution completely up to the value of pure water.

Figure 1.7 – Isometric view of vortex generator orientation (Eschenbacher et. al., 2001)

An experimental analysis on the heat transfer effect across an enhanced surface heat exchanger with drag-reducing flow was carried out by Fossa and Tagliafico (1995). Their tests were performed on an experimental facility equipped with a single-pipe type counter-current heat exchanger of 0.6 m in length, in which chilled drag-reduced solution flows in the external annular duct. The outer surface of the heat exchanger is adiabatic while the inner surface can be either heated by an electrical resistance or by primary hot water flowing internally. The annular passage for the chilled drag-reduced solution consists of an outer pipe that has an internal smooth surface and an inner pipe that can have either a smooth, finned or grooved outer surface (Figure 1.8). According to their study, the heat transfer rates in the drag-reducing flow at different surface conditions are always lower than the heat transfer in pure water flow. Hence, this technique to enhance the heat transfer rate is not very effective.

Figure 1.8 – Cross sectional view of the single-pipe countercurrent heat exchanger (Fossa and Tagliafico, 1995)

Another work in the aspect of the heat transfer enhancement in drag-reducing solution was carried out by Qi *et al.* (2001). In their study, a fluted tube-in-tube heat exchanger with four starts, as shown in Figure 1.9, was used to enhance the heat transfer in drag-reducing solution. The outer surface of this heat exchanger was well insulated to prevent heat loss. While chilled water or chilled drag-reducing solution is flowed through the annulus of the heat exchanger, shear stress is induced with the aid of the embedded spiral starts. As a result, the super-ordered micelle structure in the surfactant solution is broken down. They showed that the heat transfer capability was as high as 1.4 times of water. Thus, this method of heat transfer enhancement is more promising than other methods.

Figure 1.9 – Diagram of spirally fluted tube (Qi *et al.*, 2001)

Numerous experimental studies have been carried out to actively control turbulent flow and its concomitant transport phenomena such as friction drag and heat transfer. Ahuja and Bunin (1984) and Hsiao *et al.* (1990) used acoustic excitation for control. The most successful to date has been the so-called "synthetic jet" first demonstrated by Ingard (1953), and popularised in recent years by Glezer (1999). By using the synthetic jet technique, Glezer (1999) carried out separation control and virtual-shaping of airfoils. Flow control investigations were also performed numerically by many investigators. Kim (1992) proposed that the near-wall streamwise vortex is the most relevant turbulence structure from the perspective of drag reduction in turbulent boundary layers. This point of view is supported by the observation that streamwise vortices have been found to be responsible for both "ejection" and "sweep" events of the bursting process (Robinson, 1991). Recent studies have also shown that the high skin-friction regions in turbulent boundary layers are closely related to the near-wall streamwise vortices (Kravchnko *et al.*, 1993; Choi *et al.*, 1994). Choi *et al.* (1994) showed that a significant drag reduction is possible when the surface boundary condition is modified to suppress the near-wall streamwise vortices. They also relate the drag-reducing mechanism on a riblet surface to the restricted interaction between the riblet and streamwise vortices.

Recently, efforts have been made to use micro actuators to control vortices at separation lines. Liu *et al.* (1999) used a micro-machined permanent magnet as a micro flap, which works as disturbance at the separation line. The micro actuator is a micro balloon made of micro machined silicone rubber.

Micro synthetic jet is another control concept, which does not expose moving parts to the main flow. A synthetic jet device has a cavity with a flexible actuating membrane on one side and an orifice on the other side. If the membrane is actuated, the fluid is pumped in and out of the cavity through the same orifice. This fluid movement forms a series of vortices, which propagate away from the orifice. This principle allows the actuator to transfer momentum into the surrounding fluid without a net mass injection. Therefore, this device is called a zero mass-flux actuator.

1.4 Possible passive heat transfer enhancement in drag-reducing flow

The use of low-profile vortex generators to control boundary-layer separation had been studied in detail by Lin (2002). The author summarised the definition of low profile vortex generators and the key results obtained from research performed within the past one and a half decades. The practical advantages of low-profile vortex generators, such as their inherent simplicity and low device drag, are demonstrated to be critically important for many applications.

1.5 Objectives and Scope

Due to the importance of drag-reduction in engineering applications, the present study will concentrate on drag-reduction using surfactant additives in a two-dimensional channel flow.

With the introduction of surfactants in the fluid, the heat transfer rate will also be reduced. This latter effect will not always be advantageous, especially in the heat exchanger of a district cooling system. To improve the overall performance of the district cooling system, techniques should be implemented to enhance the heat transfer rate locally in the heat exchanger. Thus, the main objective of this project is to increase the heat transfer in a channel flow by using passive methods, such as low-profile vortex generators. Feasible active methods to enhance the heat transfer will also be mentioned in the last chapter.

Since there is no existing test facility in Nanyang Technological University for the purpose of drag-reduction and heat transfer reduction experiments, a two-dimensional water tunnel was developed. After which, this newly constructed water tunnel facility will be used:

- to examine drag reduction effect in a channel flow by using surfactant additives,
- to examine the reduction of heat transfer rate in drag reducing flows,
- to examine the dependence of the drag and heat transfer reduction using surfactant additives on different flow conditions, and
- to increase heat transfer rate in a channel flow by using passive approaches.

1.6 Layout of the report

After a brief introduction and literature review on drag-reducing, its application and various heat transfer enhancement techniques in Chapter 1, the detailed design of the water tunnel experimental facility and the function of its sub-components are presented in Chapter 2.

Chapter 3 covers an overview of the experimental set-up and procedures. The experiments include flow development check, the examination of drag reduction, the heat transfer

reduction and the enhancement of heat transfer in the drag-reducing flow. The properties of the surfactant additives that are used to facilitate the experimental study is given in this chapter. The relation derived to calculate Reynolds number, drag and heat transfer reductions is also included in this chapter.

Chapter 4 presents results obtained for the flow development check, the drag reduction, the heat transfer reduction and the enhancement of heat transfer in the drag-reducing flow experiments. These results are calculated from measured quantities such as temperatures, volumetric flowrate and pressure drop in the streamwise direction. The results are justified by comparing with these obtained by other researchers.

Conclusions for the present study are stated in Chapter 5. In addition, suggestions for future work are mentioned in detail, especially the design of the experimental set-up and procedures for active enhancement of heat transfer. Design of the new water tunnel is also given in the last part of Chapter 5.

CHAPTER TWO: Development of the Experimental Facility

2.1 Overview of the water tunnel facility

In this chapter, the development of the water tunnel facility (as shown in Figure 2.1) that will be used to study heat transfer and drag reduction in the drag-reducing flow is presented. The water tunnel facility has a rectangular test section of 0.5 m (width) $\times$ 0.05 m (height) $\times$ 6 m (length) (with an aspect ratio of 10:1). It is designed to achieve steady flow conditions at high volumetric flowrates. The reason for using a rectangular duct as the test section is to allow possible installation of a flat heating surface. The flat heating surface is essential because the final goal of this research is to enhance convective heat transfer in drag-reducing flow. Furthermore, with a flat heating surface, the implementation of various heat transfer enhancement techniques will become easier.

Figure 2.1 – Photographs of the constructed water tunnel facility

According to Gasljevic and Matthys (1997), in a duct flow, the length of the hydrodynamic developing region increases with an increase in the surfactant concentration. The present water tunnel facility has a 6-m-long rectangular duct, which is long enough to achieve fully developed flow conditions in the test section. The walls of the rectangular duct are made of transparent polycarbonate sheets to allow the use of Laser Doppler Anemometry (LDA) and Particle Image Velocimetry (PIV) for measurements of the flow field in the future.

The water tunnel facility is a re-circulating flow loop that consists of a water pumping system, a reservoir tank, an entrance section, an exit section and a 6-m-long rectangular duct as shown in Figure 2.2. The entrance section embodied a two-dimensional contracting nozzle and an array of closely packed 5.6-mm-diameter plastic tubes to stabilise the flow by reducing the large eddy motion before issuing the flow into the rectangular duct. The duct contains two regions, namely, a hydrodynamic developing region and a fully developed region at downstream of the duct.

The test section accommodates a flat heating surface for heat transfer experiments. At the exit section of the rectangular duct, an inclined plate is installed to deflect the pressure wave from the reservoir tank away from the duct. The reservoir tank, which is located at the end of the duct, stores most of the liquid in the facility. It also acts as a mixing tank for the introduction of additives. The water pumping system, which consists two units of water pump, is controlled by two frequency inverters. A Rotor-X paddlewheel flowmeter (+GF+ SIGNET) is installed along with the supply pipeline, approximately 4 m away from the water pumps to measure the volume flowrate in the re-circulating flow loop.

Figure 2.2 – Schematic of experimental facility

2.2 Entrance section

The primary function of the entrance section is to stabilise incoming water from the water pump. The incoming water is very unstable and highly turbulent. The connection between the 6-m-long rectangular duct and the entrance section is designed with a smooth contracting nozzle to guide the flow into the duct. A honeycomb rectifier, which consists of plastic tubes with an inner diameter of 5.6 mm and 125-mm-long, is installed after the contracting nozzle to further reduce the eddy motion of the flow (Figure 2.3).

Figure 2.3 – Overall external dimensions of the honeycomb rectifier

The secondary function of the entrance section is to serve as a surge tank. The role of the surge tank is to damp the pressure wave created when there is a sudden change of water flowrate in the piping system. An air vane that serves as a damping device is installed above of the entrance section. A 400 mm by 200 mm air-tight cover is installed on top of the entrance section to allow accessibility by the researcher for modification and maintenance. The whole section is made of polyvinyl chloride sheet. A 3-D view of the entrance section is presented in Figure 2.4. Its dimensions are illustrated in Appendix A.

Figure 2.4 – Rendering view of the entrance section

2.3 Long rectangular duct

In order to eliminate the influence from the side walls of the duct, the cross section of the duct must have an aspect ratio as large as possible. Thus, a cross-sectional geometry with an aspect ratio of 10:1 is chosen as one of the design criteria of the duct (Figure 2.5). A row of pressure taps shown in Figure 2.6 is embedded on the top wall of the rectangular duct. The pressure taps are spaced equally from one another at a distance of 300 mm. The embodiment of the pressure taps is shown in Figure 2.7.

The whole 6-m-long rectangular duct is made of 10-mm-thick transparent polycarbonate sheet. Polycarbonate is well known to possess better impact resistance and toughness than any other engineering plastic. However, the choice of bonding method for this kind of material is quite restricted. The bonding method, which was used to join two pieces of this material together, is elaborated in Figure 2.8. Both top and bottom walls of the duct are fastened between the two side walls by stainless steel self-tapping screws, together with a layer of silicone sealer to prevent leakage. This fabrication method is the only appropriate one because polycarbonate can neither be welded nor glued together.

The flow development region serves as a stabilising passage for the flow before approaching the test section. The entry length in the water tunnel facility can be determined experimentally. The flow condition in an internal passage is considered as fully developed only when its velocity profile does not change with increasing distance from the entrance. Thus, in this case, by plotting the pressure drop against the distance from the entrance as shown in Figure 2.9, the developing region can be predicted.

Approximately 5 m downstream from the entrance, a heating plate with dimensions 400-mm-wide by 600-mm-long is mounted flush with the bottom wall of the tunnel. It is to facilitate the study of heat transfer between a solid boundary and flowing fluid above. The heating plate consists of a lamination of a 2-mm-thick aluminium sheet and a 0.5-mm-thick stainless steel film.

Figure 2.5 – Isometric view of the rectangular duct

Figure 2.6 – Top view of the 6-m-long water tunnel

Figure 2.7 – Embodiment of the pressure-tap-point

Figure 2.8 – End view of the rectangular duct

Figure 2.9 - A sample of the entrance length approximation

Directly above the heating surface, there is an opening (Figure 2.5). During the experiments, this opening is covered with a transparent window panel that can be easily removed by unscrewing all the self-tapping screws around its four sides. In order to prevent leakage around the window panel, a special kind of permanent-wet-gasket is applied between the window panel and the opening frame.

Below the heating plate, 27 pairs of K-type thermocouples are installed across the heating section to measure the surface temperature of the heater. All these 27-pairs of thermocouple are sandwiched between the 2-mm-thick aluminium sheet and the flat heater. A thin layer of thermally conductivity silicone paste is applied between the lamination to minimise contact resistance.

On the other side of the flat heater, a low-density ceramic insulator is sandwiched between the flat heater and the stainless steel plate to minimise the heat loss from the heater. A stainless steel plate that acts as a backing is supported by an array of equally spaced bolts. The bolts create a cramping force to eliminate air gap between the lamination and to enhance heat transfer.

2.4 Exit section

An inclined ramp made of 10-mm-thick polypropylene sheet is used as the exit section. The inclined ramp is installed directly at the exit. Directly below the ramp is the reservoir tank. With this configuration, space utilisation is optimised for the construction of longer duct. The ramp is inclined at an angle of approximately 5° (arbitrary) relative to the bottom wall of the duct. This angle was defined arbitrarily to deflect pressure wave presented in the exiting

water flow away from the water tunnel. The ramp is supported by two pieces of 10-mm-thick polypropylene sheets that were welded onto both sides of the ramp as shown in Figure 2.10, so that the ramp can be properly secured onto the bracket of the reservoir tank.

Figure 2.10 – Rendering view of the exit section

2.5 Reservoir tank

The reservoir tank serves the purpose of allowing the introduction of the drag-reducing surfactant and the cooling of the re-circulating fluid. Fluid from the exit section creates a highly disorderly flow patterns in the buffer region, which is desirable for the mixing of the drag-reducing agent into the re-circulating fluid. The cooling of the re-circulating fluid is achieved with a cooling coil that is submerged in the tank. In the cooling coil, chilled water at desired temperature is pumped through it with an external re-circulating chiller system. The reservoir tank is designed and fabricated to be large enough to hold a total volume of 233 L of water, which is approximately 50% more than the volume of water contained in the 6-m-long rectangular duct. Its physical dimensions are shown in Appendix B.

Figure 2.11 – Rendering view of the reservoir tank

2.6 Water pumping system

Typically, the overall physical height of a water pump with larger capacity is much higher than one with a smaller capacity. For that reason, two water pumps with smaller capacity are connected in series instead of a larger one, so that the overall physical height of the experimental facility can be constructed lower.

Prior to the selection of the water pump from its performance graph in Appendix C, two selection parameters have to be determined, namely the total pressure drop that the pump has to overcome and the maximum volumetric flow rate that the pumps have to deliver. The head of the water pump was selected based on the required total pressure drop with the

consideration of a safety factor of 2. For the volumetric flow rate of each water pump, it was taken as half of the total volumetric flow rate that was used to calculate its required head.

The total pressure drop required for the selection of water pump is obtained through the sum of the wall frictional resistance in the flowing passage and the pressure drops across all the bends and valves. The calculation of the pressure drop due to wall frictional resistance involves summing up the pressure losses along the 76.2-mm-diameter transportation pipeline together with the 6-m-long rectangular duct.

Prior to the calculation, the following assumptions were made.

Maximum volumetric flowrate (Q_{max}) = 0.014 m³/s

Maximum Bulk Velocity in the Test Section (U_b) = 0.56 m/s

Maximum Bulk Velocity in the transportation pipeline (V_p) = 3.07 m/s

Fluid temperature = 30 °C ± 3 °C

Fluid dynamic viscosity (μ) = 0.801 × 10⁻³ N.s/m²

Fluid kinematic viscosity (ν) = 0.804 × 10⁻⁶ m²/s

Fluid density (ρ) = 1000 kg/m³

Inner height of the water tunnel (H) = 0.05 m

Inner width of the water tunnel (W) = 0.5 m

The pressure drop across the 6-m-long rectangular duct is calculated using Eq. (2.1). The derivation of this equation can be found in Chapter 3. It was derived from the relation of the Fanning friction factor together with the consideration of the cross-sectional geometry of the rectangular duct.

$$C_f = (4.545 \times 10^{-5}) \times \frac{\Delta P}{U_b^2 \times \Delta x}, \quad (2.1)$$

where C_f is the Fanning friction factor, Δp is the pressure drop over the distance Δx in the streamline direction, and U_b is the maximum bulk velocity in the rectangular duct. Rearranging Eq. (2.1), we have

$$\Delta P = \frac{C_f \times U_b^2 \times \Delta x}{4.545 \times 10^{-5}} \quad (2.2)$$

To make use of Eq. (2.2), the Fanning friction factor C_f has to be determined. It can be calculated from Eq. (2.3) obtained by Dean (1978). This relation is commonly used to predict the friction factor of water in internal flows.

$$C_f = 0.073 \times Re^{-0.25}. \quad (2.3)$$

where Re is the Reynolds number defined as

$$Re = \frac{U_b \times H}{\nu} \quad (2.4)$$

where H is the height of the rectangular duct, and ν is the fluid kinematic viscosity. By substituting U_b as 0.56 m/s, H as 0.05 m and ν as $0.804 \times 10^{-6} \text{ m}^2/\text{s}$ into Eqs. (2.2) – (2.4), the pressure drop across the 6-m-long rectangular duct is calculated to be 219 Pa.

To calculate the pressure drop in the 76.2-mm-diameter transportation pipeline, Eq. (2.5) will be used. It is derived from the relation of Fanning friction factor together with the consideration of the cross-sectional geometry of the pipe.

$$C_f = (5 \times 10^{-5}) \times \frac{\Delta P}{\Delta x \times V_p^2} \quad (2.5)$$

where C_f is the Fanning friction factor, ΔP is the pressure drop across the 76.2-mm-diameter transportation pipeline, V_p is the maximum bulk velocity in the 76.2-mm-diameter transportation pipeline and Δx is the overall length of the pipeline. Eqs. (2.5) is used together with the following definition,

$$Re = \frac{V_p \times D}{\nu} \quad (2.6)$$

where D is the inner diameter of the transportation pipeline. By substituting V_p as 3.07 m/s, D as 0.0762 m and ν as $0.8 \times 10^{-6} \text{ m}^2/\text{s}$ (Incropera and DeWitt, 1996) into Eqs. (2.5) and (2.6), and assuming the total length of the transportation pipeline to be 7 m, the pressure drop in the 76.2-mm-diameter transportation pipeline is about 4,563 Pa.

The minor loss at each fitting is calculated by Eq. (2.7) (Potter and Wiggert, 1997). The minor loss coefficients, K , at each particular fitting were obtained from Potter and Wiggert (1997) and presented in Table 2.1.

$$\text{Minor Loss} = \frac{K \times V_3^2}{2 \times g} \quad (2.7)$$

where V_3 is the bulk velocity at that particular fitting and g is the gravitational acceleration.

Table 2.1 – Computation of minor losses in the re-circulating loop (Potter and Wiggert, 1997)

Type	Loss coefficient (K)	Velocity of Water m/s (V_3)	Minor Losses (m)	Location in the facility
0.03	0.56	0.00048	Contracting Nozzle Between Entrance Section and the Water Tunnel
1	3.07	0.48	Supply pipe exit into the Entrance Section
76.2 mm Elbow	0.35	3.07	0.17	Along supply pipe, before exit into Entrance Section
0.5	3.07	0.24	Between pump inlet point and the reservoir tank
Globe valve (Fully open)	7.55	3.07	3.6	Before water pump
Globe valve (Fully open)	7.55	3.07	3.6	After water pump
Pressure Loss due to Minor Losses in fitting, $\Delta p_{fitting} =$			8.09 m ~79 362 Pa	

Location	Estimated Pressure Drop (Pa)	Estimated Pressure Drop (m)
6-m-long rectangular duct	219	0.02
76.2-mm-diameter transportation pipeline	4,563	0.47
Minor losses across all fittings	79,362	8.09

CHAPTER THREE: Experimental Overview

3.1 Outline

The heat transfer enhancement with a row of low-profile mixer in the newly constructed two-dimensional water tunnel will be studied. Prior to this, drag and heat transfer reduction phenomenon as a result of the addition of surfactants were examined. Hydrodynamic development lengths (pressure drop along the 6-m-long channel) were also measured at different surfactant concentrations to justify the fully developed flow conditions at the test section. A Reynolds number of 7000 was chosen as the lower limit to make sure that the flow in the 6-m-long rectangular duct was fully developed, while the upper limit of the Reynolds number was chosen as 20000 to minimise suspension of air bubbles in the flow.

Drag reduction in a channel flow is significant when small concentrations of surfactants are introduced (Aguilar *et al.*, 2001; Lin *et al.*, 2000; Myska and Chara, 2001). It is important to make sure that this effect is also achievable in the experimental facility before any further investigation can be performed. In order to demonstrate this phenomenon, the values of drag were measured at different Reynolds numbers with the presence of various surfactant concentrations, namely 0, 30, 70, 80 and 90 ppm. The acquired results at the various concentrations vs. Reynolds number were then plotted in one graph for comparison.

The heat transfer reduction, which accompanied the drag reduction, is another interesting phenomenon in drag-reducing fluids. In the heat transfer experiments, average heat transfer

coefficient obtained in the condition of pure tap water is compared with that of different surfactant concentrations.

Similar to the experiment for drag reduction, the heat transfer effect was initially obtained in the flow with zero concentration of surfactant and subsequently with 30, 70, 80 and 90 ppm of surfactant. The only difference between the heat transfer and drag reduction experiments was the settling time taken by the experimental facility to achieve stable condition for temperature in the test section. During the heat transfer reduction experiment, a settling time of 45 minutes was allowed for the heating surface to reach steady condition whenever there was a change in flow condition.

The heat transfer effect with different concentrations of surfactant was determined through the measurement of the power input to the heater and the temperature at various locations. The measured parameters in the heat transfer experiment were converted to Nusselt number to quantify the effect of heat transfer. The averaged Nusselt numbers over the heating surface vs. Reynolds number at various surfactant concentrations were then plotted for analysis. Details of the heat transfer experimental set-up, the experimental procedures and the acquired experimental results are presented in subsequent sections.

Gasljevic and Matthys (1997) and Matthys (1991, 1997) have shown that when high Concentrations of surfactants (~ 2000 ppm) were added into a pipe flow, the developing length would increase dramatically (e.g. to a streamline distance of $x/D = 117$, where x refers to the developing length and D refers to the diameter of the pipe). Thus, it is important to determine the developing lengths at different surfactant concentrations to ensure that the flow condition

in the test section is fully developed. The set-up for the flow-development-check experiment will be highlighted in Chapter Four. In this experiment, pressure drops at intervals of 300 mm along the channel were measured and plotted against position of the pressure measurement in a graph.

3.2 Preparation of the surfactant solutions

For the purpose of drag reduction in a fluid flow, cetyltrimethyl ammonium chloride (CTAC) together with the correct amount of sodium salicylate (Nasal) is well recognised to be an effective drag reducing agent. According to Lin *et al.* (2000), increasing the Nasal portion will result in higher negative charges in the flow to increase the neutralisation of the positive charges on the surfactant headgroups. As a result, micelles with longer structures are formed to increase the drag reduction effect.

CTAC has a chemical formula of $C_{19}H_{42}CLN$ and a molecular weight of 320, while Nasal has a chemical formula of HOC_6H_4COONa and a molecular weight of 160. CTAC is normally used as an additive to reduce surface tension. As for Nasal, it is commonly used as counter-ions. In this experimental study, CTAC and Nasal were pre-mixed with de-ionized water and stirred in a container with a magnetic stirrer for two hours, before it was added into the water tunnel facility. The proportion for CTAC/Nasal mixture is 0.5 in terms of molar ratios (CTAC/Nasal).

3.3 Data reduction

The relations used to calculate Reynolds number, Fanning friction factor and heat transfer coefficient were derived from the thermophysical properties of water. It is a common practice

in the field of surfactant drag-reducing flow to use the thermophysical properties of the solvent (for instance water) in the data reduction (Pollert *et al.*, 1994; Gasljevic and Matthys, 1997; Qi *et al.*, 1999; Li *et al.*, 2001). The thermophysical properties of water used were based on the average bulk temperature of the flowing fluid.

3.4 Experimental procedures and set-up for flow development check

When the fluid enters the duct, viscous effects become important as the boundary layer starts to develop at the entrance. The thickness of the boundary layer increases with the increase of the downstream distance. Figure 3.1 shows a diagram that explains the development of the boundary layer from the entrance. The boundary layers which developed on the top and bottom walls merge on the centreline at a certain distance downstream. Beyond this merged point, the velocity profile will not change (fully developed) with increasing distance. The distance from the entrance to the location where fully developed condition is achieved is termed as the hydrodynamic entry length.

Figure 3.1 – Flow development diagram

As Gasljevic and Matthys (1997) and Matthys (1991, 1997) had pointed out, the hydrodynamic entry length will increase significantly when drag-reducing additives are introduced into the flow. According to these studies, the entry effect in drag-reducing flow (viscoelastic fluids) might contribute significantly to the overall heat transfer and pressure drop. Thus, it is important to select the pressure tapping points (for drag reduction experiment) at locations away from the developing region so as to eliminate another source of uncertainty.

The main aim of the flow development check experiment is to identify this hydrodynamic entry length in the rectangular duct at different surfactant concentrations. This check involves the measurements of the pressure drop at intervals of 300 mm along the duct. A reference pressure-tap point is selected at about 5.46 m downstream of the duct. The acquired data at different surfactant concentrations are presented in separate graphs of pressure drop versus distance relative to the entrance. A best-fit line is then drawn across those data points. This best-fit line corresponds to a constant friction factor, as the change in pressure drop, dp , over the change in distance relative to the entrance, dx , is proportional to the Fanning friction factor, C_f , (Tennekes and Lumley, 1972), i.e.

$$\frac{dp}{dx} \propto C_f \quad (3.2)$$

In order to further elaborate this relationship, Eq. (3.2) is re-written as

$$C_f = \text{Constant} \times \frac{dp}{dx} \quad (3.3)$$

In a fully developed channel flow, the pressure drop, Δp , is a linear function of distance, Δx .

Therefore, the pressure gradient $\frac{dp}{dx}$ can be represented by the ratio of the finite differences,

$$\frac{\Delta p}{\Delta x}, \text{ i.e.}$$

$$\frac{dp}{dx} = \frac{\Delta P}{\Delta x}$$

Eq. (3.3) can be re-written as

$$C_f = \text{Constant} \times \frac{\Delta P}{\Delta x} \quad (3.4)$$

where ΔP is the pressure difference, and Δx is the distance between the pressure tapping points and the reference location.

Experimental errors can be minimised by conducting experiments at a Reynolds number of about 20000 and keeping a distance Δx of at least 2 m between any two pressure tapping points so that frictional effect between any two pressure tapping points is much larger than the error due to the surrounding disturbance. When the magnitude of the friction is relatively large, the error becomes insignificant.

The flow development check experiments were repeated at the same Reynolds number ($Re = 20000$) for various surfactant concentrations. During the experiments, pressure differences relative to the reference point near the exit of the channel were measured with a digital differential pressure transducer. Precaution was taken to ensure that the data recorded were reliable by providing a period of two minutes as settling time. Averaged readings were taken from the digital pressure transducer for analysis.

A pressure measuring system, as shown in Figure 3.2, was set up to measure the pressure drop along the duct. The pressure measurement system consists of a pressure transducer, a row of pressure-tap points that were embedded along the duct and a pressure-line-selecting switch.

The pressure transducer (Druck LPM 5000 series) has a pair of pressure-signal ports. One of the ports (low-pressure port) is connected permanently to a reference pressure tapping point located at 5.46 m downstream from the duct entrance (Figure 3.2) while the other ports (high-pressure port) is connected to the pressure-line-selecting switch. The pressure-line-selecting switch was installed between the pressure transducer and the water tunnel facility to enable selection of signal-line from the multiple pressure-tap point embedded on the facility.

The pressure-line-selecting switch is basically an enclosed cylindrical vessel with an outlet, a row inlet port for pressure signal and a purging port. The function of the outlet port is to transmit selected pressure signals to the pressure transducer. The row of inlet ports are used to tap pressure signals from varies pressure-tapping points along the duct. Air bubbles in the pressure-signal transmitting lines are removed through the purging port.

Figure 3.2– Schematic of differential pressure measurement setup

3.5 Experimental set-up and definition of drag reduction

Frictional loss in an internal flow is commonly denoted as drag. The drag is quantified by a dimensionless parameter known as Fanning friction factor. This dimensionless parameter can be calculated from the wall shear stress, which can be calculated from the measured pressure gradient along the duct.

The newly constructed water tunnel facility and a pressure transducer (Druck LPM 5000 series) were used to perform the drag reduction experiments. The pressure transducer was installed directly onto the water tunnel to determine the drag across the test section. The fluid was circulated in the facility at different flowrate by two centrifugal water pumps with Reynolds numbers ranging between 7000 and 18500.

Figure 3.3 – Schematic of the drag experimental setup

After establishing the developing length in the flow development check experiment, two pressure-tap-points were selected for the measurement of drag. One of the selection criteria for the pressure tapping points was that the distance between these two points must be as large as possible. The farther the distance of the pressure tapping points from each other, the larger the magnitude of the measured differential pressure signal. If the measured pressure difference is large, the error in the experiment would be insignificant. Thus, the pressure-tap-

points located at 3.06 and 5.46 m downstream from the entrance were selected for the experiment (Figure 3.3). At these two locations, the flow is fully developed.

The drag reduction, DR , is defined as the difference between the values of the Fanning friction factor, C_{fw} for water and that for the surfactant solution at the same Reynolds number, divided by the value for water. This is normally expressed in terms of a percentage:

$$DR = \frac{C_{fw} - C_{fs}}{C_{fw}} \times 100\% \quad (3.6)$$

where C_{fw} and C_{fs} refer to the Fanning friction factors for the water and the surfactant solution, respectively. According to Bewersdorff and Gyr (1998), the wall shear stress, τ_w , and the Fanning friction factor, C_f , are related as

$$C_f = \frac{\tau_w}{\rho \times \frac{U_b^2}{2}} \quad (3.7)$$

where ρ is the density of water, U_b is the bulk velocity and τ_w is the shear stress on the wall surface of the rectangular duct.

Figure 3.4 – Schematic of the test section

The wall shear stress τ_w is calculated from the differential pressure measured between two points that were embedded on the test section.

$$\tau_w = \frac{H \times W \times \Delta p}{(2 \times H + 2 \times W) \times \Delta x} \quad (3.8)$$

A force balance across a section of the flow in the test section is used to calculate the wall shear stress. This method was used in experimental studies by many researchers (Warholic, 1997; Michael *et al.*, 1999).

Substituting Eq. (3.8) into Eq. (3.7), the Fanning friction factor, C_f can be expressed in terms of the measured pressure difference over a distance Δx , i.e.

$$C_f = \frac{H \times W \times \Delta P}{\rho \times U_b^2 \times (H + W) \times \Delta x} \quad (3-9)$$

With further substitution of the duct geometry ($H = 0.05$ m, $W = 0.5$ m) into Eq. (3.9), C_f is reduced to

$$C_f = (4.545 \times 10^{-5}) \times \frac{\Delta P}{U_b^2 \times \Delta x}. \quad (3.10)$$

3.6 Experimental set-up and definition of heat transfer reduction

In order to determine the heat transfer effect, 27 pairs of IS-type thermocouples were installed at various locations of the test section (Figure 3.5). A voltmeter and an ammeter were used to measure the input voltage and current in order to determine the heat flux that is transferred into the flow.

The approach of the heat transfer experiments similar to that used for drag reduction experiments. Prior to the introduction of surfactant, a heat transfer baseline (with 0 ppm surfactant concentration) was measured over a range of Reynolds numbers between 7000 and 18500. After that, the heat transfer experiments were repeated with the surfactant concentrations of 30, 70, 80 and 90 ppm accordingly.

During the experiments, a direct-current electrical power supply was deployed and set to deliver a constant power of 315 W to the heater, so as to create a constant heating flux

condition in the test section. A settling time of 45 minutes was catered for the heating surface to reach steady state condition whenever there was a change in Reynolds number.

All the experimental data at different surfactant concentrations were obtained with the same time interval. The time interval had to be kept consistent for different surfactant concentrations, as the heat transfer reduction by the surfactant additives may depend on time according to experimental study of Qi *et al.* (2001).

An aluminium heating plate with an effective heating surface area of 400-mm-wide by 600-mm-long was installed flush with the bottom wall of the test section. On top of the aluminium heating plate, a 0.5-mm-thick stainless steel film is covered to prevent the flowing liquid from contacting the aluminium heating plate and to avoid corrosion on the aluminium surface.

Beneath the aluminium heating plate, 27 pairs of K-type thermocouples were installed for the measurement of temperature on the heating surface. The thermocouples were calibrated using a calibration bath (Therma Cal Inc Model 18B) prior to the installation. The layout of the thermocouples is presented in Figure 3.5.

Figure 3.6 shows the thermocouple installation. All the 27 pairs of thermocouples were embedded (sandwiched) between the 2-mm-thick aluminium heating plate and the heater. A thin layer of thermally conductive silicone paste was applied between the 2-mm thick aluminium heating plate and the heater to minimise thermal contact resistance.

All K-type thermocouples were connected to a data acquisition system (Figure 3.6). Signals from the thermocouples were conditioned and recorded by the data acquisition system for further analysis.

A low-density ceramic insulator was installed on the bottom side of the heater to minimise heat loss. The low-density ceramic board is sandwiched between the heater and a 2-mm-thick stainless steel. Below the 2-mm-thick stainless steel plate is an array of bolts that act as reinforcement for the entire assembly. These equally spaced-bolts create a distributed cramping force across the heating region to eliminate air gap between the lamination.

Figure 3.5 – Thermocouple distribution across the heating surface

Figure 3.6 - Thermocouple embodiment diagram

Figure 3.7 – Side view of the heater insulator

The heat transfer reduction, HTR , is commonly defined as the difference between the values of the Nusselt number, Nu , for water and for surfactant solution at the same Reynolds number, divided by the value for water (Gasljevic and Matthys, 1993; Qi *et al.*, 2001; Li *et al.*, 2003). It is expressed in terms of a percentage:

$$HTR = \frac{Nu_w - Nu_s}{Nu_w} \times 100\% \quad (3.11)$$

where Nu_w is the Nusselt number for water and Nu_s is the Nusselt number for surfactant solution. The average Nusselt number is calculated from the average heat transfer coefficient from the following equation:

$$Nu_i = \frac{\bar{h} \times H}{k} \quad (3.12)$$

where $\bar{h}$ is the average heat transfer coefficient along the heating wall and k is the thermal conductivity of water. The average heat transfer coefficient, $\bar{h}$, is obtained by integrating the local heat transfer coefficient over the entrance distance, x , which is measured from the leading edge of the heating surface. The local heat transfer coefficient is not a constant value along the heating wall in the streamwise direction (Figure 3.8). The average heat transfer coefficient, $\bar{h}$, can be calculated with the following equation:

$$\bar{h} = \frac{1}{L} \int_0^L h_x \cdot dx \quad (3.13)$$

where L is the overall length of the heating surface and h_x is the local heat transfer coefficient, which is a function of x (Figure 3.8).

Figure 3.8 - Local convective heat transfer coefficient over the heating surface

The local heat transfer coefficient, h_x , is calculated from Newton's law of cooling as,

$$q_s'' = h_x \times (T_{sx} - T_{bx}) \quad (3.14)$$

where q_s'' is the local heat flux, T_{sx} is the local surface temperature of the heating wall and T_{bx} is the local bulk temperature of the flow above the heating wall.

The local heat flux is calculated from the electrical power supplied to the heater with the assumption of negligible heat loss from the back of the heater, which was achieved by insulating the bottom wall of the heater. Detail of calculations performed to show that the heat loss is negligible are given in Appendix D. Re-arranging Eq. (3.14), the local heat transfer coefficient h_x becomes

$$h_x = \frac{q_s''}{(T_{sx} - T_{bx})} \quad (3.15)$$

Local surface temperature, T_{sx}

The local surface temperature, T_{sx} cannot be measured directly in this experimental set-up. They are calculated individually from the temperature that is acquired from the 27 pairs of K-type thermocouples embedded between the heater and the heating plate (shown in Figure 3.6.) A relation to calculate the local surface temperature T_{sx} was derived from the thermal circuit as shown in Figure 3.9.

In Figure 3.9, L_{al} is the thickness of the aluminium plate, L_{ss} is the thickness of the stainless steel plate, k_{al} and k_{ss} are the thermal conductivity of aluminium plate and the stainless steel plate, respectively, and T_3 is the local temperature measured directly from the thermocouple. The thermal contact resistances between the stainless steel plate, the aluminium plate and the electrical heater are neglected. As mentioned previously, a layer of silicone paste (Omega) was applied between the layers to reduce the thermal contact resistances. In addition, this lamination was held reasonably tight together with the clamping device show in Appendix E.

The thermal resistance of the heating plate was estimated and compared with the experimental result given in Incropera and DeWitt (1996). It was concluded that the magnitude of the estimated thermal contact resistance (10^{-6}) was an order smaller than the thermal resistance for the thin heating plate (10^{-5}). Thus, it can be neglected.

Figure 3.9 - Equivalent thermal circuit of the heating plate

To compute the local surface temperature, T_{ss} the equivalent thermal circuit in Figure 3.9 is represented by Eq. (3.16).

$$q_s'' = \frac{T_3 - T_{ss}}{\frac{L_{ul}}{k_{ul}} + \frac{L_{ss}}{k_{ss}}} \quad (3.16)$$

Rearranging Eq. (3.16) in terms of T_{ss} and substituting L_{ul} as 0.002 m, k_{ul} as 172 W/m·K, L_{ss} as 0.0005 m and k_{ss} as 15 W/m·K into Eq. (3.16), the relation to calculate the local surface temperature T_{ss} is established.

$$T_{ss} = T_3 - 4.5 \times 10^{-5} \times q_s'' \quad (3.17)$$

Local bulk temperature T_{bx}

The local bulk temperature, T_{bx} at an arbitrary streamwise location in the heating region is defined as follows (Eschenbacher *et. al.*, 2001):

$$T_{bx} = T_{inlet} + \frac{1}{\rho \times C_p \times U_b \times H} \times \int q_s'' dx \quad (3.18)$$

where T_{inlet} is the bulk temperature of fluid before contacting the heating region, ρ is the density of water, C_p is the specific heat of water, U_b is the bulk velocity of the flow. H is the height of the test section and x is the distance from the leading edge of the heating surface.

Solving Eq. (3.18) by assuming q_s'' as a constant over the heating surface, the local bulk temperature, T_{bx} , is rewritten as

$$T_{bx} = T_{inlet} + \frac{q_s \times x}{\rho \times C_p \times U_b \times H} \quad (3.19)$$

By substituting, ρ as 1000 kg/m^3 , C_p as 4179 J/kg.K and H as 0.05 m , Eq. (3.19) is further reduced to

$$T_{bx} = T_{inlet} + 4.8 \times 10^{-6} \times \frac{q_s'' \times x}{U_b} \quad (3.20)$$

The inlet bulk temperatures T_{inlet} in Eq. (3.20) cannot be measured directly, as the deployment of thermocouple before the heating surface will disturb the flow in the downstream, creating an additional uncertainty to the convective heat transfer analysis. Despite this, installing a thermocouple after the heating region is more appropriate, as the disturbance of the thermocouple to the flow will not have any influence to the heat transfer activities in the heating region up stream. Thus, a K-type thermocouple was installed after the heating region (at about 0.7-m-away from the leading edge of the heating surface) as shown in Figure 3.10.

This thermocouple was mounted on a three-dimensional traversing mechanism to allow free and yet controllable movement across the rectangular duct to measure the outlet bulk

Figure 3.10 – Local bulk temperature over the heating surface

The bulk temperature at the outlet is dependent on Reynolds number and surfactant concentration. Thus, at different flow conditions, the outlet bulk temperature will be obtained and substituted into Eq. (3.20) to calculate the inlet bulk temperature, T_{inlet} .

3.7 Experimental procedure and set-up for heat transfer enhancement

A row of low-profile mixers are used to enhance heat transfer in the drag-reducing flow. They are also called low-profile vortex generators when applied for separation control. The practical advantages of such devices are their inherent simplicity and low device drag (Lin *et al.*, 1990, 1991; Lin, 1999; Ashill *et al.*, 2001; Jenkins *et al.*, 2002; McCormick, 1992; Mounts and Barber, 1992). According to previous studies (Lin, 2002), the most effective vortex generator that is able to produce longitudinal vortices with low pressure loss would have a height that is between 10 to 35% of the boundary-layer thickness at the device installation location.

As shown in Figure 3.1 1, a row of low-profile mixer was installed across the heating surface about 225 mm downstream of the leading edge of the heating surface. The dimensions of each low-profile mixers are shown in Figure 3.12.

It is expected that due to the use of the low profile mixers, longitudinal vortices will be generated downstream. As shown in Figure 3.13, these longitudinal vortexes which rotate counter-clockwise to their neighboring vortices, create a series of suction fields that draw in cold fluid from the main stream into the near wall region to absorb heating energy. The cold fluid will not stay in the near wall region for long. Almost immediately after it comes into contact with the hot surface, it will be carried away by the spinning momentum and ejected into the main stream again. This bulk exchange activity helps to dissipate heat energy away from the heating surface at a higher rate, resulting in local heat transfer enhancement.

Figure 3.1 1 - A row of low-profile mixers

Three sets of flow visualization experiments with dye were performed separately to visualize the mechanism of the mixing effect that was induced by the low-profile mixer. The first set is to study the vortex interaction within a single device. The dye nozzle for this test was positioned upstream of the centre axis of the low-profile mixer and maintained as close to the wall as possible to minimise diffusion of dye. A high-resolution video recorder was used to record the rotation of the longitudinal vortex.

In order to further justify the rotational flow field induced by the low profile mixers, the dye nozzle was shifted horizontally between two low-profile mixers for the study of vortex interaction between devices.

As for the third set of flow visualization test, the effectiveness of the low-profile mixer to enhance heat transfer in drag-reducing flow was studied. In this experiment, the dye nozzle was placed on the bottom wall (low velocity flow region) at about 0.3 m upstream of the devices. Both the main flow velocity and the dye injection rate were adjusted low enough to let the dye form a straight, long and clean streamline.

To further investigate the performance of the low-profile mixers, local heat transfer coefficients were determined at three different Reynolds numbers, namely 7000, 12000 and 16200, and beyond $x/H=5$. This investigation was conducted under constant heat flux condition with thermocouples installed in the streamwise direction. The experimental results for drag and heat transfer reduction and heat transfer enhancement are given in the Chapter four.

Figure 3.12– Dimensions of the low-profile mixer

Figure 3.13– Rotational direction of vortices viewed from the end of the test section

CHAPTER FOUR: Results and Discussion

4.1 Flow development check for various surfactant concentrations

For fully developed channel flows, Tennekes and Lumlay (1972) showed that

$$U_*^2 = -\frac{h}{\rho} \times \frac{dp}{dx} \quad (4.0)$$

where $U_* = \sqrt{\frac{\tau_w}{\rho}}$ is the friction velocity. Since U_* does not change in the streamwise direction

in a fully developed channel flow, $\frac{dp}{dx}$ should be constant.

In order to justify the fully develop flow conditions in the test section, the pressure gradients in the streamwise direction were measured at different surfactant concentrations, namely 0, 20, 60, 120, 160 and 200 ppm for Reynolds number of 20000. Pressure drop versus distance from the entrance obtained for each surfactant concentration was plotted separately onto graphs. The y-axis of the graph represents the measured pressure drop while the x-axis represents distance measured from the entrance of the duct.

Based on the trend in Figure 4.1 for 0 ppm of surfactant, the flow in the water tunnel appears to have a very short hydrodynamic entry length. This feature allows experiments conducted at the entrance of the duct with pure tap water to be considered as under fully developed flow conditions.

Figure 4.1 – Pressure drop in the streamwise direction with 0 ppm of surfactant. The straight line represents the least-square linear regression to the experimental data.

When the surfactant concentration was increased to 20 ppm, the fully developed flow conditions were delayed (Figure 4.2). The flow becomes fully developed only after a distance of 0.3 m from the entrance.

With an increase of surfactant concentration to 60 ppm in the flow, the hydrodynamic entry length increased to about 0.5 m from the entrance, causing a further delay in achieving a fully developed condition (Figure 4.3). At surfactant concentration of 120 ppm, a further increase in the entry length is noted. The entry length extends further downstream to a distance about 1.2 m away from the entrance. Another effect, which can be observed from Figure 4.4, is that the gradient of the best-fit line changed significantly to a gentler slope as compared with Figures 4.1-4.3. This corresponds to a drop in the friction coefficient in the channel flow.

Figure 4.2 – Pressure drop in the streamwise direction with 20 ppm of surfactant. The straight line represents the least-square linear regression to the experimental data.

Figure 4.3 – Pressure drop in the streamwise direction with 60 ppm of surfactant. The straight line represents the least-square linear regression to the experimental data.

With 160 ppm of surfactant in the flow, further delay on the merger of the top wall boundary layer and the bottom wall boundary layer occurs, resulting in a further extension of the flow development region (Figure 4.5). The flow only becomes fully developed after a distance of 1.4 m from the entrance. The gradient of the best-fit line shown in Figure 4.5 remains unchanged as compared with that given in Figure 4.4. This might be due to the attainment maximum drag reduction (Aguilar *et al.*, 2001) in a drag-reducing fluid at this surfactant concentration. This phenomenon is better elaborated on a plot of drag vs. Reynolds number in the subsequent section.

Figure 4.4 – Pressure drop in the streamwise direction with 120 ppm of surfactant. The straight line represents the least-square linear regression to the experimental data.

Figure 4.5 – Pressure drop in the streamwise direction with 160 ppm of surfactant. The straight line represents the least-square linear regression to the experimental data.

The last set of experiments conducted at a surfactant concentration of 200 ppm. The results plotted in Figure 4.6 agree well with those obtained for the case of 160 ppm of surfactant. These results suggest that the Fanning friction factors for the two different surfactant concentrations are the same. The only effect noted during the experiment is that the pressure fluctuations in the digital display become smaller as compared to the experiments with 160 ppm of surfactant. This may be due to the presence of a larger quantity of fibre-like structures (Yang, 2002) in the flow to damp the pressure fluctuation at higher surfactant concentration.

Figure 4.6– Pressure drop in the streamwise direction with 200 ppm of surfactant. The straight line represents the least-square linear regression to the experimental data.

4.2 Effect of surfactant on drag

The influence of surfactant additives on the skin friction coefficient (Fanning friction factor) in a two-dimensional channel flow was investigated. The Fanning friction factor was calculated from the wall frictional shear stress which is directly related to the pressure-drop in the streamwise direction of the fully developed turbulent channel flow.

The experiments were conducted for velocities ranging between 0.13 and 0.31 m/s, corresponding to a Reynolds number range of 7000 to 18500, with four different surfactant concentrations, namely 30, 70, 80 and 90 ppm. Untreated tap water was used as the working fluid. The temperature of the tap water was maintained at about 29°C ($\pm 1^\circ\text{C}$) by a chiller. Prior to the introduction of surfactant additive into the flow, the Fanning friction factor was obtained and plotted as Figure 4.7 for comparison with results obtained when surfactants of various concentrations were used. The Fanning friction factors obtained by Dean (1978) for two-dimensional turbulent flow of Newtonian fluids are shown in Figure 4.7 for comparison. Dean's equation is given as follows:

$$C_f = 0.073 \times \text{Re}^{-0.25} \quad (4.1)$$

The Fanning friction factors obtained at surfactant concentration of 0 ppm decrease with an increase in the Reynolds number. This trend agrees very well with Dean's equation. However, the results obtained in the present study are slightly higher (approximately 10–12% higher) than the values obtained by Dean. This deviation between the present result and Dean's result may be due to the difference in test section aspect ratio (Chen *et al.*, 2001). The present channel flow may not have a large enough cross-section aspect ratio. As a result, the

flow near its sidewalls as well as the corners may have some influence over the pressure measurements.

The Fanning friction factors obtained for a surfactant concentration of 30 ppm are about 7% lower than those for 0 ppm at about $Re = 7500$. These factors decrease at a different slope as compared to 0 ppm case, merging with the data obtained for the case of 0 ppm at $Re > 12000$. These results indicate that the drag reduction effect will disappear when $Re \approx 12000$ for a surfactant concentration of 30 ppm. The critical Reynolds number, Re_{cri} , for drag reduction at which the drag starts to recover, is around 9000. The loss of drag reduction effect is due to the high shearing effect at high Reynolds numbers; the aggregations of the micelles that aligned in the direction of the flow to suppress turbulence may have broken down into sphere-like micelles. The maximum achievable drag reduction at surfactant concentration of 30 ppm cannot be estimated unambiguously, as this phenomenon may have occurred at a lower Reynolds number.

With 70 ppm of surfactant in the flow, the drag is reduced by about 31% (Figure 4.7). The critical Reynolds number for drag reduction increases to approximately 13000. From the trend for 70 ppm, drag reduction is not expected for $Re > 19000$. With further increases in the surfactant concentration to 80 ppm, remarkable drag reduction occurred. The maximum achievable drag reduction increases to about 50% at $Re = 14000$. However, with an increase in the surfactant concentration to 90 ppm, the drag reduction did not change apparently, the maximum achievable drag reduction increases to only about 55% at $Re = 16000$. From the results shown in Figure 4.7, we can see that the critical Reynolds number, Re_{cri} , also increases with the increase of surfactant concentration. For example, $Re_{cri} \approx 9000$ at 30 ppm and Re_{cri}

≈ 15000 and 17000 for 80 and 90 ppm, respectively. This is consistent with the results obtained by Li *et al.* (2003).

As explained by Ohlendorf *et al.* (1986), an increase in surfactant concentration will lead to an increase in the length of the rod-like micelles that will align themselves along the flow direction to suppress turbulent flow near the wall. As a result, the skin friction between the flowing fluid and the solid boundary is reduced. Thus when the surfactant concentration in the fluid is higher, longer rod-like micelle structures will be formed to achieve larger drag reduction and higher critical Reynolds number for drag reduction. As a result, a larger amount of energy from the flow is required to disrupt the super-order rod-like aggregation.

The effect of surfactant concentration on drag reduction can also be demonstrated by plotting the percentage of drag reduction using Eq. (3.6) at different Reynolds numbers as shown in Figure 4.8. The peak value at various concentrations represents the maximum drag reduction. The peak values increase with the increase of surfactant concentration until the latter reaches 80 and 90 ppm. For these two concentrations, the drag reduction agrees very well until $Re \approx 14000$. This result suggests that the flow has reached its maximum attainable drag reduction with 80 ppm at Reynolds number of about 14000 . Thus, no further drag reduction is possible at that particular Reynolds number.

Figure 4.7 – Fanning friction factor at various Reynolds numbers

For comparison with the present results on the maximum attainable drag reduction, experimental results obtained by Kawaguchi *et al.* (2002) and Li *et al.* (2003) are also included in Figure 4.8. The maximum attainable drag reduction at $Re \approx 14000$ by Kawaguchi *et al.* (2002) is approximately 64%, which is about 20% higher than the present results. However, in the experimental study by Li *et al.* (2003), their maximum attainable drag reduction agrees closely with the present results for the Reynolds number range of 12000-16000.

The difference between the present maximum attainable drag reduction and that found by Kawaguchi *et al.* (2002) may be due to the difference in the aspect ratio of the experimental

facility; the latter authors used an aspect ratio of 12.5:1. Although the aspect ratio of the channel used in the present experiments is large enough (10:1) to be considered as a two-dimension duct, the presence of unstable regions near the comers and side walls may affect the aggregation structure, causing a partial degradation of the drag-reducing ability. This may result in a slightly lower maximum attainable drag reduction. Another factor that contributes to the differences is the influence of the fluid temperature. In the present experiments, the temperature of the fluid was maintained at about 29°C. According to Li *et al.* (2003), a 2°C difference in the fluid temperature will cause a remarkable change in their drag reduction results. Thus, it is expected to see slight deviations between the present experimental results and those reported by other researchers.

Figure 4.8 – Percentage of drag reduction at various Reynolds numbers

Lin *et al.* (2000) investigated the influence of fluid temperature from 5 to 60°C on drag reduction at different Reynolds numbers. They showed that at a particular Reynolds number, the percentage drag reduction increases with the increase in solution temperature. However, with further increase of the solution temperature, the percentage drag reduction decreases. Based on their study, the drag-reducing ability of the surfactant additive is quite sensitive to the solution temperature.

4.3 Uncertainty analysis of Fanning friction factor

The overall uncertainty for the value of Fanning friction factor C_f was determined with the propagation of random uncertainty technique first proposed by Kline and McClintock (1953) and later refined by Moffat (1982). The uncertainty for C_f , denoted here by ΔC_f , is given as

$$\Delta C_f = \left[\sum_{i=1}^n \left(\frac{\partial C_f}{\partial x_i} \times \Delta x_i \right)^2 \right]^{0.5} \quad (4.2)$$

where Δx_i ($i = 1 - n$) represent the uncertainty in the measurement of x_i . Two measured parameters, namely the pressure difference Δp and the bulk velocity U_b , are involved in the determination of C_f . The uncertainty in Δp measurement specified by instrument manufacturer, is ± 1.25 Pa. The uncertainty in U_b measurement, which is calculated from the uncertainty of the flowmeter ($\pm 0.5\%$ accuracy of the measured value), is estimated to be ± 0.00175 m/s.

The uncertainty analysis was conducted based on three different Reynolds numbers, namely 7000, 12000 and 16200. The results of the uncertainty analysis on C_f , together with individual

contributions from the prescribed variables Δp and U_b , are shown in Table 4.1. It is clear that the dominant contributor to the uncertainty in C_f is the measurement of Δp . At all Reynolds numbers, the magnitudes of $\left(\frac{\partial C_f}{\partial x_i} \times \Delta x_i\right)^2$ calculated from the uncertainty of Δp were of at least one order of magnitude higher than that of U_b . The results also show that for higher Reynolds numbers, the contribution from Δp to the uncertainty of C_f decreases with Reynolds number, changing from 1.79×10^{-6} at $Re = 7000$ to 8.93×10^{-8} at $Re = 16200$. As a result, the relative uncertainty of C_f also decreases with Reynolds number, changing from 14.9% at $Re = 7000$ to 4.1% at $Re = 16200$.

Table 4.1 – Contributions of various variables to the uncertainty of C_f

Individual contribution $\left(\frac{\partial C_f}{\partial x_i} \times \Delta x_i\right)^2$			
Parameters: Re:	7000	12000	16200
Pressure difference Δp	1.79×10^{-6}	3.01×10^{-7}	8.93×10^{-8}
Bulk velocity U_b	6.89×10^{-8}	2.21×10^{-8}	9.95×10^{-9}
Absolute uncertainty ΔC_f	1.36×10^{-3}	5.68×10^{-4}	3.15×10^{-4}
Relative uncertainty, $\frac{\Delta C_f}{C_f} (\%)$	14.9	6.9	4.1

4.4 Heat transfer without surfactant

Prior to the study of the heat transfer reduction and enhancement in drag-reducing flow, heat transfer coefficients in the channel flow with tap water were obtained to establish a benchmark for comparison. The average Nusselt numbers at different Reynolds numbers are shown in Figure 4.9. The averaged Nusselt numbers were calculated from the wall-to-bulk temperature differences that were measured at constant heat flux conditions. In addition, Gnielinski's (1976) result given by Eq. (4.3) is also included for comparison (Figure 4.9).

$$Nu_m = 0.012 \times (Re^{0.87} - 280) \times Pr^{0.4} \times \left(1 + \left[\frac{D}{L} \right]^{0.667} \right) \quad (4.3)$$

Equation (4.3) attributed to Gnielinski (1976) is commonly used to predict the average heat transfer coefficients for turbulent flows of high Prandtl number in smooth channels, and the flow condition is thermally and hydrodynamically developed.

The average Nusselt numbers obtained without surfactant additive from the present experiments agree reasonably well with Gnielinski's equation for Reynolds numbers in the range of 7000 to 14000. As the Reynolds number increases, which corresponds to an increase in fluid bulk velocity and hence an increase in velocity fluctuation, heat transfer also increases. The presence of turbulent flow in the wall region would induce better mixing between low-speed region close to the wall and the high-speed region of the main stream. This increase in the mixing of fluid between different flow layers encourages the dissipation of heat energy from the wall surface to the main stream. As the Reynolds number is increased to above 14000, the average Nusselt numbers deviate slightly from Gnielinski's results.

Figure 4.9 – Averaged Nusselt number at various Reynolds numbers in tap water

The increase of Nusselt number with the increase of Reynolds number can be explained using Particle Image Velocimetry (PIV) results as shown in Figure 4.10 (a,b), which were obtained by Kawaguchi *et al.* (2002). The instantaneous velocity distribution for water flow at $Re = 20000$ (Figure 4.10 a) shows a vector expression of spatial velocity distribution. In this figure, the main flow direction is from left to right and the frame covers the height of the channel. The bulk mean velocity is subtracted from all the vectors so that the vortex motion moving with bulk mean velocity can be seen.

From Figure 4.10 (a), it is clearly shown that the velocity field possess velocity vectors which have irregular direction and magnitude. Generally, velocity vectors head to the right at the core region where y/H falls within 0.25-0.75. In contrast, velocity vectors locating closer than

$y/H = 0.125$ from the wall head to the left. Within this region, rotational eddy motion is formed. The centre location, size and magnitude of the eddies are not uniform and the Figure shows similar patterns of motion visualised by tracers in turbulent flows.

Figure 4.10 (b) shows the contour map based on the magnitude of velocity. A random pattern of complicated ridge, valley and peaks can be seen in the middle of the channel. This means that large velocity fluctuation in time and space occupies this field. In the region closer than $y/H = 0.1$ from the wall, contour lines are closely distributed. This comes from large velocity gradient near the wall as commonly seen in turbulent wall flows.

One interesting finding from the contour is the penetrations between the low speed (close to the wall) and the high speed (in the core region of the flow) fluid. These penetration fronts incline downwards and have an angle of about 30° from the wall. The penetration between low speed fluid and high speed fluid are the key factors for enhancing heat transfer in the wall region.

(a)

(b)

Figure 4.10 – Instantaneous velocity measurement in water using PIV at $Re = 21000$ (Kawaguchi *et al.*, 2002). (a) Instantaneous velocity fluctuation vectors; (b) Contour map of instantaneous velocity distribution.

4.5 Heat transfer with surfactant

The heat transfer reduction experiments were performed with surfactants at concentrations of 30, 70, 80 and 90 ppm. For each concentration, the experiments were conducted at Reynolds numbers of 7000, 12000, 16200 and 18500, respectively.

The heat transfer reduction at different Reynolds numbers are presented in terms of the local Nusselt number as shown in Figures 4.11 to 4.14. The local Nusselt number acquired for tap water (0 ppm of surfactant) is also included in these figures for comparison.

At low Reynolds number ($Re = 7000$), there is no remarkable reduction of heat transfer with 30 ppm of surfactant in the flow. This results contradict those obtained by Gasljevic and Matthys (1993). In their experimental study, drag reduction phenomenon was coupled together with a dramatic heat transfer reduction even at low surfactant concentrations. However, such a phenomenon was not observed in the present experiments at the surfactant concentration of 30 ppm. This may be due to an earlier occurrence of the critical Reynolds number for the present heat transfer reduction at this concentration. The dramatic heat transfer reduction may have occurred at a Reynolds number lower than 7000, which is the lowest value covered in the present study. In the present experimental study, Reynolds number lower than $Re = 7000$ were not investigated because free convection will be relatively important when compared with forced convection at low Reynolds numbers. This is the result of buoyancy effects of the fluid on the heating surface. Gasljevic *et al.* (2000)

had highlighted in their study that the buoyancy effects in drag-reducing flow are significant when Reynolds number is low.

Another explanation for the absence of dramatic heat transfer reduction with surfactant concentration of 30 ppm may be the degradation of surfactant aggregating near the wall, which helps to reduce mass exchange in the vertical direction. The microstructure of the micelle may be destroyed by the heating effect in the heating region of the test section as the aggregation of the surfactant micelle depends on the fluid temperature significantly (Li *et al.*, 2003).

At a surfactant concentration of 70 ppm, there is a slight (about 15%) heat transfer reduction. Corresponding to Figure 4.11, the thermal developing region ends at around $x/H = 2.5$. Beyond this point, the local Nusselt number remains constant. By increasing the surfactant concentration in the flow to 80 ppm, remarkable heat transfer reductions are noted. The heat transfer reduction increases to the maximum value of 40% at $x/H = 8.5$. When the surfactant concentration is increased to 90 ppm, the reduction of the heat transfer remains the same as for 80 ppm. The local Nusselt number distribution for 90 ppm agrees well with that of 80 ppm. The reason is that the heat transfer reduction at this Reynolds number ($Re = 7000$) may have reached its maximum attainable value. Thus, further heat transfer reduction is not possible even with higher concentration of surfactant additives in the fluid.

Figure 4.11 – Streamwise distribution of local Nusselt number in water and surfactant solutions at $Re = 7000$

At Reynolds number of 12000 (Figure 4.12), there is no apparent heat transfer reduction with 30 ppm of surfactant in the flow. The result at this concentration agrees well with that of 0 ppm. The orderly rod-like microstructures may have been broken down into sphere-like structures, which do not have the ability to suppress turbulence on the heating surface.

However, when 70 ppm of surfactant is added in the fluid, the local heat transfer distribution dropped by about 33%. With higher concentration of surfactant in the flow, the aggregation forces between the surfactant molecules become stronger. Hence, the rod-like microstructures are able to sustain themselves in the flow and to suppress the mixing effect near the heating wall. By increasing the surfactant concentration in the flow to 80 ppm, remarkable heat transfer reductions are noted. The magnitude of heat transfer reduction increases to a

maximum value of 60%. When the surfactant concentration is increased further to 90 ppm, no further reduction of heat transfer is noted. The local Nusselt number distribution in this case collapses with the case of 80 ppm. This may be because the flow has reached its maximum attainable heat transfer reduction at this Reynolds number ($Re = 12000$). Thus, further reduction of heat transfer is not possible even with higher concentration of the surfactant additives.

The result in Figure 4.12 has also shown that with the increase in surfactant concentration, there is a delay in the establishment of fully developed thermal boundary condition. For example, the thermal boundary layer only becomes fully developed after $x/H = 3$ with 70 ppm of surfactant. As for 80 and 90 ppm, the thermal developing region extends to about $x/H = 6$. This phenomenon is expected, as Gasljevic and Matthys (1997) have previously highlighted in their study that the thermal developing region for drag-reducing surfactant flows are usually significantly longer than those for Newtonian fluids. For example, they claimed that the developing region in a pipe flow with surfactant can go up to $L/D = 1000$, where L is the length of the thermal developing regions and D is the diameter of the pipe.

Figure 4.12 -Streamwise distribution of local Nusselt number in water and surfactant solutions at $Re = 12000$

When Reynolds number is increased to 16200, the local Nusselt number distributions in the case of 0 and 30 ppm scattered significantly along the streamwise direction. Similar to the previous two Reynolds number cases ($Re = 7000$ and 12000), the local Nusselt numbers obtained from both 0 and 30 ppm collapse with each other. For the case of 70 ppm at $Re = 16200$, there is only a slight reduction (about 15%) of heat transfer for $x/H \leq 4$. Beyond that point heat transfer starts to increase. This phenomenon may be due to the breakdown of the rod-like microstructures which are unable to withstand the shearing stress from the higher flowrate and the heating effect near the wall.

However, when 80 ppm of surfactant is present in the flow, the local heat transfer distribution dropped significantly by about 55%. With higher concentration of surfactant in the flow, the aggregation forces between the surfactant molecules become stronger. As a result, the rod-

like microstructures are able to sustain themselves in the flow. By further increasing the surfactant concentration in the flow to 90 ppm, further increase in heat transfer reduction was not seen. The heat transfer reduction remains at about 55%.

Figure 4.13 – Streamwise distribution of local Nusselt number in water and surfactant solutions at $Re = 16200$

When Reynolds number is increased to 18500 (Figure 4.14), the reduction of heat transfer with 70 ppm that occurred at lower Reynolds numbers (Figures 4.11 to 4.13) is only seen for $x/H \leq 2$. The results obtained in this flow condition collapse with the results obtained in the cases of 0 and 30 ppm for $x/H > 2$. For the case of 80 ppm surfactant concentration, heat transfer reduction occurred dramatically (about 50%) at the beginning of the heating surface (at $x/H \leq 4$). However, for $x/H > 4$, heat transfer starts to recover slightly. This is because as the fluid flows over the heating plate, its local temperature increases in the streamwise direction. As a result, the aggregation of surfactant is broken down and the heat transfer

effect recovers subsequently. Thus, it is expected that the heat transfer effect will be recovered fully if the Reynolds number were to be further increased.

At 90 ppm, the heat transfer rate is reduced by about 55% (Figure 4.14). After a developing length of $x/H = 6$, the thermal boundary layer becomes fully developed. Based on the result at 90 ppm shown in Figure 4.14, it is difficult to conclude that the maximum attainable heat transfer reduction is achieved as there is a possibility for further heat transfer reduction with an increase in surfactant concentrations.

Figure 4.14 – Streamwise distribution of local Nusselt number in water and surfactant solutions at $Re = 18500$

4.6 Uncertainty analysis of Nusselt number

The overall uncertainty of Nusselt number Nu , denoted by ΔNu , is given as

$$\Delta Nu = \left[\sum_{j=1}^n \left(\frac{dNu}{dy_j} \times \Delta y_j \right)^2 \right]^{0.5} \quad (4.4)$$

where Δy_j ($i = 1 - n$) represent the uncertainty in the measurement of y_j . The individual variables for y_j include heater input current I , heater input voltage V , surface temperature T_s and bulk temperature T_b . These variables are measured with voltmeter, ammeter and K-type thermocouples, separately. The uncertainties of the instrument are estimated to be 10.16 A, ± 0.07 V and ± 0.25 K, respectively.

The results of the uncertainty analysis on Nu , together with individual contribution from the four variables at three different Reynolds numbers, are shown in Table 4.2. As shown in Table 4.2, the main contributor to the uncertainty in Nu is the measurement of T_s and T_b . At

all Reynolds numbers, the values of $\left(\frac{\partial C_f}{\partial y_i} \times \Delta y_i \right)^2$ calculated from the uncertainty of T_s and T_b

are at least one order of magnitude higher than other variables. The results show that the contribution from the uncertainty of temperature measurements increases with Reynolds number. As a result, the uncertainty $\Delta Nu / Nu$ increases with Reynolds number, changing from 8% at $Re = 7000$ to 15.5% at $Re = 18500$.

Table 4.2 – Contributions from various variables to the uncertainty of Nu

Individual contribution $\left(\frac{\partial Nu}{\partial y_j} \times \Delta y_j \right)^2$			
Parameters: \ Re:	7000	12000	18500
Heater input Current I	0.05	0.10	0.15
Heater input Voltage V	0.05	0.11	0.15
Surface temperature T_s	9.91	44.79	92.88
Bulk temperature T_b	9.91	44.79	92.88
Absolute uncertainty ΔNu	4.46	9.48	13.64
Relative uncertainty, $\frac{\Delta Nu}{Nu} (\%)$	8	11.7	15.5

4.7 Heat transfer enhancement

In the present study, heat transfer in drag-reducing flow is enhanced with a row of low-profile mixers or vertex generators installed at $x/H = 5$ from the leading edge of the heating surface and aligned in the spanwise direction. These devices were glued onto the heating surface directly to create an array of longitudinal spiraling vortices that help to disrupt the orientation of the orderly aligned micelle structures as these vortices promote large-scale bulk motion of fluid particles between the main stream and the near-wall region. The effectiveness of the devices on promoting the heat transfer rate will be presented hereafter.

Based on the studies of Torii *et al.* (2002) and Lin *et al.* (2002), the pressure drop caused by longitudinal spiraling vortices is normally insignificant. However, its contribution to the mixing in the vertical direction is significant. Thus, this enhancement method is suitable to control the overall performance in the heat exchanger of a district cooling system.

Prior to the heat transfer coefficient measurements, a flow visualisation experiment with dye was performed to illustrate the mechanism of the mixing enhancement induced by the low-profile mixers. This experiment was conducted for a surfactant concentration of 90 ppm. A dye nozzle was positioned directly upstream of the low-profile mixer and maintained as close to the wall as possible to minimise the diffusion of the dye.

A high-resolution video recorder was used to capture the rotation of the longitudinal vortex. Three frames extracted from the video file are shown in Figures 4.15 and 4.16 to illustrate the rotational flow field. Two of the slides in Figure 4.15 show the flow field at different times with the assistance of the dye nozzle that positioned at the centre of the mixer.

Figure 4.15 – Photographs of the longitudinal vortices rotating in counter clockwise direction at time intervals (a) $t = 0$ and (b) $t = 20$ seconds.

It can be seen that as the dye approaches the mixer, it splits into two separate paths over the device. Upon departure from the top edge of the two wings, these two dye paths form two spiraling vortices that spill over the right-side and the left-side wing in clockwise and anti-clockwise rotational directions, respectively. Due to counter-clockwise rotation of the two

longitudinal vortices, these vortices are able to sustain themselves across the heating surface, acting as stirrers along the centre axis of the particular low-profile mixer. Therefore, it seems that the low-profile mixer helps to eject heated fluid from the near wall region and replenish with colder fluid from the main flow.

In order to further justify the above explanation, the dye nozzle was shifted horizontally between two low-profile mixers as shown in Figure 4.16. Similar to the previous cases (Figure 4.15), the dye was injected slowly upstream of the devices. Corresponding to the tacking of the dye as it approaches the devices, the dye beam splits into two separate paths. One of the paths flows towards device ‘A’ while the other path flows towards device ‘B’. The dye path that moves towards device ‘A’ was swept toward the bottom corner and then was thrown vertically upward along the side wall of the left-side wing. Part of the dye was diffused into the main flow while the main part was sucked back towards the bottom wall direction at further downstream, forming an anti-clockwise rotational vortex in the streamwise direction.

As for the other dye path that flowed towards device ‘B’, similar to the one that flowed to device “A”, it flowed towards the bottom corner of the right-side wing and was thrown up vertically into the main flow, forming a longitudinal vortex rotating in clockwise direction. By superimposing all the longitudinal vortices, a suction and ejection flow field that draws in colder fluid from the main flow to replace the ejected hotter fluid in the near wall region was created.

Figure 4.16 – Photograph of longitudinal vortices rotating in counter-clockwise direction

The effectiveness of the low-profile mixers on the enhancement of heat transfer in drag-reducing flow was studied with another flow visualization experiment. The dye nozzle in this experiment was placed on the bottom wall at about 30 cm upstream of the devices. Both the velocity of the main flow and the dye injection rate was adjusted low enough to ensure the flow in the channel to be laminar to form a straight, long and clean streamline as shown in Figure 4.17. From Figure 4.17, it can be seen that as the dye exits from the nozzle, it stretched into a straight path at the upstream of the devices. However, as the dye passes the low-profile mixer, it disappeared completely as shown in the enlarged view. The disappearance of the dye near the bottom wall after the mixer was caused by the longitudinal vortices induced by the devices.

Figure 4.17 – Effectiveness of the low-profile mixers at low Reynolds number

Pressure drop caused by the low-profile mixers are checked through the pressure tappings that are situated on the top wall of the tunnel. The results show that the effect of low-profile mixers to the pressure drop along the tunnel is quite small (being only about a few Pascals),

indicating that the drag induced by the mixers is insignificant. This is in agreement with the reports by Lin (2002).

To further investigate the performance of the low-profile mixers, local heat transfer coefficients were determined for $x/H \geq 5$ in the streamwise direction (noted: x is measured from the leading edge of the leading surface). The experiments were conducted at a constant heat flux condition with Reynolds numbers of 7000, 12000 and 16200, respectively. The results obtained from these experiments are shown in Figures 4.18 - 4.20. For comparison, results obtained at 0 and 90 ppm surfactant concentrations without the low-profile mixers are also shown in the figures.

At a Reynolds number of 7000 (Figure 4.1 S), the Nusselt number obtained at 90 ppm for the case with mixers is about 190% higher than the case without the mixer. It is also 70% higher than that for pure water flow without the use of the mixers. The maximum attainable Nusselt number changes from 30 to about 87, with the latter value being much higher than the value obtained by Gnielinski (1976) in a fully developed channel flow, where a value of 55 was reported. The significant increase in heat transfer rate might be related to the suction and ejection field generated by the low-profile mixers, resulting in bulk exchange of fluid particles between the main flow and the near-wall region. In the region of $0 < x/H < 5$ where no mixers were installed, the flow is not influenced by the device. Thus, the heat transfer remain the same as when the mixer were not used.

Figure 4.18 - Streamwise distribution of local Nusselt number with low-profile mixers

(Re = 7000)

When Reynolds number increases to about 12000 (Figure 4.19), the local Nusselt number with mixers in 90 ppm condition increases from 87 at Re = 7000 to about 95, which is about 200% higher than the case when mixers were not used. This result indicates that the heat transfer rate downstream of the mixers increases with the increase in Reynolds number. The heat transfer coefficient with the mixers is about 30% higher than pure water, which is much lower than for the case of Re = 7000. This is because the local Nusselt number in pure water is higher at Re = 12000 than at Re = 7000.

Figure 4.19 - Streamwise distribution of the local Nusselt number with low-profile mixers

($Re = 12000$)

With Reynolds number of about 16200 (Figure 4.20), the local Nusselt number with mixers in 90 ppm increases further to about 100, which is about 190% higher than the case without the mixers. The Nusselt number downstream of the mixers is about 25% higher than in pure water. This result seems to suggest that the performance of the mixers in 90 ppm surfactant solution relative to the pure water flow decreases with the increase of Reynolds number. Based on this trend, if Reynolds number of the experiment is allowed to increase further, the heat transfer rate under the conditions of 0 ppm without mixers and 90 ppm with mixers may merge.

Figure 4.20 - Streamwise distribution of the local Nusselt number with low-profile mixers
(Re = 16200)

The results presented in the form of local Nusselt number in Figures 4.18 - 4.20 show the deterioration of heat transfer enhancement in the streamwise direction. It is noted that the influence of the low-profile mixers to the heat transfer rate is not applicable for all downstream locations. For instance at $x/H = 7$ in Figure 4.18, heat transfer enhancement starts to decline. The location of the onset of this decline shifts further downstream with an increase in Reynolds number. For example, at Reynolds number of 16200 (Figure 4.20), the onset of decline in heat transfer enhancement starts only at $x/H = 8.4$. The degradation of heat transfer enhancement might be due to two reasons: one may be related to the loss of the spiraling momentum that sustains the longitudinal vortices while the other reason may be due to the loss of kinetic energy from the longitudinal vortices to the slower flow region near the heating surface.

In comparison with the results of previous investigations in heat transfer enhancement in drag-reducing flow, the effectiveness of the present technique is found to be better. For example, Eschenbacher *et al.* (2001) with a triangular winglet mounted on the flat heating surface were able to recover the heat transfer to the level similar to the case without surfactant (with $Nu \approx 55$) at Reynolds number of 7000. On the other hand, with the use of the present technique, heat transfer is enhanced ($Nu \approx 85$) to a level that is about 70% higher than that for the case without surfactant.

In the experimental studies by Nakabe *et al.* (1998) and Li *et al.* (2001j), an impinging jet and a mesh-screen inserted in the channel were deployed, respectively, to enhance the heat transfer rate in the drag-reducing flow. The maximum attainable heat transfer enhancements by these respective techniques are about 100% and 200% compared to the case without the enhancement device. For the present technique, an enhancement of heat transfer rate of about 200% in drag-reducing flow was achieved, within the experimental range of Reynolds numbers. Although the method used by Nakabe *et al.* (1998) could only achieve a heat transfer enhancement of about 100% relative to the case without the enhancement device, due to the enhancement mechanism which corresponds very well to the behaviour of the longitudinal vortices, it contributes very little to the overall drag. On the other hand the method of Li *et al.* (2001) may not be very practical due to the large pressure drop caused by the screen-mesh, even though it can increase the heat transfer by 200%. Furthermore, such a high enhancement rate of Li *et al.* (2001) is only possible at high Reynolds numbers ($Re > 13000$). For Reynolds number lower than this value, the shear effect in the flow may not be sufficient to break down the orderly aligned rod-like micelles. In practical applications, when the dimensions of the heat exchangers are normally small, the Reynolds number in the

exchangers is low. Therefore, the present method employing in heat exchangers may be more practical.

CHAPTER FIVE: Conclusions and Recommendations for Future Work

5.1 Conclusions

In the present study, a two-dimensional water tunnel with dimensions of 0.5 m (width) $\times$ 0.05 m (height) $\times$ 6 m (length) (with an aspect ratio of 10:1) was designed and fabricated to investigate drag and heat transfer in a fully developed drag reducing flow.

Drag reduction was examined for four different surfactant concentrations, namely 30, 70, 80 and 90 ppm over a Reynolds number range of 7000–18500. The maximum achievable drag reductions for the four different surfactant concentrations are 7, 30, 50 and 54% respectively. The critical Reynolds number, Re_{cri} , increases with an increase in surfactant concentrations. At 70 ppm surfactant concentration, $Re_{cri} = 13000$ while it is 17000 for 90 ppm.

The heat transfer coefficients were examined for surfactant concentrations of 0, 30, 70, 80 and 90 ppm respectively at the Reynolds numbers of 7000, 12000, 16200 and 18500. At all these Reynolds numbers, there is no remarkable reduction of heat transfer with 30 ppm of surfactant concentration in the flow. At a Reynolds number of 7000, the heat transfer reduction is only about 15% when 70 ppm of surfactant is present in the flow. Its thermal developing region ends at around $x/H = 2.5$. Beyond this point, the local Nusselt numbers remain consistent. By increasing the surfactant concentration in the flow to 80 and 90 ppm, remarkable heat transfer reductions (–40%) are noted.

When Reynolds number is increased to 12000, dramatic heat transfer reduction occurred for surfactant concentrations of 70, 80 and 90 ppm. With 70 ppm, the local heat transfer

distribution dropped by about 33% compared with pure tap water. Its thermal boundary layer became fully developed at $x/H = 3$. By increasing the surfactant concentration in the flow to 80 and 90 ppm, further heat transfer reduction by about 60% was noted. For these two concentrations, the developing length of the thermal boundary layer extends to $x/H = 6$.

At $Re = 16200$, with surfactant concentrations of 0 and 30 ppm, the local Nusselt numbers agree well with each other. With 70 ppm of surfactant, the heat transfer is reduced by 15% and recovered to the value close to that of pure tap water at $x/H \geq 4$. However, with 80 and 90 ppm of surfactant in the flow, the heat transfer reduction can be as high as 55%.

At $Re = 18500$, the reduction of heat transfer with 70 ppm becomes negligible after $x/H = 2.5$. By increasing the surfactant concentration to 80 ppm, heat transfer reduction is apparent at the beginning of the heating surface (for $x/H \leq 4$) and starts to recover slightly after $x/H = 4$. The results for 70 and 80 ppm of surfactant concentrations seem to suggest that the rod-like micelle structures may be broken down at certain downstream locations, resulting in an increase in the heat transfer effect rates. At 90 ppm, the maximum achievable heat transfer reaches 55%.

A row of low-profile-mixer was installed in the spanwise direction on the heating surface to enhance the heat transfer rate for a surfactant concentration of 90 ppm at $Re = 7000$, 12000 and 16200. At $Re = 7000$ in the drag-reducing flow, the Nusselt number obtained with mixers is about 190% higher than the case without mixers. This significant increase in heat transfer rate may be related to the suction and ejection fields generated by the low-profile-mixers, resulting in bulk exchange of fluid particles between the main flow and the near-wall region.

When the Reynolds number is increased to 12000, the local Nusselt number with mixers increases by 200%. With Reynolds number being equal to 16200, the local Nusselt number with mixers is also 190% higher than the case without mixers. The present results have also shown that the heat transfer rate is enhanced by 32~75%, depending on the magnitude of Reynolds number, relative to pure tap water due to the use of the low-profile-mixer. In comparison with the work of other researchers, this result is found to be better.

The local Nusselt numbers for all Reynolds numbers show a deterioration of heat transfer enhancement. It is noted that the influence of the low-profile-mixers to the heat transfer rate is not sustainable throughout the whole heating plate. For instance at $Re = 7000$ (Figure 4.18), the heat transfer enhancement starts to decline at $x/H = 7$. The location of the onset of this decline shifts further downstream with an increase in the Reynolds number. For example, at Reynolds number of 16200 (Figure 4.20), the onset of decline in heat transfer enhancement starts only at $x/H = 8.4$. The degradation of heat transfer enhancement might be due to two reasons: one may be related to the loss of spiraling momentum that sustains the longitudinal vortices while the other reason may be due to the loss of kinetic energy from the longitudinal vortices to the slower flow region near the heating surface.

5.2 Recommendations for future work

Both passive and active approaches on heat transfer enhancement in drag-reducing flow should be investigated in the future to elucidate the mechanism of these techniques.

The experimental study has shown that low-profile-mixer can enhance heat transfer effectively. Thus, further investigations on the flow field behind the low-profile-mixers

should be performed with the use of Particle Image Velocimetry (PIV) to further understand the enhancement mechanism. The PIV technique enables one to capture the instantaneous velocity field in the flow.

Only a few active techniques were used previously to study the enhancement of heat transfer in drag-reducing flow. Qi *et al.* (2003) used ultrasonic energy to enhance the heat transfer rate. The ultrasonic energy was used to break down the microstructure of the micelles. They found that the effectiveness of this technique on the enhancement depends on the surface tension, fluid viscosity, solution temperature, morphology of the micelles and intensity of the ultrasonic energy. Another active approach is to use an impingement device such as that employed by Nakabe *et al.* (1998). In their experimental study, an impinging jet was applied into the flow at a cross-flow direction with an angle of 45° from the bottom wall. The nozzle of the jet was mounted flush to the tunnel wall. It was reported that this method can enhance heat transfer rate over a wider region when compared with the case of a conventional impinging jet nozzle normal to the target wall.

Another possible active enhancement method is by embedding arrays of micro actuators on the heating surface. The deployment of such micro actuators to enhance heat transfer rate in the drag-reducing flow has not been reported in the literature.

Some of the feasible ways to fabricate micro actuators is by forming arrays of micro-balloons, micro-flaps, micro-syntactic jets and micro nozzles. The main purpose of the actuators is to add a small amount of external energy to the near wall flow to disturb the flow structure so that the rod-like micelles structures can be broken down to spherical-like

micelles, resulting in the enhanced heat transfer rate. Out of the four ways, the deployment of micro balloons as a heat transfer enhancement device seems to be most feasible and easiest to fabricate.

5.2.1 Design of micro-actuators for active heat transfer enhancement

Basically, the micro actuators consist of arrays of micro balloons and a precise pressure regulating system. The micro balloons will interact with the flow close to the wall, while the precision pressure regulating system controls the movement of the micro balloons. The height of the balloon is of order of millimetres. It is desired to design the actuators as small as possible so that it will only interact with the flow structure near the heating surface rather than the main flow.

One method to fabricate these micro balloons is by laminating a thin flexible membrane onto a solid surface with through holes drilled on it so that when there is a pressure difference between the membrane and the flow close to the wall, semi-sphere-like balloons will be formed.

The membrane selected to fabricate these actuators must be elastic and inert to chemicals so that when it contacts with the surfactant in the flow it will not degrade, but continue to maintain its flexibility to form semi-sphere-like balloons.

One of the possible raw materials that can be used is silicone rubber. Silicone rubber can be easily purchased in different forms, namely, paste form, liquid form with hardening catalyst for solidification and in thin sheet form. The thinnest sheet available in the market is about

0.5-mm-thick. The thickness of the silicone rubber is critical because it influences the membrane thermal resistance, which will retard heat transfer rather than enhancing it. The available thickness may be too large for this application. Therefore, only the paste form or liquid form with hardening catalyst for solidification is appropriate as the thickness of the membrane can be controlled during fabrication.

Another factor that needs to be considered during fabrication of the micro-ballons is the uniformity of the membrane thickness across the heating surface. The size of the individual micro actuator must be uniform over the whole heating plate. Thus, during the installation of the silicone rubber, special care should be exercised to maintain a constant thin-layer over the entire heating plate.

One possible design of the actuators is shown in Figure 5.1. The flat heating plate is covered by a thin layer of silicone rubber. The plate is heated with fine element attached firmly onto the bottom surface to ensure uniform heating throughout the whole surface.

The silicone rubber will generate micro balloons if the pressure in the air chamber increases. By manipulating the pressure difference on the two sides of the silicone rubber, the motion and size of the micro-actuators can be controlled easily with a precision pressure regulating system. For example, if a stationary bump of specified height is required, the pressure at the higher-pressure-side will be adjusted and locked to maintain the size and the shape of the bump. If a dynamic bump is required, the precision pressure regulating system can be switched to dynamic mode where a wave generator is used to create a pulsating pressure to

manipulate the movement of the actuators. A flow chart of the pressure regulating system for the micro actuators is shown in Figure 5.2.

Figure 5.1 – Conceptual design of the micro-actuators

Figure 5.2 – Precision pressure regulating system for the micro-actuators

5.2.2 Proposed experiment for the micro-actuators

The design of the micro-actuators allows the enhancement of the heat transfer to be achieved in two different modes, i.e. passive and active. By setting and fitting the balloon to different heights, it becomes a passive enhancement device. By manipulating the pressure difference across the membrane, the behaviour of the actuators can be controlled in a desired manner and frequency. In this case, it can be used as an active heat transfer enhancement device. For the passive mode experiments, focus may be placed on optimising the height of the micro actuators. The experiments can be performed with different actuator heights at various Reynolds numbers and surfactant concentrations to obtain higher heat transfer rate in the flow. As for the active mode experiment, varying the shape of the excitation wave and its frequency will allow one to optimise the heat transfer enhancement at a particular flow condition. For instance at a fixed Reynolds number, the surfactant concentration, temperature and the resonant frequency of the channel flow system can be fixed. By varying the frequency of the excitation wave, the optimal operation mode of the pressure regulating system may be obtained.

5.2.3 Proposal for a new water tunnel facility

The present water tunnel facility used to perform all experiments in this report was fabricated from resources available at that time. Problems such as leakage, vibration, flow separation in the piping system when the facility run at high flowrate and limited range of Reynolds numbers have been encountered during the experiments. There is therefore, need to redesign and re-fabricate a whole new water tunnel facility to support future experimental studies.

A three-dimensional view of the new water tunnel is shown in Figure 5.3. The tunnel has a 4.5-m-long rectangular duct with a cross-section area of 400 mm x 30 mm (the aspect ratio is 13.33 : 1). A stainless steel water pump (maximum flow rate: 14 L/s) is used to re-circulate the fluid in the facility. With this configuration, the highest Reynolds number achievable in the test section is about 40000.

A heating section situated at the end portion of the tunnel consists of a flat heating surface installed on the top of the tunnel wall. 64 pairs of thermocouples can be embedded in the heater to measure the wall temperature in the streamwise direction. The heater can be removed easily from the facility.

Figure 5.3 – Three-dimensional view of the proposed water tunnel

A list of the design requirements is given below.

- Dimensions of the tunnel: 300-mm-height, 400-mm-width and 4.5 m long.
- Side walls are made of 25-mm-thick transparent material.
- Dismantleable top wall.
- Pressure tapping points on the bottom wall along the 4.5-m-long tunnel.
- Smooth contracting entrance tank.
- Equipped with adjustable “Bolt & Nuts” for tunnel alignment.
- Separated supporting structure for entrance section.
- Separated supporting structure for water pumping system.
- One stainless steel water pump with maximum flow rate of 14 L/s.
- Frequency inverter with over load protector for water pump controller.
- PVC piping system.
- All structures, entrance tank and reservoir tank must be powder coated to prevent corrosion.
- Entrance and reservoir tank must be made from stainless steel sheets.
- Screens and honeycomb must be installed at the entrance tank to remove larger eddy flow structures before the fluid enters the tunnel.
- All electrical components must be installed in a standard panel with safety protection features (e.g. electrical leakage, overload protection for water pump)
- The 4.5-m-long tunnel must be levelled horizontally with a tolerance of ± 0.25 mm.

The features of the components are summarised in Table 5.1

Table 5.1 – Summary of the main feature of each component

Items:	Descriptions:	Remarks:
A:	Settling Tank (Include: Honeycomb and Contraction)	Stabilise flow before entering into the tunnel.
B:	4.5-m-Long Two-dimensional Tunnel (Inner Dimensions: 4.5-m-length $\times$ 0.4-m-width $\times$ 0.03-m-height)	
C:	Storage Tank	Capacity: $\sim 0.8 \text{ m}^3$
D:	Stainless Steel Water Pump	52 m^3/hr
E:	Electromagnetic Flow Meter	$\pm 0.5\%$ of measured value for flow above 0.5 m/s
F:	Drainage	
G:	Spacer	Maintain constant spacing between top and bottom wall
H:	Adjustable leg	Levelling of the whole facility

Figure 5.4 elaborates the overall dimensions of the water tunnel facility. The total length of the facility is 7.08 m and its width is 0.76 m. The overall dimensions are determined by the available space in the laboratory. Table 5.2 compares the proposed water tunnel facility with those used by other researchers.

Figure 5.4 - External dimensions of the proposed water tunnel

Table 5.2 -Comparison of the proposed water tunnel facility with those used by other researchers

No.	Features	Proposed Water Tunnel Facility for Future Work	Existing Water Tunnel Facility	Water Tunnel Facility used by Kawaguchi <i>et al.</i> (2002)	Water Tunnel Facility used by Eschenbacher, <i>et al.</i> (2001)
1.	Aspect Ratio (W/H)	13.33	10	12.5	20
2.	Hydrodynamic Entry Length (L/H)	133.33	100	125	204
3.	$Re_{\max} = \frac{U_b \cdot H}{\nu}$	~40 000	~19 000	~70 000	~44 000
4.	Tunnel Wall Thickness	25 mm	12 mm	20 mm	Information not provided
5.	Tunnel Width $\times$ Height	400 $\times$ 30 mm	500 $\times$ 50 mm	500 $\times$ 40 mm	500 $\times$ 25 mm
6.	Tunnel Material	Transparent Acrylic Resin (Full tunnel-length)	Polycarbonate (Full tunnel-length)	Transparent Acrylic Resin (at test section only)	Transparent Acrylic Resin (Full tunnel-length)
7.	Flow Rate Control	Frequency Inverter	Frequency Inverter	Valves	Valves

*Legends:*Width of Tunnel: W Length of Tunnel: L Height of Tunnel: H Average Velocity: U_A Kinematic Viscosity: ν

REFERENCES

- Aguilar, G., Gasljevic, K. and Matthys, E. F. (2001). Asymptotes of Maximum Friction and Heat Transfer Reductions for Drag-Reducing Surfactant Solutions. *International Journal of Heat and Mass Transfer*, 44, pp. 2835.
- Ahuja, K. K. and Burrin, R. H. (1984). Control of Flow Separation by Sound. *AIM Paper* New York, pp. 2284.
- Ashill, P. R., Fulker, J. L. and Hackett, K. C. (2001). Research At DERA on Sub Boundary Layer Vortex Generators (SBVGs). *AIM Paper 39th Aerospace Sciences Meeting and Exhibit*, Reno, NV, pp. 887.
- Bergles, A. E. (1981). *Application of Heat Transfer Augmentation*. Hemisphere.
- Bewersdorff, H. W. and Gyr, A. (1998). Drag Reduction by Polymers and Surfactants. In Rys, F. and Gyr, A., *Progress in Physical Processes and Chemical Reactions in Liquid Flows* Switzerland, pp. 209.
- Candau, S. J. and Oda, R. (1998). Viscoelasticity of Salt Free Micellar Systems: Effect of the Packing Parameter. 216th *ACS National Meeting*, Boston.
- Candau, S. J. and Oda, R. (2001). Linear Viscoelasticity of Salt-Free Worm-Like Micellar Solutions. *Colloids and Surface: Physico-chemical Engineering Aspects*, 183, pp. 5.

Chen, Y. C., Ma, C. F., Yuan, Z. X., Xia, Z. Z., and Guo, Z. Y. (2001). Heat Transfer Enhancement with Impinging Free Surface Liquid Jets Flowing over Heated Wall coated by a Ferrofluid. *International Journal of Heat and Mass Transfer*, 44, pp. 499.

Choi, H., Moin, P. and Kim, J. (1993). Direct Numerical Simulation of Turbulent Flow Over Riblets. *Journal of Fluid Mechanics*, 255, pp. 503.

Choi, H., Moin, P. and Kim, J. (1994). Active Turbulence Control for Drag Reduction in Wall-Bounded Flows. *Journal of Fluid Mechanics*, 262, pp. 75.

Chou, L. C., Christenen, R. N. and Zakin, J. L. (1989). The Influence of Chemical Composition of Quaternary Ammonium Salt Cationic Surfactants on their Drag Reducing Effectiveness. In: Sellin RHJ, Moses RT, editors. *Drag Reduction Fluid Flows*, pp. 141.

Dean, R. B. (1978). Reynolds Number Dependence of Skin Friction and Other Bulk Flow Variables in Two-Dimensional Rectangular Duct Flow. *Journal of Fluids Engineering*, 100, pp. 215.

Eschenbacher, J. F., Joko, M., Nakabe, K. and Suzuki, K. (2001). Heat Transfer in the Wake Behind a Longitudinal Vortex Generator Immersed in Drag-Reducing Channel Flows. *Journal of Visualization*, 4, pp. 262.

Fossa, M. and Tagliafico, L. A. (1995). Experimental Heat Transfer of Drag-Reducing Polymer Solutions in Enhanced Surface Heat Exchangers. *Experimental Thermal and Fluid Science*, 10, pp. 221.

Gasljevic, K. and Matthys, E. F. (1993). On Saving Pumping Power in Hydronic thermal Distribution Systems through the Use of Drag-Reducing Additives. *Energy and Buildings*, 20, pp. 45.

Gasljevic, K. and Matthys, E. F. (1997). Experimental Investigation of thermal and Hydrodynamic Development Regions for Drag-Reducing Surfactant Solutions. *Journal of Heat Transfer*, 119, pp. 80.

Gasljevic, K., Aguilar, G. and Matthys, E. F., (2000). Buoyancy Effects on Heat Transfer and Temperature Profiles in Horizontal Pipe Flow of Drag-reducing Fluids. *International Journal of Heat and Mass Transfer*, 43, pp. 4267.

Glezer, A. (1999). Shear Flow Control Using Fluidic Actuator Technology. *1st Symposium on Smart Control of Turbulence*, Tokyo, pp. 39.

Gnielinski, V. (1976). New Equations for Heat and Mass Transfer in Turbulent Pipe and Channel Flow. *International Chemical Engineering*, 16, pp. 359.

Gupta, J. P. (1985). *Fundamentals of Heat Exchanger and Pressure Vessel Technology*. Hemisphere.

Hsiao, F. B., Liu, C. F. and Shyu, J. Y. (1990). Control of Wall-Separated Flow by Internal Acoustic Excitation. *AIAA Journal*, 28, pp. 1440.

Imai, S. and Shikata, T. (2001). Viscoelastic Behavior of Surfactant Thread-Like Micellar Solutions: Effect of Additives. *Journal of Colloids Interface Science*, 244, pp. 399.

Incropera and DeWitt (1996). *Fundamental of Heat and Mass Transfer*. Wiley, Fourth Edition.

Ingard, U. (1953). On the Theory and Design of Acoustic Resonators. *Journal of Acoustical Society of American*, 25, pp. 1037.

Jenkins, L., Gorton, S. A. and Anders, S. (2002). Flow Control Device Evaluation for an Internal Flow with an Adverse Pressure Gradient. *AIAA Paper*, 40th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, pp.0266

Kawaguchi, Y., Daisaka, H., Li, P. W., Yabe, A., Hishida, K. and Maeda, M. (1997). Study on a Thermal Boundary Layer of Drag Reducing Surfactant Solution – Measurements of Temperature Fluctuation. *Proceedings of the ASME Fluids Engineering Division*, 244, pp. 375.

Kawaguchi, Y., Segawa, T., Feng, Z. P. and Li, P. W. (2002). Experimental Study on Drag-Reducing Channel Flow with Surfactant Additives-Spatial Structure of Turbulence Investigated by PIV System. *International Journal of Heat and Fluid Flow*, 23, pp. 700.

Kim, J. (1992). Study of Turbulence Structure Through Numerical Simulations: the Perspective of Drag Reduction. In Agard Report (R-786), Agard FDP/VKI Special Course on "Skin Friction Drag Reduction", March 2-6, Vki, Brussels, Belgium.

Kim, W. J. and Yang, S. M. (2000). Microstructures and Rheological Response of Aqueous CTAB Solutions in the Presence of Benzyl Additives. *Langmuir*, 16, pp. 6084.

Kline, S. J. and McClintok, F. A. (1953). Describing Uncertainties in Single-sample Experiments. *Mechanics Engineering*, pp. 3.

Kravchenko, A. G., Choi, H. and Moin, P. (1993). On the Relation of Near-Wall Streamwise Vortices to Wall Skin Friction in Turbulent Boundary Layers. *Physical Fluids*, 5, pp. 3307

Li, F. C., Kawaguchi, Y., Hishida, K. (2003). Investigation on Heat Transfer Characteristics of Drag-Reducing Flow with Surfactant Additive by Simultaneous Measurements of Temperature and Velocity Fluctuations in Thermal Boundary Layer. 6th ASME-JSME Thermal Engineering Joint Conference, March 16-20.

Li, P. W., Kawaguchi, Y. and Yabe, A. (2000). Feasibility Study of New Heat Transportation System with Drag-Reducing Surfactant Additives. *Proceedings of Symposium on Energy Engineering in the 21st Century*, Hong Kong, 1/9-13, pp. 716.

Li, P. W., Kawaguchi, Y., Daisaka, H., Yabe, A., Hishida, K. and Maeda, M. (2001). Heat Transfer Enhancement to the Drag-Reducing Flow of Surfactant Solution in Two-Dimensional Channel with Mesh-Screen Inserts at the Inlet. *Journal of Heat Transfer*, pp. 779.

Lin, J. C., Howard, F. G., Bushnell, D. M. and Selby, G. V. (1990). Investigation of Several Passive and Active Methods for Turbulent Flow Separation Control. *AIAA Paper, AIAA 21st Fluid Dynamics, Plasma Dynamics and Lasers Conference*, Seattle, WA, June 18 - 20, pp. 1598.

Lin, J. C., Howard, F. G. and Selby, G. V. (1990). Small Submerged Vortex Generators for Turbulent Flow Separation Control. *Journal Space Rockets*, 27, pp. 503.

Lin, J. C., Howard, F. G. and Selby, G. V. (1991). Exploratory Study of Vortex Generating Devices for Turbulent Flow Separation Control. *AIAA Paper, AIAA 29th Aerospace Sciences Meeting*, Reno, NV, Jan 7-10, pp. 42.

Lin, J. C. (1999). Control of Turbulent Boundary Layer Separation using Micro-Vortex Generators. *AIAA Paper, 30th AIAA Fluid Dynamics Conference*, Norfolk, VA, Jun 28 – July 1, pp. 3404

Lin, J. C. (2002). Review of Research on Low-Profile Vortex Generators to Control Boundary-Layer Separation. *Progress In Aerospace Sciences*, 38, **pp.** 389.

Lin, Z., Zheng, Y., Davis, H. T., Scriven, L. E., Talmon, Y., and Zakin, J. L. (2000). Unusual Effects of Counterion to Surfactant Concentration Ratio on Viscoelasticity of a Cationic Surfactant Drag Reducer. *Journal of Non-Newtonian Fluid Mechanics*, 93, pp. 363.

Lin, Z., Lu, B. and Zakin, J. L. (2001). Influence of Surfactant Concentration and Counterion to Surfactant Ratio on Rheology of Wormlike Micelles. *Journal of Colloids Interface Science*, 239, pp. 543.

Liu, C. (1999). Out-of-Plane Magnetic Actuators with Electroplated Permalloy for Fluid Dynamics Control. *Sensors and Actuators*, 78, pp. 1997.

Magid, L. J., Han, Z., Li, Z. and Butler, P. D. (2000). Tuning the Contour Lengths and Persistence Lengths of Cationic Micelles: The Role of Electrostatics and Specific Ion Binding. *Journal Physical Chemical*, 104, pp. 6717.

Matthys, E. F. (1991). Heat Transfer, Drag Reduction and Fluid Characterization for Turbulent Flow of Polymer Solutions: Recent Results and Research Needs. *Journal of Non-Newtonian Fluid Mechanics*, 38, pp. 313.

Matthys, E. F. (1997). Heat Transfer to Non-Newtonian Fluids. In Cengel, Y. A., *Introduction to Thermodynamics and Heat Transfer*, McGraw-Hill, pp. 563.

McCormick, D. C. (1992). Shock-Boundary Layer Interaction Control with Low-Profile Vortex Generators and Passive Cavity. *AIAA Paper*, 30th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, Jan 6-9, pp. 64.

Menger, F. and Persypkin, A. (2001). A Combinatorially-Derived Structural Phase Diagram for 42 Zwitterionic Geminis. *Journal American Chemical Society*, pp. 5615.

Michael, D., Warholic, Gavin, M., Schmidt and Thomas, J. H. (1999). The Influence of a Drag-Reducing Surfactant on a Turbulent Velocity Field. *Journal of Fluid Mechanics*, 388, pp. 1.

Mounts, J. S. and Barber, T. J. (1992). Numerical Analysis of Shock Induced Separation Alleviation using Vortex Generators. *AIAA Paper, 30th AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, Jan 6-9, pp. 751.

Moffat, R. J. (1982). Contributions to the Theory of Single-Sample Uncertainty Analysis. *Journal of Fluid Engineering*, 104, pp. 250.

Myska, J. and Stern, P. (1998). Significance of Shear Induced Structure in Surfactant for Drag Reduction. *Colloid Polyscience*, 276, pp. 816.

Myska, J. and Chara, Z. (2001). The Effect of a Zwitterionic and Cationic Surfactant in Turbulent Flows. *Experiments in Fluids*, 30, pp. 229.

Myska, J., Lin Z., Stepanek, P. and Zakin, J. L. (2001). Influence of Salts on Dynamic Properties of Drag Reducing Surfactants. *Journal of Non-Newtonian Fluid Mechanics*, 97, pp. 251.

Myska, J. and Mik, V. (2003). Application of a Drag Reducing Surfactant in the Heating Circuit. *Energy and Building*, 35, pp. 813.

Nakabe, K., Higashio, A., Chen, W., Suzuki, K. and Kim, J. H. (1998). An Experimental Study of the Flow and Heat Transfer Characteristics of Longitudinal vortices induced by an Inclined Impinging Jet into a Crossflow. *Proceedings 11th International Heat Transfer Conference*, 5, pp. 439.

Ohlendorf, D., Interthal, W. and Hoffmann, H. (1986). Surfactant System for Drag Reduction: Physico-Chemical Properties and Rheological Behavior. *Rheology Acta*, 25, pp. 468.

Pollert, J., Zakin, J. L., Myska, J. and Kratochvil, P. (1994). Use of Friction Reducing Additives in District Heating System Field Test at Kladono Krocehlavy. *Proceedings of International District Heating and Cooling Conference*, Czech Republic, pp. 156.

Potter, M. C. and Wiggert, D. C. (1997). *Mechanics of Fluids*, 2nd Edition. Prentice Hall.

Qi, Y. Y., Kawaguchi, Y., Lin, Z. Q., Ewing, M., Christensen, R. N. and Zakin, J. L. (1999). Enhanced Heat Transfer of Drag Reducing Surfactant Solutions. *Proceedings of the 11th European Drag Reduction Working Meeting*, Prague, Czech Republic, pp. 42.

Qi, Y. Y., Kawaguchi, Y., Lin, Z. Q., Ewing, M., Christensen, R. N. and Zakin, J. L. (2001). Enhanced Heat Transfer of Drag Reducing Surfactant Solutions with Fluted Tube-In-Tube Heat Exchanger. *International Journal of Heat and Mass Transfer*, 44, pp. 1495.

Qi, Y. Y., Weavers, L. K. and Zakin, J. L. (2003). Enhancing Heat Transfer Ability of Drag-Reducing Surfactant Solutions with Ultrasonic Energy. *Journal of Non-Newtonian Fluid Mechanics*, 116, pp. 71.

Raghavan, S. R. and Kaler, E. W. (2001). Highly Viscoelastic Wormlike Micellar Solutions formed by Cationic Surfactants with Long Unsaturated Tails. *Langmuir*, 17, pp. 300.

Raghavan, S. R., Edlund, H. and Kaler, E. W. (2002). Cloud-Point Phenomena in Wormlike Micellar Systems Containing Cationic Surfactant and Salt. *Langmuir*, 18, pp. 1056.

Robinson, S. K. (1991). Coherent Motions in the Turbulent Boundary Layer, *Annual Review Fluid Mechanics*, 23, pp. 601.

Roethig, T. R. and Matthys, E. F. (1988). Eddy Diffusivities for Non-Isothermal Tube Flow of Drag-Reducing Polymer Solutions, *Proceedings Xth International Congress on Rheology*, Sydney, Australia, Australian Society of Rheology, Sydney, Aug 14-19, pp. 227.

Tennekes, H. and Lumley, J. L. (1972). *A First Course In Turbulence*. MIT Press.

Tien, C. P. (2001). Keeping Cool. *Straits Times*, Source: Singapore JTC International.

Toms, B. A. (1948). Some Observations on the Flow of Linear Polymer Solutions through Straight Tubes at Large Reynolds number. *Proceedings of 1st International Congress on Rheology*, 2, pp. 135.

Torii, K., Kwak, K. M. and Nishino, K. (2002). Heat Transfer Enhancement Accompanying Pressure-loss Reduction with Winglet-type Vortex Generators for Fin-tube Heat Exchangers. *International Journal of Heat and Mass Transfer*, 45, pp. 3795.

Warholic, M. D. (1997). *Drag Reduction in Turbulent Channel Flow by Additives*. PhD Thesis, University of Illinois, Urbana-Champaign.

Webb, R. L. (1995). *Principles of Enhanced Heat Transfer*. Hemisphere.

Yang, J. (2002). Viscoelastic Wormlike Micelles and their Applications. *Current Opinion in Colloids and Interface Science*, 7, pp. 276.

APPENDIX A

Dimensions of entrance section

APPENDIX B

Dimensions of reservoir

APPENDIX C

Water pump performance graph

(Extracted from CombiMag Catalogue)

APPENDIX D

Detail calculation of heater losses

An analysis on the heat loss, q_L was done initially. However, it was found to be relatively small (4.8 W) as compared with the heat transfer loss in the upward direction (442.78 W). Therefore, q_L was not taken into consideration for the analysis in this thesis. The following is the detailed estimation of the heat loss:

The relation derived to estimate q_L is represented in the form of an equivalent thermal circuit as shown:

Equivalent thermal circuit of heater insulation

R_c is the thermal contact resistance between the ceramic insulator and the stainless steel back-plate. L_c is the thickness of the 6.7-mm-thick low-density ceramic board, L_{ss} is the thickness of steel back-plate, k_c and k_{ss} refer to the thermal conductivity of the ceramic board and stainless steel back-plate respectively, T_1 is the heater temperature and T_2 is the outer wall temperature (back of the heating assembly). Both were measured directly with the embedded thermocouples. The magnitude of q_L can be calculated using the following relation derived from the equivalent thermal circuit shown above.

$$q_L = \frac{T_1 - T_2}{\frac{L_{ss}}{k_{ss} \times A_s} + \frac{R_c}{A_s} + \frac{L_{ss}}{k_{ss} \times A_s}} \quad , \quad (A)$$

where L_c is 0.0067 m, k_c is 0.045 W/mK, L_{ss} is 0.002 m, k_{ss} is 14.9 W/mK and A_s is 0.24 m². In this experimental set-up, the lamination was clamped tightly and R_c was assumed to be small compared with the other resistances, Thus Eq. (A) can be simplified as:

$$q_L = \frac{T_1 - T_2}{0.62} \quad (B)$$

During the experiments, $T_1 - T_2$ was measured to be about 3°C when $\text{Re} = 10000$. It was observed that when $\text{Re} = 0$, $T_1 - T_2$ increased dramatically. Therefore, $T_1 - T_2$ was taken as 3°C to estimate the heat loss.

APPENDIX E

Lamination clamping device

Back view of heater assembly

APPENDIX F

Publications arising from this thesis

Zhou, T., Leong, K.C. and Yeo, K.H. "Experimental study of heat transfer enhancement in a drag-reducing two-dimensional channel flow", *International Journal of Heat and Mass Transfer*, under review, 2005.

Yeo, K.H., Zhou, T. and Leong, K.C., "Experimental study of passive heat transfer enhancement in drag-reducing flow", *Heat Transfer in Components and Systems for Sustainable Energy Technologies, Proceedings of the Heat SET 2005 Conference*, 5-7 April 2005, Grenoble, France, pp. 519 – 524, ed. B. Thonon.

Yeo, K.H., Zhou, T. and Leong, K.C., "Experimental studies on drag reduction and heat transfer enhancement in a two-dimensional turbulent channel flow", *International Conference on Jets, Wakes and Separated Flows*, ICJWSF-2005, October 5-8, 2005, Toba-shi, Mie, Japan, accepted for presentation.