21st Australasian Fluid Mechanics Conference
Adelaide, Australia
10-13 December 2018

Investigation of heat transfer enhancement in dimpled pipe flows

A. W. Albanesi[1], K. D. Daish[1] B. Dally[1] and R. C. Chin[1]

[1]School of Mechanical Engineering University of Adelaide, South Australia 5005, Australia

Abstract

It is well established that dimpling of tubes can help enhance the heat convection coefficient. The effects of the dimples shape, depth and distribution on heat transfer and pressure drop for laminar and turbulent flows are of interest to this study. The aim of this paper is to quantify the effects of dimpling on the performance of 11mm ID tubes used in a submarine shell and tube heat exchanger. Both experiments and calculations were conducted, using water as the working fluid, over a wide range of Reynolds numbers ranging from 1500 to 24000, based on bulk velocity and tube diameter. Four tubes with different dimpling arrangements were tested. The CFD results show that a tube with 2mm deep single dimples arrangement, operating in the laminar flow regime, gives the maximum improvements in heat transfer of 10.23% over the smooth tube. The largest pressure drop increase recorded was 102.8% under identical flow conditions for the same dimpled tube. Temperature contour plots were used to demonstrate the mixing effect of the dimples under laminar flow conditions inside the tube.

Introduction

Heat exchangers are used in vast engineering applications [7]. One of the most common configuration for heat exchangers is the shell and tube combination. Various methods have been proposed in the literature to improve the performance of heat exchangers, such as, surface geometric modifications of the pipes. It is well established that dimpling of a pipes surface leads to higher heat transfer rate. The dimples in the pipe essentially have a tripping effect on the fluid flow leading to an increase in mixing of the fluid and reduces the growth of the thermal boundary layer. However, with such modification of the surface, the pressure drop in the pipe increases as well. Therefore, the improvement in the performance of a heat exchanger has to take into consideration the ratio of increased heat transfer to the increase in pressure drop across the pipe.

Geometric modifications in a heat exchanger tube have been proven to provide a greater rate of heat transfer [1]. Geometric properties can be modified using fins, ribs, dimples or protrusions. Internal fins can be used on the inner tube walls to promote turbulence and increase forced convection heat transfer. The fins have a relatively low cost to machine, high reliability and produce a low increase in pressure drop. The weakness of finned tubes is that they increase the rate of fouling and are not as effective in the laminar and transitional flow region [5].

A twisted tape insert is made by manipulating a metallic strip into a helical shape and placing it into the tube. The tapes aim to induce transverse mixing through producing a swirling flow through the tube. The weaknesses of a twisted tape insert is that it is difficult to fix in 1 place and can be easily fouled [5]. Research by Deshmukh et al. [3] found that when using a delta wing vortex generator at Reynolds numbers less than 750, the heat transfer enhancement was significantly high. However, their data indicates the presence of large axial and circumferential vibrations which would need to be considered in low noise use cases such as submarines.

Banekar et al. [1] reported that dimples on the surface of a tube showed the greatest increase in heat transfer relative to the pressure drop penalty, when compared to other geometric modification techniques. Dingare and Rao [6] support this theory stating the thermal boundary layer growth is reduced due to the dimples, resulting in better mixing of the fluid inside the tube and hence greater heat transfer.

Vignesh et al. [8] used both Computational Fluid Dynamics (CFD) and experimentation to investigate whether the use of spherical dimples could enhance the rate of heat transfer when compared to a plain tube. The CFD analysis showed that the temperature difference and flow velocity is larger for the dimpled tube compared to the plain tube, however the pressure is increased. The experimental setup consisted of a double pipe heat exchanger, electric geyser to supply the hot water and a control system. The inlet temperature of the water was held constant whilst the flow rate was varied four times. The experimental results show that the overall heat transfer coefficient was between 56% to 64% larger then when compared to the plain tube. The effectiveness was also increased by 55%.

El-Said and Abou Alsood [4] experimented with injecting compressed air into a water filled heat exchanger to improve the heat transfer characteristics of the device. The experimentation used k-type thermocouples to measure the temperature difference, a vane anemometer to measure the flow rate of injected air and a digital flow meter to measure flow rate of water. The experiment showed an increase in heat transfer between 131% and 176% when compressed air was injected into the heat exchanger.

A flow experiences a pressure drop when pressure has been lost and it cannot be recovered. The fluid moving through a pipe is usually driven by a mechanical pump, which has a running cost associated with it. To minimise the pumping power required in a system, the pressure drop across the pipe should be reduced. The Darcy Weisbach equation provides a relationship for the friction factor of the pipe surface to the pressure lost throughout the pipe [2].

$$P_1 - P_2 = \frac{f\rho L V^2}{2D} \tag{1}$$

Where f is the friction factor, L is the length of the pipe, V is the bulk velocity of fluid in the pipe. D is the diameter of the pipe, P_1 and P_2 are the pressures at the inlet and outlet of the pipe respectively.

Approach

The following section details the methods used to calculate the desired results. These methods involve analytical calculations, CFD and experiments. These methods utilise the data in table 1.

Analytical calculations

Analytical calculations are used to validate the computational models. CFD results may vary from the actual values due to numerical errors and hence validation is essential for CFD. This section contains calculations for the pressure drop across a 1.92m long tube.

The pressure drop across a tube can be calculated using equation 1. Where f is the friction factor for the surface of the tube. For a laminar flow, the friction factor is given by [2]:

$$f = \frac{64}{Re} \tag{2}$$

For turbulent flow the friction factor is [2]:

$$f = (0.79 \ln Re - 1.64)^{-2} \tag{3}$$

The equation for calculating the Reynolds number of the flow in a tube is [2]:

$$Re = \frac{\rho V D}{\mu} \tag{4}$$

Where ρ is the density of the water and μ is the dynamic viscosity of water.

Using the above equations, The friction factor, Reynolds number and hence pressure drop across a 1.92 m tube with 11.08mm ID has been calculated and presented in table 2 and 3.

Table 1: Properties.

Property	Value	Units
Density, ρ	992.32	kg/m^3
Specific Heat of Water, Cp	4178	J/kg.K
Thermal Conductivity of sea water, k	0.63	W/m.K
Prandtl number, Pr	4.45	-
Dynamic Viscosity, μ	6.71E-04	Ns/m^2
Length, L	1.92	m
Diameter, D	0.01108	m

Table 2: Flow rates.

Property	Flow rate lt/min	Re_D
Case 1 (C1)	0.56	1573
Case 2 (C2)	3.15	8912
Case 3 (C3)	5.74	16252
Case 4 (C4)	8.33	23592

Computational fluid dynamics

CFD is a powerful tool to simulate flow characteristics in a variety of applications. Currently there are analytical equations to describe the heat transfer and pressure drop through a smooth tube, however as the dimpled tube is a unique geometry there are no such equations. Through numerical techniques the domain is broken down into small, finite volumes where the governing flow equations can be solved for each element. Therefore, the flow field and heat transfer data can be calculated through software. CFD provides a visual analysis of the flow around the dimples using velocity vectors and streamlines which can not be seen through experiments. This allows investigation into the dimple shape and depth versus flow characteristics which is of great interest. ANSYS CFX software has been used to model a single heat exchanger tube. The tube is 1.92 m in length with a 11.08 mm inner diameter. The initial conditions for the CFD analysis were based on the following assumptions for simplification of the problem;

1. Pipe inlet temperature is 313.15^{o}K.
2. The bulk temperature of the fluid external to the tube is constant along the entire length at 293.15^{o}K.
3. The inlet velocity profile is fully developed and corresponds to case 1 to 4 flow rates in table 2.
4. A convective heat transfer coefficient of 800 W/m^2K was set at the wall boundary from the outer fluid to the internal fluid.

The fluid used was water at 1 atmosphere without the buoyancy model considered. The Reynolds number for the upper flow rate is highly turbulent and therefore a turbulence model is required to accurately represent the flow conditions. Accuracy at the wall boundary was considered highly important to capture convection between the tube wall and the fluid. This was amplified for the dimpled tube as swirling was induced near the tube wall. The SST model was designed to provide more accurate predictions of flow separation close to the wall boundary. Hence, the SST model was selected for the simulations. To ensure convergence, a residual target of $1\text{x}10^{-5}$ was chosen for all equations in the heat transfer simulation, in addition, user points at the inlet and outlet monitored the temperatures.

The SST model requires a grid resolution of $y^+ < 2$. Therefore, $y^+ < 2$ can be used to determine the height of the first cell y_1. The first cell height for each flow rate can be calculated using equation 5, with $D = 0.01108m$. The first layer thickness (in m) for case 1-4 was; 2.05E-4, 4.10E-5, 2.34E-5 and 1.66E-5. The main tool used to refine the initial mesh was an inflation layer. The max elements size was set to 1 mm to limit the size of the quadrilaterals in the centre of the tube.

$$y_1 = D \times y^+ \sqrt{74} Re^{-13/14} \tag{5}$$

The dimpled tube designs were modelled using Autodesk Inventor. The dimples were made on the outer surface of the tube. The four different dimpled tube designs consisted of two different dimple depths, that were pitched every 60^{o} either singularly or as a double. The shallow and deep dimples were extruded down 0.75 mm and 1.16 mm respectively. The designs will be referred to as; double pitch deep depth (Design 1), double pitch shallow depth (Design 2), single pitch deep depth (Desgin 3), single pitch shallow depth (Design 4), smooth tube (Design 5), 2mm double depth (Design 6) and single deep double frequency (Design 7). See Figure 1 & 2. Designs 2 & 4 are similar but with shallower dimples.

Figure 1: CAD drawing of Design 1.

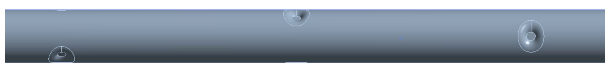


Figure 2: CAD drawing of Design 3.

Experiments

Heat transfer and pressure drop experiments of straight and dimpled tubes have been conducted over a wide operating range. The tubes were submerged in a water tunnel which provided a large, slow moving volume of water surrounding the tubes. The data collected included the inlet and outlet temperatures as well as the pressure difference of the water flowing through the tube. The temperature difference, ΔT , is used to determine the rate of heat transfer from the hot water to the large cold body of water. Pressure readings were obtained at the inlet and outlet of the tube by a manometer.

The apparatus consisted of an external water heater, water tunnel, retort stands, double ferrule tube connections, garden hose, variable area flow meter and a pressure pump. Errors in the experiment could occur from the inlet temperature varying between flow rates and tubes due to the tolerances in the water heaters control system and heat lost in the tubing before reaching the heat exchanger tube. Inaccuracies in the pressure loss measurements were due to the random errors from the lack of precision within the measuring equipment. The pressure manometer was divided into increments of 2 mm, hence giving an error of 2 mm as the manometer measured two locations simultaneously. This corresponded to a numerical error of $\pm$ 20 Pa, significantly effecting the case 1 flow rate results which were expected to be between 30 to 50 Pa.

Results

Analytical results

The water temperature difference ΔT, and pressure drop in the smooth tube was calculated using the equations presented in the section above. The results are represented in table 4.

From table 4 the key values of interest are the pressure drop across the tube subjected to an inlet temperature of 313.15^{o}K and each flow rate. These results are compared to the CFD outputs for validation.

Experimental results

The temperature difference, ΔT, for the dimpled tubes was normalised against the results from the smooth tube, providing a percent increase in heat transfer for each of the flow rates. The results using this method has been provided graphically in Figure 3.

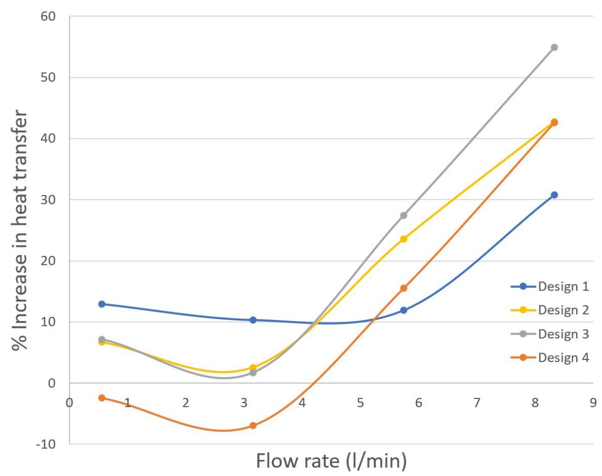


Figure 3: Experimental heat transfer

For the lowest flow rate, case 1, the dimpled designs 1, 2 and 3 experienced an increase in heat transfer of up to 13%, while design 4 decreased performance by 2.4%. This trend is continued at the case 2 flow rate, however with all tubes performing 3-6% worse. Case 3 flow rate saw improvements between 12% and 24%. All designs saw large improvements ranging from 31% to 55% at the highest flow rate, case 4. Note that design 1 performed roughly the same between flow rates 1-3, while all other dimpled designs experienced a large heat transfer increase at turbulent flow rates.

The pressure loss at the lowest flow rate varied significantly between the tubes. This is believed to be due to precision errors in the measuring equipment. Case 2 flow rates saw the pressure loss increase ranging between 25% and 55%. The pressure loss reduced and appeared to stabilise between case 3 and 4 at 10% to 40%.

Validation results

CFD simulations were run with the corresponding inlet and water tunnel temperatures for validation of the model. Table 3 provides results for CFD and experiments heat dissipation. Results showed the CFD output to be consistently lower than the experimental data for both the smooth and dimpled tube. Smooth tube errors of 20%, 15.4%, 18% and 7.4% and design 1 errors of 24.6%, 15.5%, 26.1% and 7.5% were calculated for case 1-4 flow rates, respectively.

Table 3: Heat transfer validation

Flow rate (l/min)	Smooth tube Experiment (Watts)	Smooth tube CFD (Watts)	Design 1 Experiment (Watts)	Design 1 CFD (Watts)
0.56	395	316	446	336
3.15	1251	1058	1380	1165
5.74	998	819	1117	875
8.33	937	867	1226	1133

During the experiment, water was circulated over the tube at 0.095m/s, meaning a small amount of convection occurred on the outer surface of the tube. The CFD model assumed the outer fluid to be still, therefore not calculating the heat transfer due to convection on the outside of the tube. This is believed to be why the experimental data is consistently higher than the CFD data.

Table 4 shows analytically calculated pressure drop for the smooth tube using equation 1. Compared to the CFD results for pressure drop, errors of 0.64%, 0.87%, 0.76% and 1.63% for the case 1-4 flow rates respectively are observed. These errors are quiet small and provide confidence in the modelling of the flow characteristics. The theoretical results also suggest that there was an error in the experimental data for pressure drop.

Table 4: Smooth tube CFD and analytical results

l/min	Theory (Pa)	CFD (Pa)
0.56	32.26	32.47
3.15	827.97	820.76
5.74	2336.20	2318.55
8.33	4474.43	4401.5

Computational fluid dynamics results

Table 5 contains the results from the CFD simulations when the initial and boundary conditions are kept constant for all designs. The inlet temperature was 40 ^{o}C with a bulk outer fluid temperature of 20 ^{o}C and a heat convection coefficient of 800 W/m^2K. This h value was chosen as it was approximately the median value obtained from the experimental data. Keeping the conditions the same for all runs allows for an accurate comparison of the designs with respect to one another.

Design 3 (single deep dimple) provided the largest ΔT across the tube at case 1 flow rate with a 6.4% increase over the smooth tube. This increase comes at a cost of a 43.7% increase in pressure drop across the tube. For the case 2 and 3 flow rates, Design 1 (Deep double dimple) possessed the greatest heat transfer improvements with a 0.66% and 0.9% increase for the respected flow rates.

Table 5: CFD results

Tube	1	2	3	4	5	6	7
Case 1							
ΔP (Pa)	46	40	47	41	32	64	51
ΔT (K)	9.6	9.5	10.0	9.4	9.4	10.4	10.2
Case 2							
ΔP (Pa)	1015	830	866	776	821	1450	1037
ΔT (K)	3.83	3.78	3.8	3.79	3.8	3.84	3.81
Case 3							
ΔP (Pa)	3059	2503	2620	2339	2319	4047	3179
ΔT (K)	2.34	2.33	2.33	2.33	2.32	2.35	2.34
Case 4							
ΔP (Pa)	6052	4895	5124	4535	4402	8927	6353
ΔT (K)	1.690	1.682	1.686	1.683	1.724	1.695	1.685

The results for the case 4 flow rate all showed a small decrease in the heat transfer, proposing the dimples are more effective in the laminar flow regions. Due to the flow being turbulent, mixing occurs more readily meaning the dimples do not provide any increase in mixing.

The two shallow dimples designs performed poorly compared to the smooth tube and in some cases providing no heat transfer improvement. This led to the conclusion that a deeper dimple provides a greater increase to the heat transfer. To test this theory, a new design was created based on Design 3 but with dimples 2mm deep called design 6. The results are shown in table 5 and show an increase of 10.23% for the case 1 flow rate with slight improvements over any other dimpled design.

The deeper dimple provided a larger increase to heat transfer but it came at a significant pressure drop increase of 74.5-102.8%. To determine the optimal location of the dimple, Design 3 was modified with a smaller distance between the dimples along the length of the tube, effectively doubling the amount of dimples. This design is called Design 7 and the results are presented in table 5. Design 7 showed an increase in heat transfer of 7.4% for the case 1 flow rate, slightly better than design 3. The benefit of design 7 is it has a 50% pressure drop increase which is significantly less than the 2mm deep dimples. This concludes that dimple depth and frequency has a direct effect on heat transfer through out the tube.

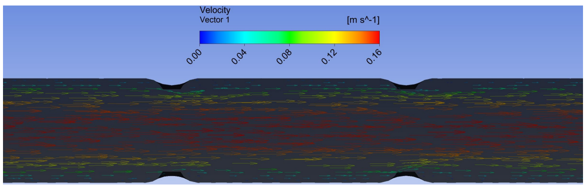
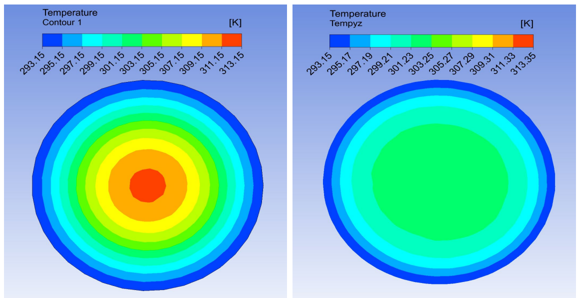


Figure 4: Design 1 velocity vectors around the dimples

It is believed that the dimples disrupt the flow and force mixing of the fluid. The cooled fluid at the wall surface is forced to mix with the not-cooled centre fluid in the tube. This increases the temperature at the wall surface and hence increases the temperature difference between the fluid and the wall. Figure 5 shows that the outlet flow is better mixed for Design 1 when compared to Design 5 as the temperature distribution is more uniform. Figure 4 supports this as the velocity vectors show the flow over the dimples being forced towards the centre of the tube.



(a) Design 5 (b) Design 1

Figure 5: Outlet temperature contour plot

Conclusions

The CFD analysis showed a trend that dimples in the surface of a tube increase heat transfer. The greatest improvements seen in the CFD comparison were with Design 6 in the laminar flow region seeing a maximum of 10.23% increase in heat transfer. It was also found that the dimple depth corresponds to the increase in heat transfer but also an increase in pressure drop. The largest pressure drop increase recorded was 102.8% at laminar conditions for design 6. The CFD results were compared to analytical calculations with a difference between the results of approximately 1%. The experimental data showed approximately a 20% difference to the CFD results for the heat transfer simulations.

References

[1] Banekar, Y.D. Bhegade, S.R. and Sandbhor, M.V. Dimpled Tube Heat Exchanger, International Journal of Science, Engineering and Technology Research, 4, 1632-1635, 2015.

[2] Bergman, T. Lavine, A. Incropera, F. and Dewitt, D., Fundamentals of Mass and Heat Transfer, John Wiley & Sons Inc., 978-0470-50197-9, 2011.

[3] Deshmukh, P. Prabhu, S. and Vedula, R., Heat transfer enhancement for laminar flow in tubes using curved delt wing vortex generator inserts, Applied Thermal Engineering, 106, 1415-1426, 2016.

[4] El-Said, E. and Abou Alsood, M., Experimental investigation of air injection effect on the performance of horizontal shell and multi-tube heat exchanger with baffles, Applied Thermal Engineering, 134, 238-247, 2018.

[5] Ji, W. Jacobi, A. He, Y. and Tao, W., Summary and evaluation on single-phase heat transfer enhancement techniques of liquid laminar and turbulent pipe flow, International Journal of Heat and Mass Transfer, 88, 735-754, 2015.

[6] Rao, A. and Dingare, S.V.,A Review on Dimpled Tube Heat Exchanger,International Journal of Current Engineering and Technology, 7, 236-239, 2017.

[7] Thulukkanam, K. Heat Exchanger Design Handbook, Taylor Francis Group., 978-1-4398-4213-3, 2013.

[8] Vignesh, S. Moorthy, V. and Nallakumarasamy, Experimental and CFD Analysis of Concentric Dimple Tube Heat Exchanger, International Journal of Emerging Technologies in Engineering Research, 5, 18-26, 2017.