

Thermal performance of periodic serpentine channels with semi-circular and triangular cross-sections

JH Fourie

***Mini-dissertation submitted in partial fulfilment of the requirements for the degree
Master of Engineering in Nuclear at the Potchefstroom Campus of the North-West
University***

Supervisor: Dr. Jan-Hendrik Kruger

May 2011



**NORTH-WEST UNIVERSITY
UNIBESITHI YA BOKONE-BOPHIRIMA
NOORDWES-UNIVERSITEIT
POTCHEFSTROOMKAMPUS**

Acknowledgements

I would like to thank all the employees at Aerotherm for their support regarding help with STAR-CCM+. I would especially like to thank Cristiaan de Wet for sacrificing much of his time to help me with the simulations. Further I would like to thank Martin for the opportunity to make use of your company's facilities at Aerotherm.

I want to thank Prof. Jat du Toit from the School of Mechanical Engineering, Potchefstroom Campus. Professor's help with regards to the Nusselt number methodology was extremely useful.

I would like to thank Johann Venter with whom I worked closely during the project. Without you knowing, you really motivated me in the times that I struggled to continue. We had some tough problems to overcome during our projects, but it was made easier by working with you.

Finally, I want to thank Dr Jan-Hendrik Kruger from the Post-graduate School of Nuclear Science and Engineering, Potchefstroom Campus, for being my study leader during the full project duration. I took up a lot of your time but you always helped me with the persistent problems.

Summary

Printed Circuit Heat Exchangers (PCHE's) are very efficient compact exchangers considered as recuperators in the Brayton cycle of new generation high temperature reactor designs.

The heat exchangers use multiple plate layers containing tortuous micro-channels to enhance the heat transfer between the working fluids. Understanding the flow phenomena and its effects on heat transfer inside the channels would ultimately lead to improved exchanger designs.

A numerical study was conducted on the heat transfer behaviour of fully developed laminar flow in three-dimensional serpentine channels with semi-circular and equilateral triangular cross-sections. Computational Fluid Dynamics (CFD) was used to investigate the heat transfer characteristics of various physical designs under different flow conditions.

The design parameters for serpentine channels were: the bend's radius of curvature (R_c), the channel diameter (d), the half-wavelength of the channel (L) and Reynolds number (Re). Results were obtained for a range of configurations ($1.0 \leq \frac{R_c}{d} \leq 1.8$, $\frac{L}{d} = 4.5$) and flow conditions ($50 \leq Re \leq 400$). Water was used as fluid and the uniform surface temperature boundary condition was applied to channel walls.

Results showed a definite increase in heat transfer when Re was increased. Heat transfer enhancement due to fluid mixing was a maximum at the highest Reynolds number since the inertial effects dominated the viscous flow effects. However, a slight decrease in enhancement was noted as the bend radius of curvature increased, due to weaker mixing of the flow.

In several test cases, secondary flow effects separate from main vortices and mixing were observed to influence the heat transfer. Future research topics are suggested to further clarify these phenomena.

Table of contents

Acknowledgements	I
Summary	II
List of Figures	VI
List of Tables	IX
List of mathematical operators	IX
List of symbols	X
List of Greek symbols	XII
List of acronyms	XII
List of definitions	XIII
Chapter 1 – Introduction	1
1.1 What are Printed Circuit Heat Exchangers (PCHE's)?	1
1.2 Channel design	2
1.3 Advantages	2
1.4 Nuclear engineering design	3
1.5 Problem statement	4
1.6 Objectives	5
1.7 Chapter overview	6
Chapter 2 – Literature study	7
2.1 Nusselt number (Nu)	7
2.2 Flow characterisation	13

2.3 Geometric parameters.....	18
2.4 Reynolds number	20
2.5 Prandtl number	20
2.6 Chapter summary.....	21
 Chapter 3 – Simulations	 23
3.1 Method of investigation	23
3.1.1 Nusselt number methodology	23
3.1.2 Computational fluid dynamics methodology	25
3.1.3 Meshing	34
3.1.4 Physics	37
3.2 Scope	39
3.2.1 Assumptions	39
3.2.2 Boundary conditions	39
3.2.3 Straight channel	40
3.2.4 Serpentine channel.....	41
3.3 Chapter summary.....	45
 Chapter 4 – Results and Discussion.....	 47
4.1 Straight channels	47
4.1.1 Numerical mesh	47
4.1.2 Circular straight channel	49
4.1.3 Semi-circular straight channel	51
4.1.4 Triangular straight channel.....	51
4.1.5 Conclusion	51
4.2 Serpentine channel.....	52

4.2.1 Numerical mesh	52
4.2.2 Reynolds number.....	54
4.2.3 Radius ratio, R_{cd}	58
4.2.4 Conclusion	61
4.3 Flow characteristics	61
4.4 Chapter summary.....	70
 Chapter 5 – Conclusions	 71
 Chapter 6 - Recommendations.....	 73
 References	 75
 Appendix A	 77
A.1 Results for straight channels	77
A.1.1 Circular straight channel.....	77
A.1.2 Semi-circular straight channel.....	78
A.1.3 Equilateral triangular straight channel.....	79

List of Figures

Chapter 1

Figure 1.1 - Plates having different flow schemes stacked to form a block	1
Figure 1.2 - Schematic showing a multi-pass counter flow configuration	2
Figure 1.3 - Maximum design pressure versus temperature for different materials during a preliminary material selection for a HTGR recuperator	3

Chapter 2

Figure 2.1 - Tabulated values of the Nusselt number for some cross-section shapes.....	8
Figure 2.2 - Three units of a serpentine path shape.	10
Figure 2.3 - Schematic showing the heat transfer enhancement factor versus Reynolds number values for a serpentine channel having different cross-section shapes.....	11
Figure 2.4 - Schematic showing the serpentine path shape with semi-circular cross-section	12
Figure 2.5 - Mean heat transfer enhancement factor as a function of Reynolds number.....	13
Figure 2.6 - Heat transfer enhancement as function of the proportional distance within a single serpentine channel.....	16
Figure 2.7 - Schematic showing the velocity contours downstream of the bends in a serpentine unit.....	16
Figure 2.8 - Schematic showing heat transfer enhancement factor versus the radius ratio in a semi-circular serpentine channel.	18
Figure 2.9 - Schematic showing heat transfer enhancement versus the half wavelength to channel diameter ratio	19
Figure 2.10 - Schematic showing the heat transfer enhancement factor versus Reynolds number values for a semi-circular cross-section	20
Figure 2.11 - Schematic showing the variation of the heat transfer enhancement factor versus Prandtl number values.	21

Chapter 3

Figure 3.1 - Schematic showing the variation of the peripherally averaged Nusselt number versus the dimensionless axial distance	24
Figure 3.2 - Schematic shows two examples of different mesh types.	25
Figure 3.3 - Schematic shows mass flow through a cubical control volume	26
Figure 3.4 - Schematic shows a illustration of cell P and its adjacent neighbouring cells	31

Figure 3.5 - Schematic showing differences within the prism layer section for meshes having stretching factors of 1.3 and 1	35
Figure 3.6 - Snapshot of the distortion occurring when the prism layer thickness is set too high for the corresponding cell target size.	36
Figure 3.7 - Visual representation of the plane locations at which the Nusselt numbers were calculated for the Reynolds number investigations.	43
Figure 3.8 - Schematic shows a channel where $Rc/d = 2$	44
Figure 3.9 - Visual interpretation of the plane locations at which the Nusselt numbers were calculated for the radius ratio investigations.	45

Chapter 4

Figure 4.1 - Schematic shows the velocity boundary layers for internal flow.	48
Figure 4.2 - Schematic shows the temperature boundary layers for internal flow.....	48
Figure 4.3 - Schematic illustrating the obtained Nusselt number versus the dimensionless axial distance on a logarithmic scale.	50
Figure 4.4 - Schematic showing the convergence of the Nusselt number versus the distance from the inlet on a logarithmic scale.	50
Figure 4.5 - Graph of heat transfer enhancement factor versus Reynolds number for semi-circular and triangular serpentine channels.	55
Figure 4.6 - Schematic showing the enhancement factor for the proportional distances in the semi-circular serpentine channel for different values of the Reynolds number.....	56
Figure 4.7 - Schematic showing the enhancement factor versus the proportional distances in the triangular serpentine channel for different values of Reynolds number.	56
Figure 4.8 - Cross-sectional velocity profiles at 20 plane locations inside a semi-circular serpentine channel, indicating high velocity regions within the flow domain.....	57
Figure 4.9 - Cross-sectional velocity profiles at 20 plane locations inside a triangular serpentine channel, indicating high velocity regions within the flow domain.....	57
Figure 4.10 - Graph of heat transfer enhancement factor versus radius ratio for semi-circular and triangular serpentine channels.....	58
Figure 4.11 – The enhancement Factor versus the proportional distance for a semi-circular serpentine channel.	60
Figure 4.12 – The enhancement Factor versus the proportional distance for equilateral triangular serpentine channel.	60

Figure 4.13 - Velocity contours from the Rosaguti et al. (2006) investigation at locations of 1 diameter length downstream of the bends ($Re = 200$).	62
Figure 4.14 - Velocity contours from the current investigation at the location of 1 diameter length downstream of the bends ($Re = 200$).	63
Figure 4.15 - Velocity contours from the current investigation at the location of 1 diameter length downstream of the bends. ($Re = 400$).	63
Figure 4.16 - Schematic shows the streamlines for the semi-circular serpentine channel with flow of $Re = 400$. The schematic shows the post bend-1 region.	64
Figure 4.17 - Schematic shows the streamlines for the semi-circular serpentine channel with flow of $Re = 400$. The schematic shows the post bend-2 region.	64
Figure 4.18 - Schematic shows the streamlines for the semi-circular serpentine channel with flow of $Re = 400$. The schematic shows the post bend-3 region.	65
Figure 4.19 - Schematic shows the streamlines for the semi-circular serpentine channel with flow of $Re = 400$. The schematic shows the post bend-4 region.	65
Figure 4.20 - Schematic shows the streamlines as velocity vectors, indicating the flow direction	66
Figure 4.21 - Top view of a semi-circular serpentine channel. The streamlines show radial movement in the post bend regions.	67
Figure 4.22 - Schematic shows a vortex adjacent to the roof of bend-4 in a serpentine channel with semi-circular cross-section.	68
Figure 4.23 - Top view of a triangular serpentine channel. The streamlines show rotational movement in the post bend regions.	69
Figure 4.24 - Schematic shows a small vortex adjacent to the roof of bend-4 in a serpentine channel with triangular cross-section.	69

Chapter 6

Figure 6.1 - Results for surface extruder mesher vs. polyhedral cell mesher.	73
---	----

List of Tables

Chapter 3

Table 3.1 - The mass inflow, outflow and the net mass flow rate over a control volume.	27
Table 3.2 - Details regarding the straight channel investigations.	40
Table 3.3 - Flow velocities and Reynolds numbers for a semi-circular serpentine channel. ...	42
Table 3.4 - Flow velocities and Reynolds numbers for a triangular serpentine channel.	42

Chapter 4

Table 4.1 - Sets of meshing parameters for grid independence of serpentine simulations.	52
Table 4.2 - Summary of the results obtained from investigating the Reynolds number.	54
Table 4.3 - Summary of the results obtained from investigating the radius ratio.	58

Appendix A

Table A.1 - Results from the straight channel with circular cross-section.	77
Table A.2 - Results from the straight channel with semi-circular cross-section.	78
Table A.3 - Results from the straight channel with triangular cross-section.	79

List of mathematical operators

$\frac{D}{Dt}$	Material derivative with respect to time
$\frac{\partial}{\partial t}$	Partial derivative with respect to time
$\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}$	Partial derivative with respect to the Cartesian coordinate system's dimensions
$\int_{\Omega} d\Omega$	Integral over area Ω
div	Divergence of a vector field
$grad$	Gradient of a scalar field
$\sum_{f=1}^n$	Summation of property values over n faces
$\vec{\nabla}$	Del, geometric specific gradient operator
$\Rightarrow$	Material implication

List of symbols

A	Heat transfer surface area
$\vec{A}_f$	Outward pointing face area vector
a, b	Geometric properties for hydraulic diameter calculations (refer to figure 2.1)
a	Coefficient describing convective and diffusive effects
c_p	Specific heat capacity at constant pressure
c_v	Specific heat capacity at constant volume
Dn	Dean's number
d	Channel diameter
d_h	Hydraulic diameter
e_{Nu}	Heat transfer enhancement factor
F	Mass flux
G^*	Guessed Rhie and Chow correction term
$H1$	Uniform surface heat flux heat transfer boundary condition
$H2$	Uniform surface heat flux heat transfer boundary condition with wall conductivity
h	Convection heat transfer coefficient
i	Internal energy
k_f	Fluid thermal conductivity
L	Half wavelength
$\frac{L}{d}$	Half wavelength to channel diameter ratio
Nu	Nusselt number
Nu_m	Mean Nusselt number (refer to figure 3.1)
$Nu_{path\ shape}$	Nusselt number for a specific channel type
$Nu_{straight}$	Nusselt number for a straight channel
Nu_T	Nusselt number for a straight channel for the T heat transfer boundary condition (refer to figure 2.1)
Nu_x	Peripherally averaged local Nusselt number
$nb\ (subscript)$	Refers to neighbouring cell faces
N, S, W, E	Neighbouring nodes of node P. (North, South, West and East)
n, s, w, e	Neighbouring faces of node P. (north, south, west and east)
P	Pressure

P (subscript)	Refers to node P
P'	Difference between the final pressure P and the guessed pressure P^*
P^*	Pressure guess value
Pr	Prandtl number
Q	Heat flow
q	Volumetric source rate
$\bar{q}''_w$	Peripherally averaged local wall heat flux
R	Specific gas constant
R_c	Bend radius of curvature
$\frac{R_c}{d}$	Radius ratio
Re	Reynolds number
S_E	Energy source term (including effects from potential energy changes)
S_M	Momentum source term (including the effects from surface stresses)
S_U	Momentum source term for the U cell
S_ϕ	Source term of property ϕ
S_{ϕ_P}	Volumetric source term of property ϕ
s_M	Viscous stress terms
T	Uniform surface temperature heat transfer boundary condition
T_b	Bulk fluid temperature
T_f	Fluid temperature of a selected cell
T_s	Substance temperature
T_w	Wall surface temperature
u, v, w	Velocity components in Cartesian coordinate system
$\vec{u}_f$	Velocity component in direction of the face area vector
u^*	Velocity guess value
$\vec{u}, \vec{v}, \vec{w}$	Velocity vectors in Cartesian coordinate system
V	Volume of cell
V_P	Volume of cell P
v	Flow velocity
v_{ave}	Cross-section average velocity at local axial position

$\vec{V}_f$	Face velocity vector
V_m	Mean velocity of a serpentine channel
w	Axial flow velocity
x	Axial length from the point of heating
x, y, z	Control volume side lengths in Cartesian coordinate system
x^*	Dimensionless axial distance

List of Greek symbols

Γ	Diffusion coefficient
Γ_f	Diffusion coefficient in the direction of the face area vector
ΔT	Difference in temperature between a solid surface and the surrounding fluid
$\Delta x, \Delta y, \Delta z$	Change in the x, y, z directions respectively
$\Delta \rho$	Change in density
δ_x	Distance between two relevant CV nodes or faces
λ	Secondary viscosity (relates stresses to the volumetric deformation)
μ	Dynamic viscosity (relates stresses to linear deformation)
ρ	Fluid density
ρ_f	Fluid density at face
ϕ_f	Scalar property (bulk fluid temperature, boundary heat flux)
ϕ	Property to be calculated
Ψ	Internal energy source due to viscosity effects
Ω	Integral area

List of acronyms

1D	One-Dimensional
2D	Two-dimensional
CFD	Computational Fluid Dynamics
CV	Control Volume
FVM	Finite Volume Method
HTGR	High Temperature Gas-cooled Reactor
HX	Heat Exchanger

IHX	Intermediate Heat Exchanger
PCHE	Printed Circuit Heat Exchanger
PL	Prism Layer
SIMPLE	Semi-Implicit Method for Pressure-Linked Equations

List of definitions

Boundary layer	The velocity and temperature layers immediately adjacent to the surface of a solid material, past which the bulk fluid (liquid or gas) is flowing.
Brayton Cycle	A thermodynamic cycle used for power generation, consisting of two constant-pressure (isobaric) processes interspersed with two constant-entropy (isentropic) processes with gas as working fluid.
Compact heat exchanger	A wide definition of what characterized by surface area densities of $200 \frac{\text{m}^2}{\text{m}^3}$ and more, and also hydraulic diameters of 14 mm or lower.
Compressible fluid	A fluid in which a density variation may occur due to a change in the fluid's pressure. The characteristics of the fluid may change as the density varies. Gasses are generally considered as compressible and liquids as incompressible fluids.
Conductive heat transfer	A diffusive process within matter where thermal energy flows from a higher temperature region to a lower temperature region.
Containment building	The structure enclosing a nuclear reactor, preventing leakage of radiation into the surroundings under normal- and emergency conditions. The structure is made of steel or reinforced concrete and its maximum design pressure is in the range of 410 to 1400 kPa.
Convection	Transport mechanism due to molecule movement in a fluid.

Convective heat transfer	A transfer process where thermal energy is transferred between a solid material and a fluid and more heat is added or removed because of fluid movement than with conduction alone.
Counter flow	Fluids flow in opposite directions in adjacent parts of a heat exchanger.
Cross flow	Fluid flow paths cross each other in adjacent parts of a heat exchanger.
Diffusion	A passive transport mechanism involving the migration of molecules from a highly concentrated region to a less concentrated region.
Diffusion bonding	A solid-state process for joining metals by using only heat and pressure to achieve atomic bonding through molecular movement.
Flow separation	When the fluid's boundary layers no longer follows the contour of the solid material. This is because the residual momentum of the fluid may be inadequate for the flow to proceed into regions of increasing pressure. The residual momentum is the momentum left after overcoming the viscous forces.
Fully developed	The velocity or temperature boundary layers do not change anymore, when following the direction of flow.
Grain growth	The enlargement of individual grains within a metal or alloy occurs when the metal or alloy is exposed to a temperature which is above the recrystallization temperature for that material.

Header	A component connected to the ends of a heat exchanger unit, which channels fluid into or out of the internal tubes of the unit.
Heat load	The amount of heating or cooling needed to maintain desired temperatures in the fluids passing through a HX.
Inertial forces	An apparent force affecting bodies with mass and movement. In rotational systems, the Coriolis- and centrifugal forces are both inertial forces at work on the fluid.
Laminar	A fluid flow regime in which the fluid travels smoothly and in regular paths. The velocity, pressure, and other flow properties at each global location in the fluid domain remain constant.
Newtonian fluid	A fluid in which the strain of the element is proportional to the stresses exerted on a fluid element.
Parallel flow	Fluids flow in the same direction in adjacent parts of a heat exchanger.
Photo-chemical etching	Fabrication process for machining away selected areas on sheet metal by using photoresist and etchants.
Pressure drop	A loss in fluid pressure between the inlet and the outlet of a HX unit due to frictional losses incurred during internal pipe flow.
Primary flow	When fluid flow is visible as simple flow patterns behaving like an inviscid fluid, which has no viscosity.

Recuperator	The term recuperator refers to a heat exchanger used for heat recovery in a thermodynamic cycle.
Secondary flow	A relatively minor flow superimposed on the primary flow through the action of friction usually in the vicinity of boundaries. The flow speed and direction in secondary flow differs from that of the primary flow.
Turbulent	A fluid flow regime in which the fluid undergoes irregular fluctuations, or mixing. The movement of the fluid at any point is continuously undergoing microscopic changes in magnitude and direction, which results in swirling and eddying as the bulk of the fluid moves in a particular direction.
Unsteady flow	Non-steady (unsteady or transient) flow changes the bulk velocity and/or direction with time. Also refers to fluid flow in which material properties of the flow change with respect to time. Some characteristics of turbulent flow may be present.
Unstructured mesh	A computational domain consisting of connected elements not organized in an orderly manner with various topologies and irregular size/volume.
Viscous dissipation	The transformation of the mechanical energy to internal energy in a moving fluid.
Viscous forces	Related to viscosity and responsible for a fluid's resistance to be deformed by shear forces during flow.

Chapter 1 – Introduction

1.1 What are Printed Circuit Heat Exchangers (PCHE's)?

Printed Circuit Heat Exchangers (PCHE's) are compact type heat exchangers developed and produced for industrial applications by companies such as Heatric Ltd (Heatric, 2010). The name originates from the process by which the plates are manufactured, whereby micro- or milli-sized flow channels are photo-chemically etched into the plates on one side. This process is in essence the same used to manufacture electronic printed circuits and from there the name 'Printed Circuit Heat Exchangers' according to Hesselgreaves (2001).

The etched plates are welded onto each other using a process called diffusion bonding, which involves pressing assemblies of loose plates against each other under high temperature and pressure. This promotes grain growth over the plate interfaces to achieve the same strength and thermal conductance as the base material.

A block is formed by bonding a number of plates together. The individual blocks are welded together, forming the core of the heat exchanger. Finally, the headers are connected to the core, to complete the unit (Heatric, 2010). Figure 1.1 shows a number of plates stacked together with the primary and secondary flow paths.

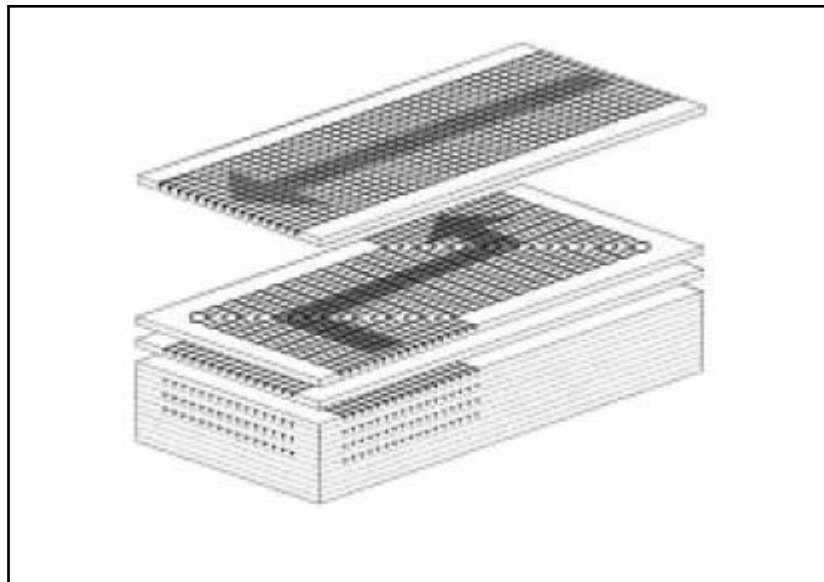


Figure 1.1 – Schematic shows plates having different flow schemes, stacked to form a block. Modified from Heatric (2010).

1.2 Channel design

The plates may be etched to produce a wide variety of flow stream configurations. An individual plate may either contain a single fluid, or different fluids flowing in separate channels on the same plate. The flow stream configurations may include counter flow, parallel flow or cross flow. Multi-pass counter/cross flow is also possible (Heatric, 2010). Figure 1.2 shows a multi-pass counter flow configuration.

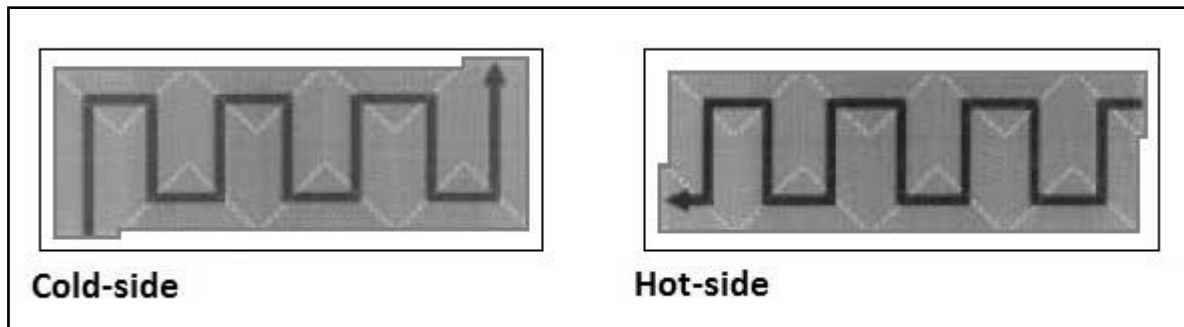


Figure 1.2 - Schematic showing a multi-pass counter flow configuration. Modified from Heatric (2010).

The micro-channels typically have hydraulic diameter (d_h) values which range between 1.5 and 3 mm (Hesselgreaves, 2001). The channels may be straight or zigzag, depending on various factors such as the type of fluid used (gas or liquid); the required heat load or specified pressure drop and channel length.

The solid material between the channels is called the “land”, and its typical width is approximately equal to 0.5 mm, depending on the design pressure of the fluid flowing inside the channels. The number of blocks welded together to form a PCHE unit is determined by the designed heat load. Some of the biggest units have a heat transfer surface area of more than 2500 m².

1.3 Advantages

As previously mentioned the diffusion bonding process promotes grain growth over the plate interfaces, resulting in little loss of material strength and the exchanger unit being capable of handling extreme operating conditions. The PCHE is suitable for use in e.g. high-temperature, corrosive and radioactive environments typical of the nuclear and chemical industries.

Flow channels are manufactured following a desired pattern to promote high heat transfer characteristics, without incurring significant pressure losses. Efficiencies up to 98% are possible and large surface area densities in order of 1300 m²/m³ are typical.

Printed circuit heat exchangers have the advantage of being up to 85% more compact in overall physical size than conventional heat exchangers for the same thermal load (Heatric, 2010).

1.4 Nuclear engineering design

PCHE technology is being considered for the role of intermediate heat exchanger (IHx) in new High Temperature Gas-cooled Reactor (HTGR) designs. This type of reactor can use a Brayton Cycle to drive a gas turbine and in turn requires a high efficiency recuperator. The recuperator will need to operate in a radioactive environment under high temperature and pressure for a design lifetime of 60 years (Pra et al. 2008). The compactness of the PCHE makes it even more attractive to the nuclear industry since all components that are exposed to radioactive fluids must be installed inside the main containment building.

No design code exists for approved nuclear materials and Heatric (2010) has done research on materials that may be used to build the recuperator. Figure 1.3 shows the materials considered to be most suitable for the recuperator and plots strength (deduced from maximum design pressure) versus temperature. The best suited material, based on strength and corrosion resistance is Alloy-617 which is a nickel-chromium-cobalt-molybdenum alloy, also known as Inconel-617. This specific material is highly corrosion resistant at high temperatures, but still allows for normal manufacturing methods.

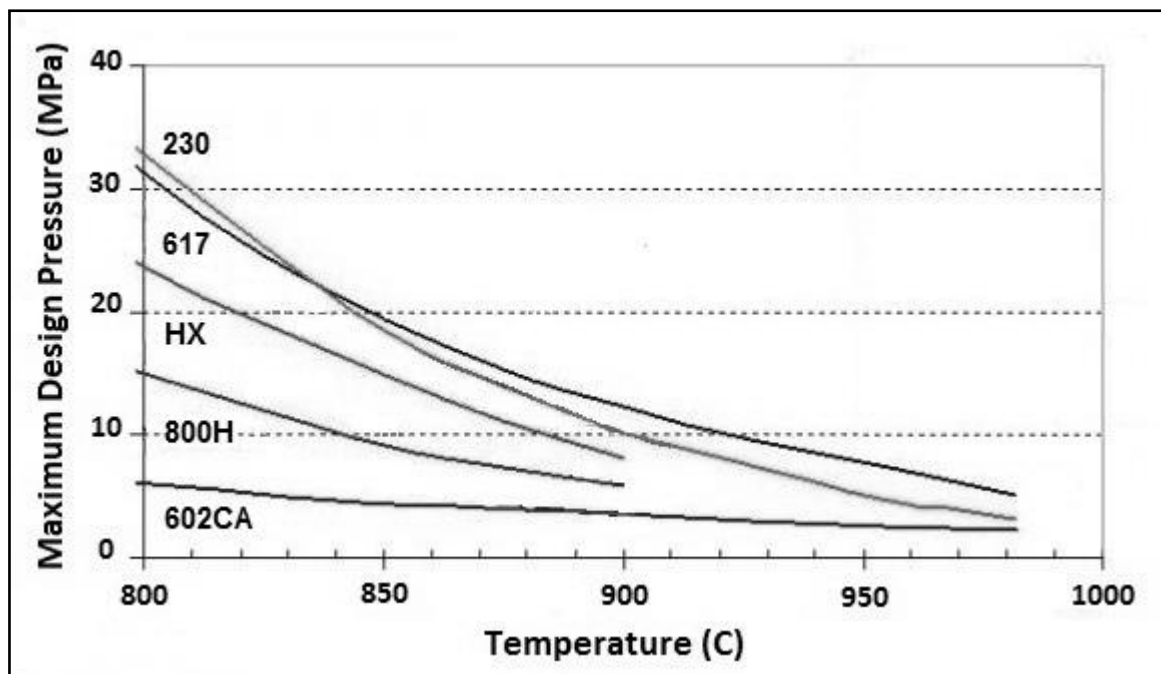


Figure 1.3 - Maximum design pressure versus temperature for different materials during a preliminary material selection for a HTGR recuperator. Modified from Heatric (2010).

1.5 Problem statement

Cost estimation for heat exchangers is done early during the thermal/hydraulic design process since cost is usually the most important parameter considered during the design process (Southall et al. 2008).

Saving weight is an important design criterion since weight determines the capital cost of a heat exchanger. To save weight and cost, it is important to increase the compactness of the unit. This can be done by increasing the effective heat transfer area or increasing the overall heat transfer performance.

Enhancements in a PCHE unit's heat transfer performance can reduce the size of the unit while retaining the capability of handling the same thermal load.

An investigation into the heat transfer behaviour of micro-channel flow could provide information on improving the performance of compact heat exchangers. The PCHE's plate layout has a clear influence on the heat transfer behaviour and finding the best geometric configuration is essential in promoting higher surface area densities.

Finding the best configuration for heat transfer proves to be a valid focus area to be used in the future design of compact heat exchangers.

The technology is relatively modern and the phenomena of flow in micro-scale channels have only recently been investigated in detail. The motivation behind previous investigations has purely been driven by the need to find the channel configuration which promotes maximum heat transfer while limiting the pressure drop through the unit (Rosaguti et al. 2005).

The heat transfer and the pressure drop are strongly dependent on the shape and layout of the micro channels. Having wavy channels enhances heat transfer compared with straight channels but without incurring a significant pressure loss.

The layout of the wavy channel has a strong influence on the heat transfer behaviour. Studies have been done on a number of channel layouts with different cross-section shapes to quantify these influences, but research is still ongoing.

Numerical investigations of the detail flow patterns inside specific configurations and cross-sections may explain the heat transfer behaviour and subsequently ways can be derived to enhance thermal performance of printed circuit heat exchangers.

1.6 Objectives

The serpentine channel shape was chosen as it is a promising channel path shape and has been proven to give significant enhancement for flow conditions used in this study. The semi-circular and equilateral triangle cross-sections were investigated in parallel with the serpentine channels.

CFD has a very successful reputation for accurately investigating geometries and flow conditions similar to the current study; therefore CFD was used to investigate the flow in the channels.

Firstly, the accuracy of the results from the CFD application must be verified by simulating straight channels for circular, semi-circular and triangular cross-section shapes and comparing the results with published theoretical values.

Next the simulations on the serpentine channel must be done. The influencing factors of micro-channel flow were identified and it was essential for the current study to investigate flow phenomena for a pre-defined range of geometries and flow conditions, being the main variables of PCHE design.

Therefore the following objectives were defined to successfully investigate the relevant factors of micro-channel flow.

- To establish a relationship between the flow and heat transfer for different geometric configurations and flow conditions.
- Investigate flow phenomena for serpentine channel flow and identify the flow structures with the most dominant influence on heat transfer.
- Identify local variations in heat transfer within a serpentine unit and explain their presence.
- Find a geometric configuration and the flow conditions that have the best heat transfer behaviour and benefits most from using tortuous channels compared to straight channels.

1.7 Chapter overview

The investigation was based on the numerical evaluation of a serpentine channel with semi-circular and equilateral triangular cross-sections for different flow conditions.

The literature study (Chapter 2) describes the flow phenomena and heat transfer behaviour unique to serpentine channels as was found in published literature. Furthermore an idea is given of what was expected prior to modeling, what aspects required consideration and what problems might have been encountered during the investigation.

Simulations (Chapter 3) follows and it describes the heat transfer calculation methodology that was used for all simulations and the scope of investigation that was defined for the flow conditions and geometric variations. Details are given about the numerical mesh and physics conditions that was applied to the simulation models.

In Chapter 4 the results from the numerical investigation (described in Chapter 3) are presented and discussed. The ultimate configuration which performed best for the conditions under investigation is presented and discussed.

The last two chapters are Conclusions (Chapter 5) and Recommendations (Chapter 6). In the conclusions a summary is given of all the findings and the conclusion that was drawn with regards to the flow phenomena and resulting heat transfer behaviour.

The recommendations section contains some details regarding practical problems experienced during the project. Some suggestions are given with regards to ways to do similar investigations in a more efficient way.

Results obtained from the straight channel simulation configurations are tabulated in Appendix A.

Chapter 2 – Literature study

2.1 Nusselt number (Nu)

Calculations involving convective heat transfer from a surface to an internal flowing fluid require the value of the Nusselt number (Nu) to be available. The Nusselt number's value is unique for different flow conditions and is a non-dimensional parameter used to calculate the convection heat transfer coefficient (h).

The convection heat transfer coefficient is used to calculate the convection heat transfer between a solid material and an adjacent fluid. The heat transfer equation is given below.

$$h = \frac{Q}{A \cdot \Delta T} \quad (1)$$

A Heat transfer surface area

h Convection heat transfer coefficient

Q Heat flow

ΔT Difference in temperature between a solid surface and the surrounding fluid

The Nusselt number gives an indication of the local temperature gradient between a fluid flowing one side, and a solid material forming a wall on the other side (Rousseau, 2010). The temperature gradient is determined by the ratio of the convective to conductive heat transfer.

From the Nusselt number equation below it is seen that the convection heat transfer coefficient can be calculated if a numerical value for the Nusselt number is available.

$$Nu = \frac{h d_h}{k_f} \quad (2)$$

d_h Hydraulic diameter

k_f Fluid thermal conductivity

For thermal and hydraulically^{*} fully developed laminar conditions, the Nusselt number is a constant value independent of the Reynolds number (Re), Prandtl number (Pr), or axial location (x) within the channel, duct, or tube. It does however vary for different heat transfer boundary conditions.

The Nusselt numbers for the most common boundary conditions are $Nu = 4.36$ for a circular tube having uniform surface heat flux ($H1$) across the length of the tube and $Nu = 3.657$ for a

* Hydraulically fully developed flow refers to steady flow with a fully developed velocity profile.

circular tube having uniform wall surface temperature (T) along the length of the tube (Incropera et al. 2007).

The values for the Nusselt number are also determined by both the shape of the cross-section and the route of the channel path. In a straight channel, the Nusselt number will stay constant for a particular cross-section as long as the flow conditions stay laminar and fully developed. Figure 2.1 shows some values for the Nusselt number for different cross-sections for a uniform wall surface temperature (T).

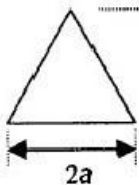
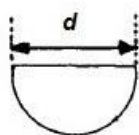
Duct shape	Hydraulic diameter	Nusselt number
Circular:		
	$d_h = 2a$	$Nu_T = 3.657$
Triangular (equilateral):		
	$d_h = \frac{4b}{3} \quad a = \frac{2b}{\sqrt{3}}$	$Nu_T = 2.49$
Semi-circular segment:		
	$d_h = \frac{d}{1 + \frac{2}{\pi}}$	$Nu_T = 3.323$

Figure 2.1 - Tabulated values of the Nusselt number for some cross-section shapes. Applies to fully developed laminar flow in straight channels typical of compact heat exchangers. a , b , d and d_h are some geometric properties. Uniform wall surface temperature boundary condition (T). Modified from Hesselgreaves (2001).

For turbulent flow the Nusselt number is a function of the Reynolds number and the Prandtl number and various correlations are available to calculate the Nusselt number value. One of the most commonly used correlations is the so-called Dittus-Boelter equation for flow in a circular duct (Rousseau, 2010).

This equation is not relevant to micro-channel flow since design conditions keep flow laminar at all locations within the flow domain. Steady laminar flow is preferred in micro-channel heat exchangers since unsteady flow in micro channels is associated with a decrease in heat transfer.

Unsteady flow starts to occur at $Re = 400$ and well below the region of transition between laminar and turbulent flow at $Re \cong 2300$ (Geyer et al. 2007).

The magnitude of the Nusselt number is directly proportional to the amount of heat exchanged between the solid wall material and the fluid flowing adjacent to the wall. For this reason, the Nusselt number obtained for a specific path layout (i.e. serpentine, trapezoidal) is compared with the Nusselt number for a straight path layout with the same cross-sectional shape. Doing the comparison in such a manner means that any increase in heat transfer could only be ascribed to the difference in the path layout.

Rosaguti et al. (2006) illustrated this method of comparison by using the same fluid, hydraulic diameter, cross-section shape, and path length for different layouts. Other parameters may similarly be isolated and their influence on the heat transfer may be investigated, to identify other possible enhancements.

From this comparative method, the heat transfer enhancement factor originated. The heat transfer enhancement factor (e_{Nu}) is defined below.

$$e_{Nu} = \frac{Nu_{path\ shape}}{Nu_{straight}} \quad (3)$$

$Nu_{path\ shape}$ Nusselt number for a specific channel type

$Nu_{straight}$ Nusselt number for a straight channel (given in figure 2.1)

Various studies have been conducted on micro-channel flow and heat transfer. Some studies have been presented in the following articles. Geyer et al. (2007) conducted a computational fluid dynamics (CFD) investigation on laminar fully developed flow and heat transfer from trapezoidal path shapes with semi-circular cross-sections. Gupta et al. (2008) conducted simulations on heat transfer from fully-developed laminar flow inside trapezoidal channels having equilateral triangle cross-sections. Rosaguti et al. (2006) conducted a CFD investigation on laminar fully developed flow and heat transfer from serpentine path shapes with semi-circular cross-sections.

Rosaguti et al. (2007) investigated heat transfer and fully-developed laminar flow in sinusoidal channels having circular- and semi-circular cross-sections. Rosaguti et al. (2005) used CFD to investigate heat transfer and fully-developed laminar flow within serpentine channels having circular cross-sections.

All the above-mentioned studies made use of the heat transfer enhancement factor (e_{Nu}) to illustrate improvements due to the different geometric configurations relative to straight channels.

The conclusions from these studies indicated that higher thermal performance resulted from tortuous path layouts compared to heat transfer in straight channels. The same trends were also observed for the different cross-section shapes that were investigated.

Gupta et al. (2008) calculated the thermal performance of an equilateral triangular cross-section and a triangular cross-section with one corner rounded. The results from the study was compared with a previous study done by Rosaguti et al. (2006), which calculated the thermal performance for a semi-circular cross-section.

Both studies investigated identical serpentine shaped channels. The particular cross-sectional shape swept along a serpentine path in periodic fashion. Three units of a serpentine channel are shown in figure 2.2.

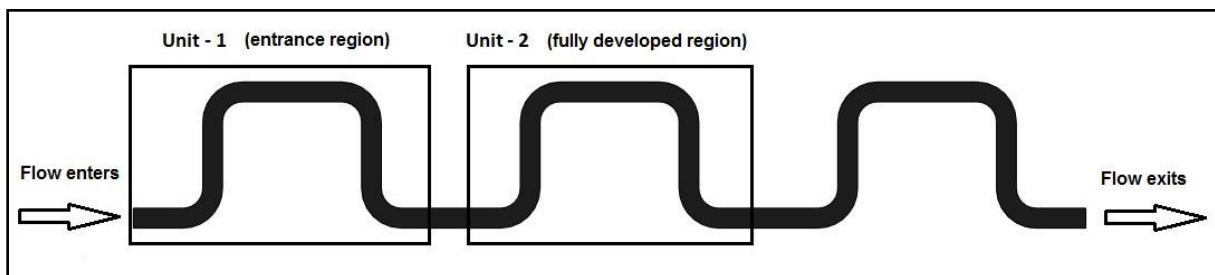


Figure 2.2 - Three units of a serpentine path shape.

Thermal-hydraulic conditions for both studies were identical. The particular conditions were fully developed laminar flow with a Reynolds number up to 200 and a constant hydraulic diameter for the flow cross-section throughout the length of the channel. The entrance length required to establish fully developed flow in serpentine channels was shown to be equivalent to the length of less than one unit. Flow developed in unit-1 (named the entrance region in figure 2.2) so that by the time it entered unit-2, the flow was fully developed.

During their studies, computational fluid dynamics was used to calculate the Nusselt number for the cross-sectional shapes under investigation. Additionally, Gupta et al. (2008) included the Nusselt number for the square cross-section which was found to have the best heat transfer performance of all the geometric configurations inside the scope of the investigation. Figure 2.3 shows the comparison of the results obtained for the different cross-sections following a serpentine channel layout.

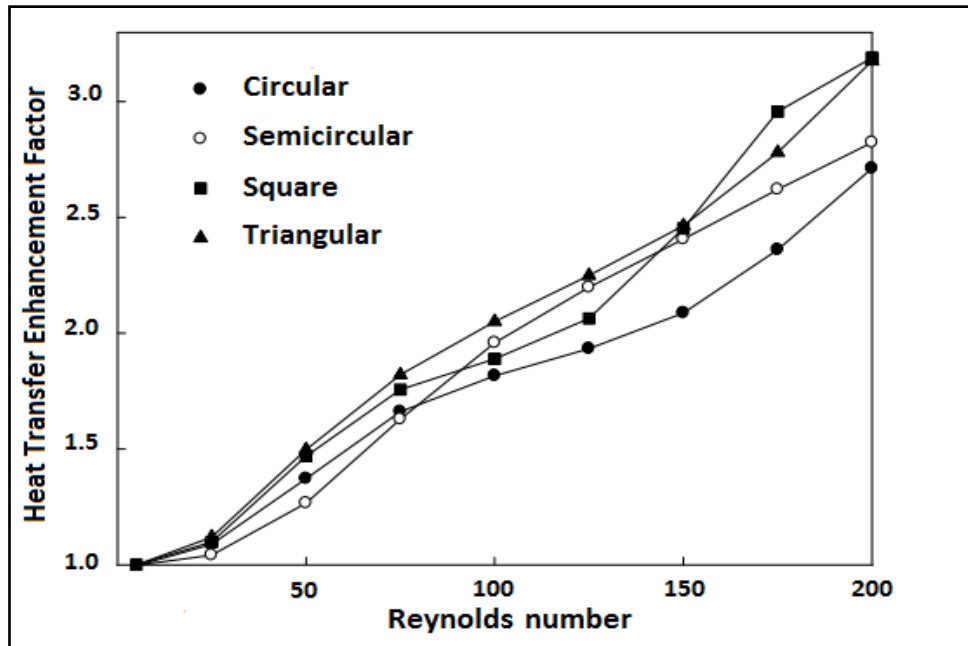


Figure 2.3 - Schematic showing the heat transfer enhancement factor (e_{Nu}) versus Reynolds number values for a serpentine channel having different cross-section shapes. Modified from Gupta et al. (2008).

In figure 2.3, the enhancement factor for the various cross-sections can be observed. It is evident that the cross-sections were all exhibiting comparatively the same trend across the range of Reynolds numbers. It is noted from the schematic that for $Re \geq 100$, all the non-circular shapes consistently performed better than the circular cross-section shape.

The heat transfer is directly proportional to the Reynolds number meaning that the thermal performance increased as flow sped up. It was also observed that the other cross-sections performed better than the circular cross-section, except at low Reynolds numbers.

In figure 2.4 a schematic is shown to explain the geometric parameters applicable to a serpentine shaped channel. A configuration was formed by setting three parameters to prescribed values. The three parameters were: the radius of curvature for the bends (R_c), the channel diameter (d) and the half wavelength of a single unit (L).

Results obtained from previous studies were presented by making use of dimensionless ratios known as the radius ratio $\frac{R_c}{d}$, and the half wavelength to channel diameter ratio $\frac{L}{d}$.

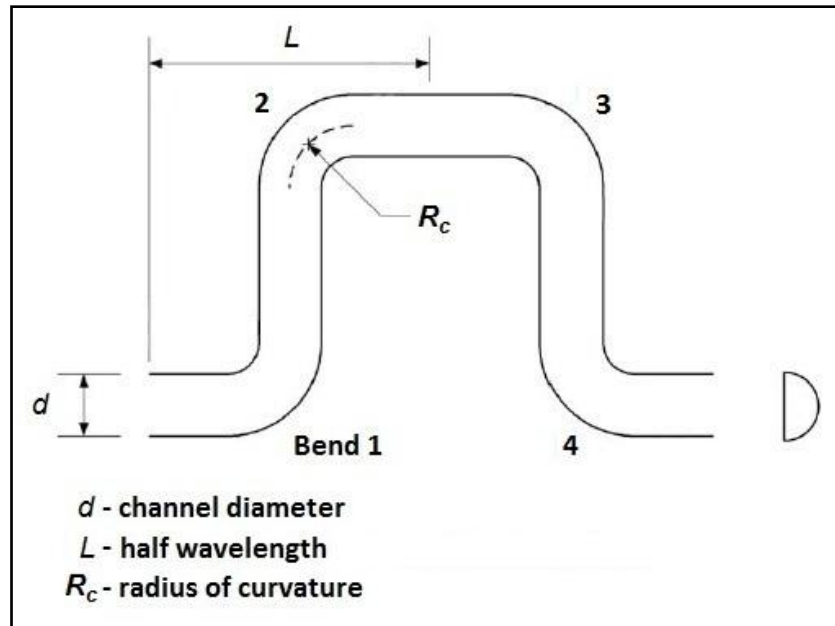


Figure 2.4 - Schematic showing the serpentine path shape with semi-circular cross-section. The geometric parameters, L , d and R_c are also shown. Modified from Rosaguti et al. (2006).

Rosaguti et al. (2006) determined the effect that the Reynolds number have on the heat transfer enhancement for various heat transfer boundary conditions. The heat transfer enhancement factor was calculated while incrementally changing the Reynolds number from 50 to 450.

Results of Reynolds number values below 50 showed negligibly small enhancements while flow characterized by Reynolds number values above 450 was seen to be unstable, resulting in lower heat transfer. The geometric configuration, characterized by the radius ratio $\frac{R_c}{d}$ and the half wavelength to channel diameter ratio $\frac{L}{d}$, was kept constant while varying the Reynolds number.

Figure 2.5 shows the results for both of the heat transfer boundary conditions, namely, uniform wall surface temperature (T) and uniform surface heat flux boundary condition with wall conductivity (H_2).

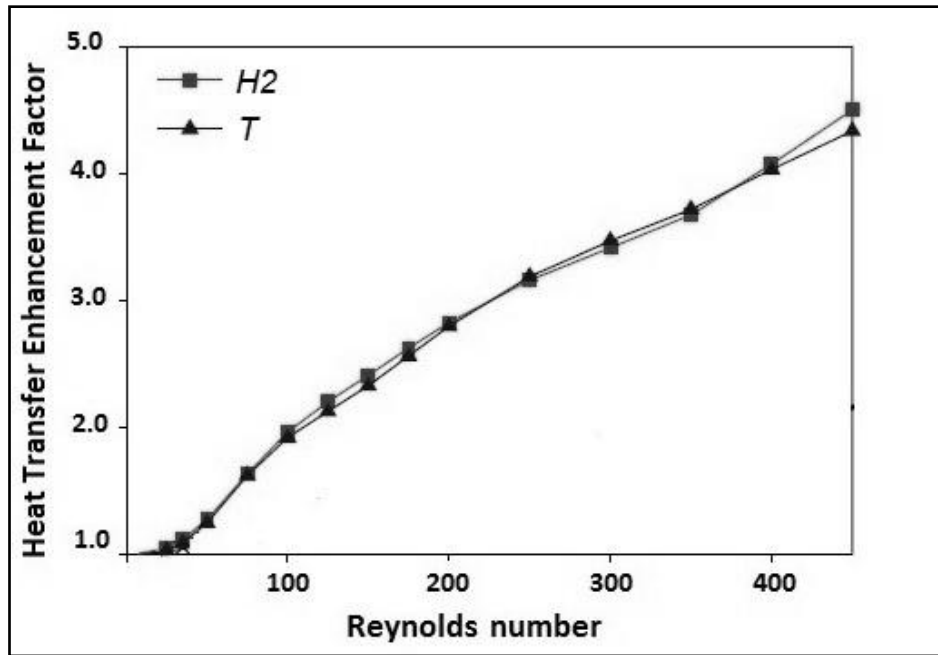


Figure 2.5 - Mean heat transfer enhancement factor (e_{Nu}) versus the Reynolds number for constant geometric ratios, $\frac{L}{d} = 4.5$, $\frac{R_c}{d} = 1$ and $Pr = 6.13$. Modified from Rosaguti et al. (2006).

It is observed that the enhancement factor increased as the Reynolds number increased and the values were very much the same for both of the thermal boundary conditions across the range. Rosaguti et al. (2006) showed that the heat transfer characteristics in serpentine channels could be 4 to 5 times better than in straight channels for both types of heat transfer boundary conditions.

2.2 Flow characterisation

From the literature, it became apparent that heat transfer at a particular point within a channel was largely dependent on the flow behaviour at that point. Flow altering events upstream influenced heat transfer taking place at the point downstream. Inducing any change in flow disrupted stable flow patterns within the fluid flow and subsequently improved the convection heat transfer between the wall surface and the fluid. This behaviour is called mixing. The biggest motivation behind the various investigations of micro-channel flow and heat transfer was the need to find the configuration that promotes maximum mixing (Rosaguti et al. 2005).

One way of enhancing the heat transfer during internal flow is by changing the fluid's flow direction. Changing the flow direction increases fluid mixing. Hot fluid in contact with the wall surface is forced to mix with cooler fluid in the centre of the channel cross-section thus increasing the temperature gradient at the wall surface.

Mixing causes disruptions in the thermal boundary layer and gives a more uniform fluid temperature profile as a function of the radius (Rosaguti et al. 2006). This behaviour means the fluid motion is used more efficiently in exchanging heat with the wall surface thus aiding the desired feature of increased surface area density in compact heat exchangers.

Serpentine channels make use of bends to create disruptions in the flow. A typical bend is shown in figure 2.4. Rosaguti et al. (2006) conducted a numerical investigation on fully developed laminar flow and heat transfer behaviour within serpentine channels with semi-circular cross-sections. The study revealed that bends rapidly changed the fluid momentum, having the effect of rotating the fluid around an axis parallel to the direction of flow creating so-called Dean's vortices.

The Dean vortices' strength changed after each bend. The vortices rotational direction changed after every second bend. Depending on the rotational direction of the flow and the direction of the upcoming bend, the bend cancelled or strengthened the Dean vortices and subsequently influenced the heat transfer performance of the fluid (Rosaguti et al. 2006).

It was shown that secondary flow structures such as Dean vortices could delay the onset of flow separation and turbulence. It meant that unsteady flow only started to occur at a Reynolds number of 350 and higher (Rosaguti et al. 2006).

The strength of the Dean vortices is a function of both the Reynolds number and the bends' geometric properties. This is evident when looking at the Dean number equation (Dn) presented by equation (4). The Dean number is dependent on the Reynolds number and is inversely related to the radius ratio $\frac{R_c}{d}$. The equation gives an indication of the ratio between the inertial forces (in this case centrifugal forces) to the viscous forces (Rosaguti et al. 2006).

$$Dn = Re \left(\frac{d}{R_c} \right)^{0.5} \quad (4)$$

Dn	Dean number
d	Diameter of cross-section (side length in case of equilateral triangle)
R_c	Radius of curvature
Re	Reynolds number

The Reynolds number can be calculated from equation (5) and can similarly be interpreted as the ratio between the inertial forces and the viscous forces (Rousseau, 2010).

This means that at low Reynolds numbers the secondary flow is suppressed when viscous forces are large compared to the inertial forces. This corresponds to a lower heat transfer enhancement. As the value of the Reynolds number increases, the strength of the vortices increases due to the influence from inertial forces (Rosaguti et al. 2006).

$$Re = \frac{\rho v d_h}{\mu} \quad (5)$$

v Flow velocity

μ Dynamic viscosity (relates stresses to linear deformation)

ρ Fluid density

Dean vortices have a considerable effect on heat transfer since they mix the fluid very efficiently, preventing the formation of a thermal boundary on the outer wall surface as was suggested by Schönfeld and Hardt (2004).

The effect bends have on heat transfer enhancement is shown in figure 2.6. Heat transfer seems to be highest at positions exiting bend-1 and bend-3. Because of the strong secondary flow induced, the Dean vortices have the largest influence following these bends.

Figure 2.7 shows the velocity contours following each bend in a single serpentine unit. By examining the velocity contours at positions corresponding to one diameter length downstream of the bends, the changes in the vortices' rotational direction are visible. The directions of rotation shown for the sections in the schematic have all been taken relative to the positive flow direction.

From figure 2.7, it is seen that the rotational directions of the vortices exiting bend-1 (section A-A) and bend-3 (section C-C) are opposite to another when looking in the direction of flow, and vortices exiting bend-2 (section B-B) and bend-4 (section D-D) are also rotating in opposite directions. From the bend angles, it can be determined that bend-1 and bend-4 have the same leftward angle and bend-2 and bend-3 are angled to the right. This means that vortices exiting bend-2 and bend-3 have a clockwise rotation direction, and vortices exiting bend-1 and bend-4 have an anti-clockwise rotational direction.

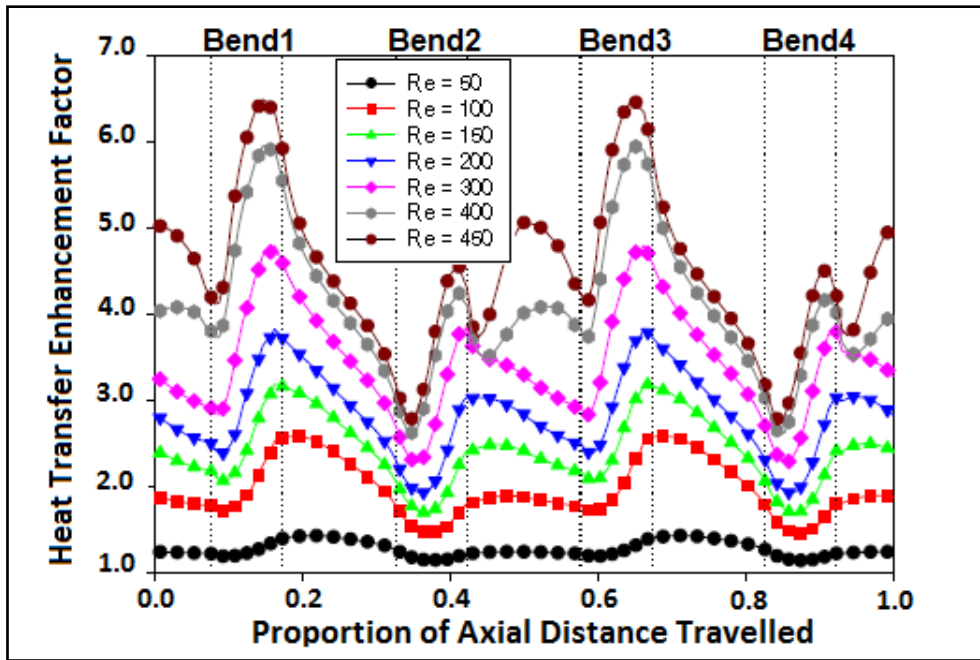


Figure 2.6 - Heat transfer enhancement (e_{Nu}) as function of the proportional distance within a single serpentine channel. The dashed lines indicate the entrance and exit positions of the four bends. The results apply to conditions corresponding to: $50 \leq Re \leq 450$, $Pr = 6.13$, $\frac{R_c}{d} = 1$ and $\frac{L}{d} = 4.5$ for the uniform heat flux boundary condition (H2). Modified from Rosaguti et al. (2006).

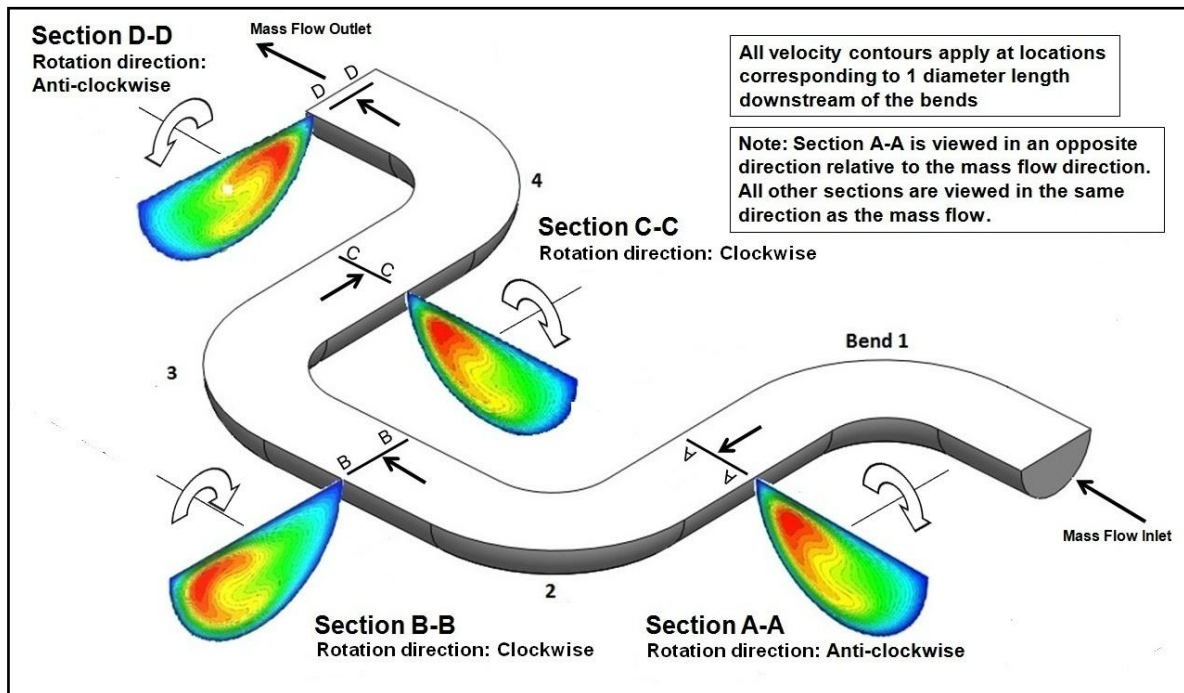


Figure 2.7 - Schematic showing the velocity contours downstream of the bends in a serpentine unit. The geometric ratios are set to, $\frac{L}{d} = 4.5$, $\frac{R_c}{d} = 1$ and $Re = 200$. The rotational directions are relative to the direction of flow. Velocity contours extracted from Rosaguti et al. (2006).

Following bend-1, the rotational direction of the flow reverses in bend-2 which weakens the effect that bend-2 has on the heat transfer. Similarly, following bend-3 there is a reversal of the rotational direction in bend-4 which results in the reduction of the heat transfer in bend-4. However, flow leaving bend-3 has the same rotational direction as bend-2, causing the vortices to be strengthened, enhancing the heat transfer in bend-3. Likewise the rotational direction at the exits of bend-4 and bend-1 remain the same, resulting in an enhancement of the heat transfer in bend-1 of the next unit, and so the process repeats for every unit in the channel.

The heat transfer enhancement that results from the increased rotational speed is clear when looking at the amplitudes of the enhancement factor present at bend-1 and bend-3 in figure 2.6.

Bend-1 and bend-3 are called curvature-reinforcing bends. Bend-2 and bend-4 are called alternating bends which disrupt the secondary flow induced by bend-1 and bend-3, and start to reverse the rotational direction of the vortices.

After exiting a bend, the flow recovers in straight channels between the bends for flow with $Re \leq 300$. The recovery stabilizes the flow, which considerably decreases both mixing and heat transfer to the centre of the bulk fluid. This effect can be seen in figure 2.6 in the sections between the bends. To counter this, short straight sections should prevent the flow from stabilizing.

For flow with $Re > 300$, the straight sections linking bend-4 to bend-1 and bend-2 to bend-3, do not seem to reduce the heat transfer significantly. The reason for this behaviour is still unclear.

In serpentine channel flow, the heat transfer is seen to be influenced by the bend geometries. The length of the straight sections, which influence the effect of the Dean vortices, is in turn determined by variations in the geometric parameters.

2.3 Geometric parameters

Now follows a detailed description of the various geometric parameters and their influence on heat transfer behaviour.

The effectiveness of mixing is influenced by three geometric parameters namely, the radius of curvature (R_c), the channel diameter (d), and the half wavelength of the channel (L) (shown in figure 2.8 and figure 2.9). The heat transfer enhancement factor was calculated for two specified geometric ratios namely, $\frac{R_c}{d}$ and $\frac{L}{d}$. The heat transfer enhancement factor for a semi-circular cross-section is shown for different values of the radius ratio $\frac{R_c}{d}$ in figure 2.8.

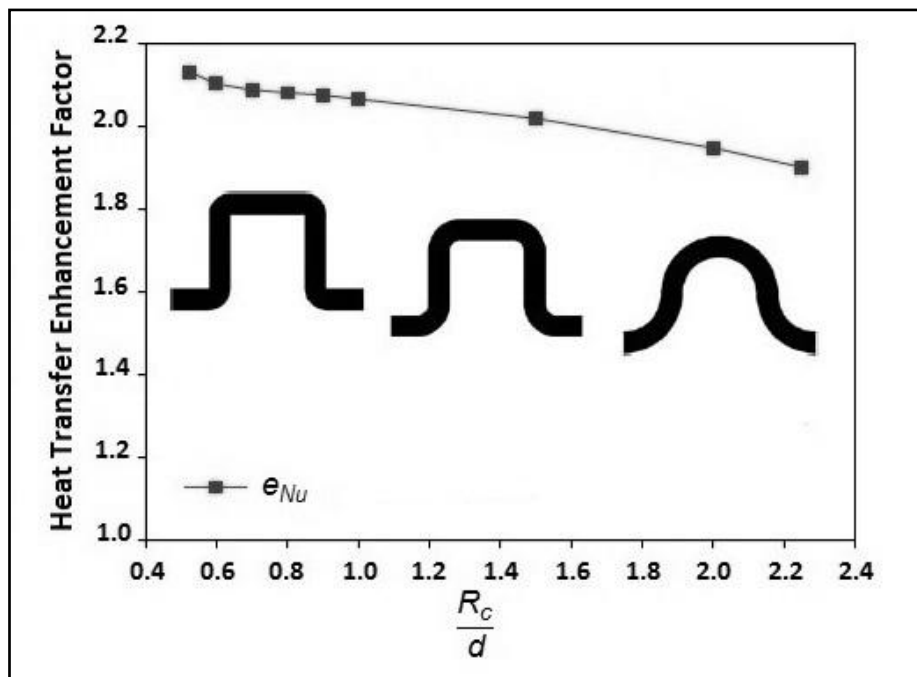


Figure 2.8 - Schematic showing heat transfer enhancement factor (e_{Nu}) versus the radius ratio $\frac{R_c}{d}$ in a semi-circular serpentine channel. $Re = 110$, $Pr = 6.13$ and $\frac{L}{d} = 4.5$ for the uniform heat flux boundary condition (H2). Modified from Rosaguti et al. (2006).

From the schematic, in figure 2.8, the enhancement factor can be seen to decrease approximately linearly as the radius ratio $\frac{R_c}{d}$ increases. Looking at a case where $\frac{R_c}{d}$ has a small value, assuming the value of the diameter (d) stays constant, the momentum will be forced to change rapidly through the bends and this will promote heat transfer in the bend regions.

The smaller radius was unfortunately followed by a longer straight section and since the heat transfer is decreased in the straight sections ultimately, there does not seem to be any significant advantage in using a very small radius of curvature (Rosaguti et al. 2006).

The second geometric ratio, for which heat transfer enhancement was investigated, is the half wavelength to channel diameter ratio $\frac{L}{d}$. Figure 2.9 shows the effects that this geometric ratio has on the heat transfer enhancement.

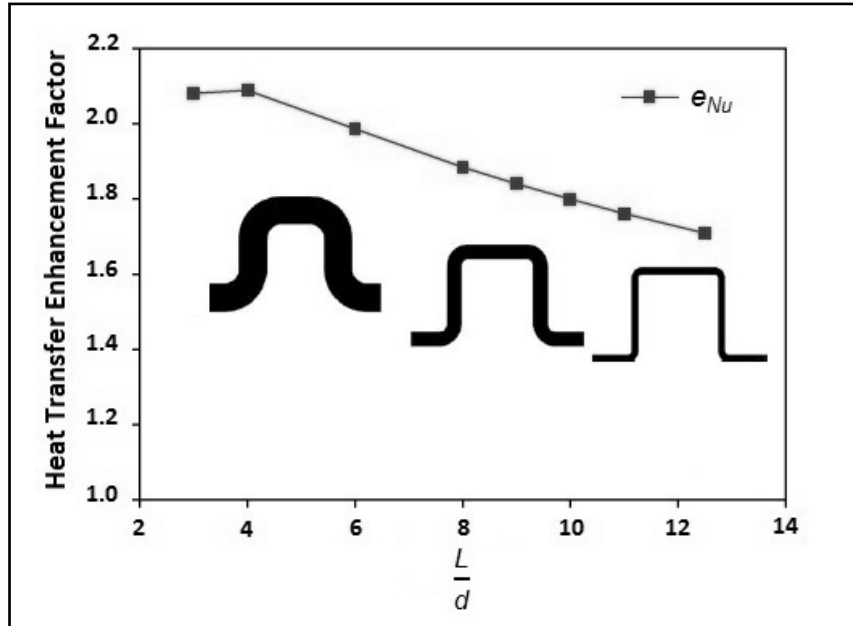


Figure 2.9 - Schematic showing heat transfer enhancement (e_{Nu}) versus the half wavelength to channel diameter ratio $\frac{L}{d}$. $Re = 110$, $Pr = 6.13$ and $\frac{R_c}{d} = 1$ for the uniform heat flux boundary condition (H2). Modified from Rosaguti et al. (2006).

Figure 2.9 shows a decrease in the heat transfer enhancement as the geometric ratio $\frac{L}{d}$ increases. This is due to the increased length of the straight sections causing the flow to stabilise, quickly dampening all the mixing effects produced by the bends (Rosaguti et al. 2006).

2.4 Reynolds number

The heat transfer enhancement factor for a semi-circular serpentine channel is shown for a range of Reynolds numbers in figure 2.10. Increasing the Reynolds number causes an increase in strength of the Dean vortices.

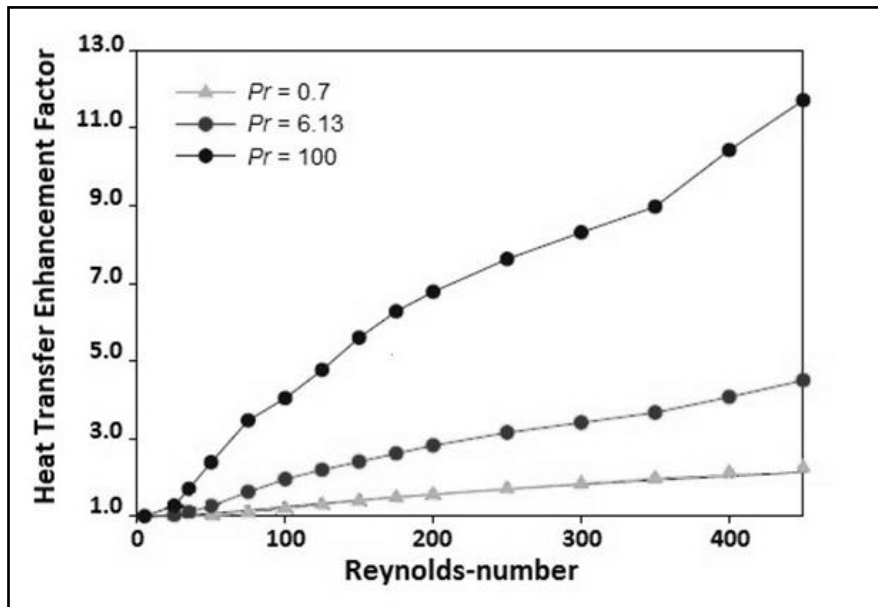


Figure 2.10 - Schematic showing the heat transfer enhancement factor (e_{Nu}) versus Reynolds number values for a semi-circular cross-section. Water has $Pr = 6.13$, $\frac{L}{d} = 4.5$, $\frac{R_c}{d} = 1$ for the uniform heat flux boundary condition (H2). Modified from Rosaguti et al. (2006).

According to Rosaguti et al. (2006), flow with a Reynolds number above 450 became unconditionally unsteady. For this reason, there exists a maximal limitation to the mass flow for which enhancement of the heat transfer in micro-channel heat exchangers occurs.

2.5 Prandtl number

The Prandtl number represents the ratio of a fluid's ability to transport momentum due to diffusion within the velocity boundary layer to the fluid's ability to conduct heat due to diffusion within the thermal boundary layer (Rousseau, 2010).

The heat transfer enhancement depends on the Prandtl number. Kalb and Seader (1972) showed that the Nusselt number is proportional to Pr^n . Fluids having a Prandtl number within the range, $0.7 < Pr < 175$, have $n = 0.2$ (Rosaguti et al. 2006). This range includes Prandtl number values for most gases and liquids such as water ($Pr = 6.13$). Figure 2.11 shows the enhancement factor for different Prandtl number values.

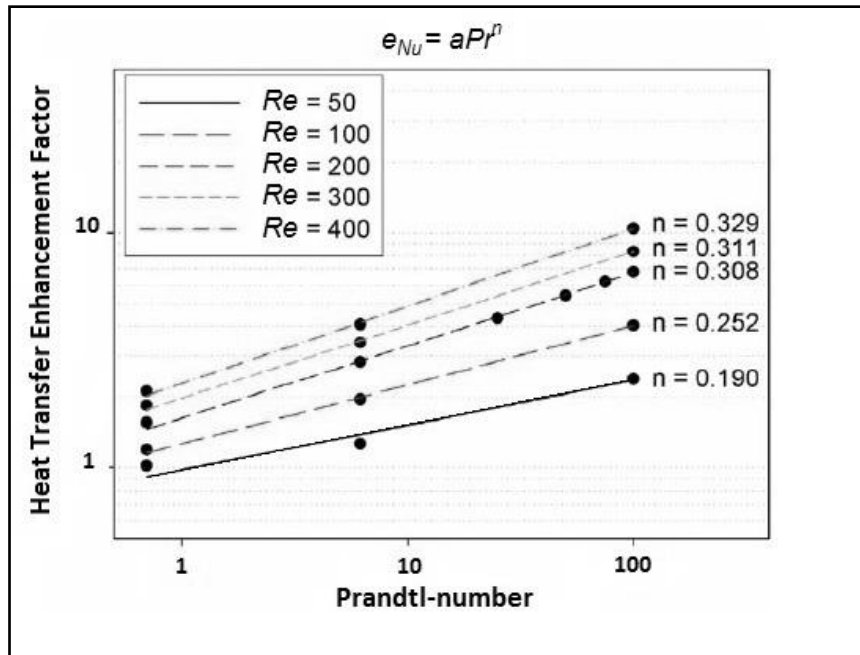


Figure 2.11 - Schematic shows the heat transfer enhancement factor (e_{Nu}) versus Prandtl number values. Modified from Rosaguti et al. (2006).

2.6 Chapter summary

From previous investigations on micro-channel heat transfer it was concluded that the most efficient way of representing results was to make use of the heat transfer enhancement factor which compared the Nusselt number value obtained for the tortuous channel with the theoretical Nusselt number value for a straight channel having the same cross-sectional shape.

The semi-circular and triangular cross-sections were to be expected to perform better than the circular cross-section in the serpentine channel as was seen in figure 2.3 for the Reynolds numbers under investigation.

Flow behaviour is mainly determined by the presence of Dean vortices which are responsible for the increased heat transfer. The strength and direction of the vortices vary depending on the location within the serpentine unit. Due to the vortices induced in the post-bend regions of the channel, heat transfer is significantly enhanced in these regions.

Flow was found to be stable up to Reynolds number values of 450, beyond this value; flow was seen to become unstable resulting in a decrease in overall heat transfer.

The channel geometry influences the strength of the Dean vortices and thus also the heat transfer. Reducing the bend radius causes more vigorous mixing, subsequently enhancing heat transfer. However the flow stabilizes in the straight sections between the bends, reducing the heat transfer enhancement again.

Increasing the mass flow rate causes an increase in the Reynolds number and subsequently a definite increase in heat transfer enhancement was noted as Re increased.

Chapter 3 – Simulations

The chapter is divided into three subsections. The first is the method of investigations which describes the workings of a general CFD program and how it was used in the current study. Then the scope section follows which gives the range for which the investigation was done including the assumptions and boundary conditions. Finally the chapter summary gives a synopsis of the most important information of chapter 3.

3.1 Method of investigation

The analysis was done using CD-adapco's STAR-CCM+ computational fluid dynamics application (CD-adapco, 2009).

The numerical investigation was split into two separate groups of test cases.

To verify the methodology of the calculation method used for the analysis, the first aim was to model straight channels with circular, triangular, and semi-circular cross-sections. Verification was considered to be completed when the Nusselt number values from the numerical models compared within acceptable tolerances with the empirical values of the cross-sections (shown in figure 2.1). The methodology used to compare the Nusselt number values, with the published values is described in the section 3.1.1.

The second group consisted of simulations of serpentine channels with semi-circular and triangular cross-sections for the conditions defined in the scope of investigation (described later in this chapter).

The method of investigation consists of four sections. The first section is the Nusselt number methodology detailing important equations and some literature about the Nusselt number. In the next section, the CFD methodology, a detailed explanation of the CFD process is given. Then follows the meshing section where the influencing parameters of the build-in mesher of STAR-CCM+ is discussed. Finally follows the physics section explaining the physical modelling process unique to STAR-CCM+.

3.1.1 Nusselt number methodology

From the work of Shah & London (1978) it is clear that the Nusselt number should start with very high values at the inlet of the channel and decrease until the Nusselt number value stabilizes at a value calculated at a distance downstream of the inlet corresponding to a dimensionless axial distance of 0.1 (figure 3.1). The value at which the Nusselt number remains constant is the empirical value given earlier in figure 2.1.

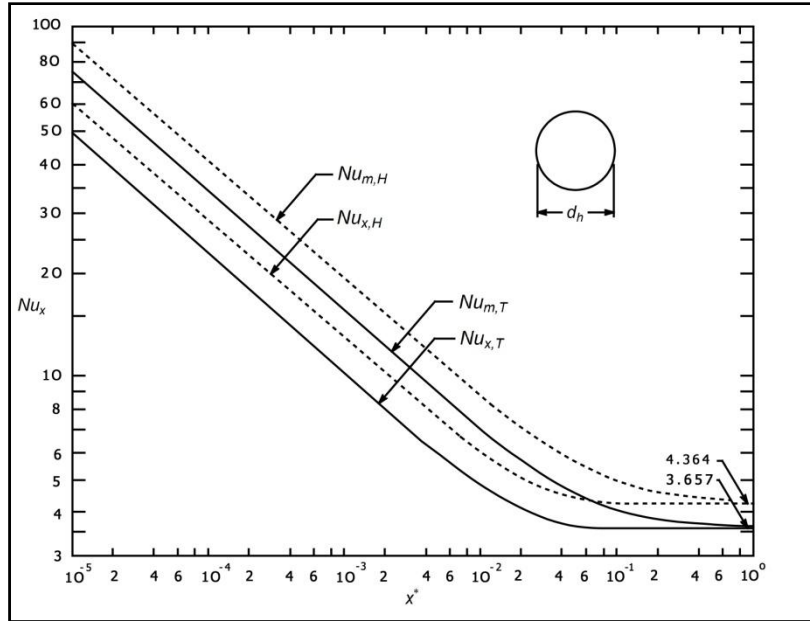


Figure 3.1 - Schematic showing the variation of the peripherally averaged Nusselt number (Nu_x) and the mean Nusselt number (Nu_m) versus the dimensionless axial distance (x^*) for uniform wall surface temperature (T) and uniform heat flux boundary conditions (H). Modified from Shah & London (1978).

Du Toit (2003), investigated a semi-circular channel to determine the value of the friction factor and Nusselt number for laminar flow. The numerical approach from the aforementioned study provided some equations to calculate the peripherally averaged local Nusselt number (Nu_x) for internal flow. The equations are stated below.

$$Nu_x = \frac{\bar{q}''_w}{(T_w - T_b)} \frac{d_h}{k_f} \quad (6)$$

$$T_b = \frac{\int_{\Omega} \rho c_p w T_f d\Omega}{\int_{\Omega} \rho c_p w d\Omega} \quad (7)$$

c_p Specific heat capacity at constant pressure

$\bar{q}''_w$ Peripherally averaged local wall heat flux

T_b Bulk fluid temperature

T_f Fluid temperature of a selected cell

T_w Wall surface temperature

w Axial flow velocity

Ω Integral area

Henceforth, the peripherally averaged local Nusselt number will be used directly as the Nusselt number.

Figure 3.1 shows the Nusselt number values for internal pipe flow, plotted on a logarithmic scale against the dimensionless axial distance for uniform surface temperature (T) and uniform surface heat flux (H) boundary conditions. The dimensionless axial distance is defined below.

$$x^* = \frac{x}{d_h Re Pr} \quad (8)$$

Pr Prandtl number

x Axial length from the start of heating at the Cartesian point of origin (0, 0, 0).

x^* Dimensionless axial distance

3.1.2 Computational fluid dynamics methodology

STAR-CCM+ and some other CFD software make use of the Finite Volume Method (FVM) to solve flow phenomena within a user specified physical region called the computational domain. The computational domain represents the region within which the numerical simulation should be conducted. The CFD software firstly divides the domain into smaller volumes according to the user's instructions. The subdivided domain is referred to as the grid or the mesh and consists of a large number of non-overlapping cells which are treated as Control Volumes (CVs).

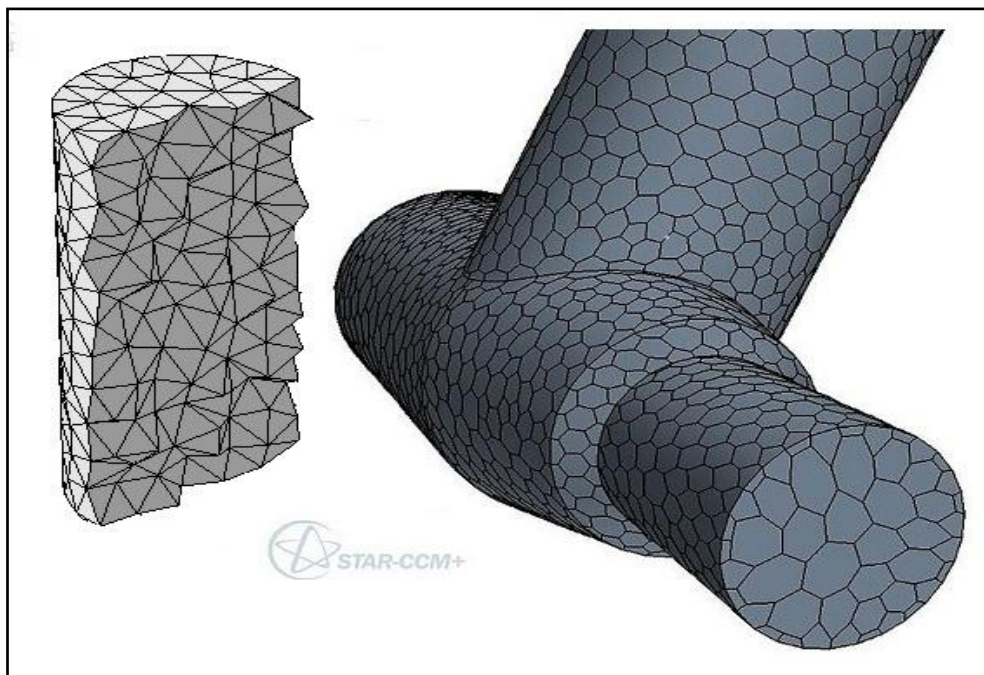


Figure 3.2 - Schematic shows two mesh examples with different cell shapes. The model on the left was meshed using tetrahedral cells and the model on the right was meshed using polyhedral cells.

The computational domain may consist of thousands or millions of individual cells (as shown in figure 3.2).

The CVs have a specific volume shape; in the case of STAR-CCM+ the CVs may be cubical, tetrahedral or polyhedral in shape. Figure 3.2 shows a cylindrical section on the left side of the schematic that was meshed using a tetrahedral mesher. The image on the right is a hammer-like component which was meshed using a polyhedral mesher.

The next step of modeling is to set up the physical conditions which apply to the model to be simulated. It includes, specifying the correct conditions such as: the viscous regime (laminar or turbulent), material selection, choosing between transient problems or steady state conditions, mesh motion presence or stationary modeling and many other physics options.

An important step in the modeling process is to specify the boundary conditions applicable to the domain wherein the simulation is to be done. Without correct boundary values the CFD code will not obtain an accurate solution or any solution at all.

CFD software makes use of a set of equations called the Navier-Stokes equations, to solve for mass-, momentum-, and energy balances in each control volume in the computational domain.

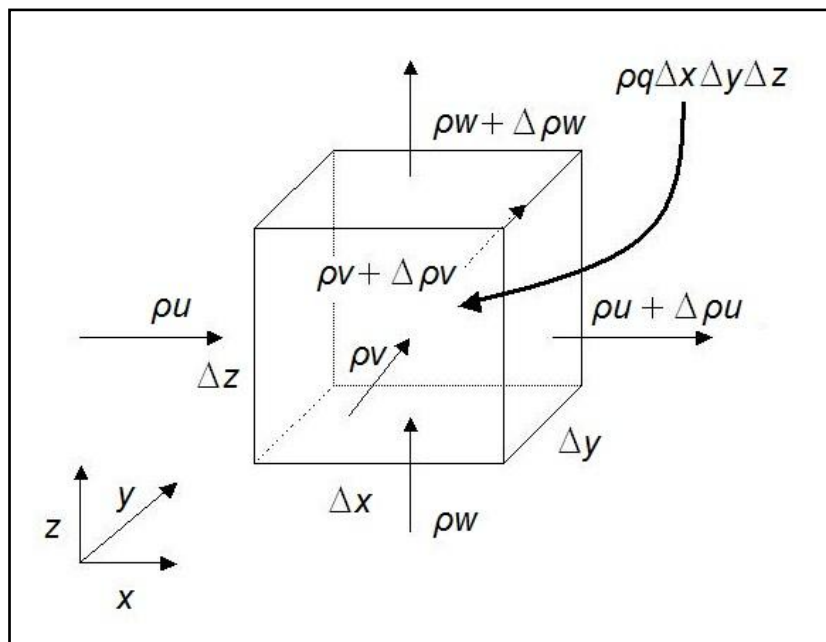


Figure 3.3 - Schematic shows mass flow through a hexagonal control volume of dimensions, Δx , Δy , Δz where u , v , w are the velocity components, ρ is the fluid density and q is the volumetric source rate.

The methodology was therefore derived from the fundamental conservation equations of mass, momentum and energy. In the case of the conservation of mass, the theory states that the rate of increase of mass in a control volume equals the net rate of flow into the control volume in the absence of a mass source. The same principle applies to the conservation of momentum and energy.

There are two discernible mechanisms by which mass, momentum or energy may cross the boundary of a control volume: diffusion and convection.

In figure 3.3, a control volume is shown through which mass enters and exits as indicated for the 3 axes of the Cartesian coordinate system.

Mass conservation principles were applied to the control volume in figure 3.3. The calculations and results are tabulated in Table 3.1. The calculations involved subtracting the outflow mass flow from the inlet mass flow for the x , y and z direction. The sum of the net mass flow rates for the three axes is the total net mass flow through the control volume.

Table 3.1 - The mass inflow, outflow and the net mass flow over the CV boundary in the x , y , and z directions of the CV shown in figure 3.3. Note that $\rho q \Delta x \Delta y \Delta z$ is the mass generation rate.

Direction	Inflow	Outflow	Net mass flow (Inflow – Outflow)
x	$\rho u \Delta y \Delta z$	$(\rho u + \Delta \rho u) \Delta y \Delta z$	$-\Delta \rho u \Delta y \Delta z$
y	$\rho v \Delta x \Delta z$	$(\rho v + \Delta \rho v) \Delta x \Delta z$	$-\Delta \rho v \Delta x \Delta z$
z	$\rho w \Delta x \Delta y$	$(\rho w + \Delta \rho w) \Delta x \Delta y$	$-\Delta \rho w \Delta x \Delta y$
Sum	$\rho u \Delta y \Delta z + \rho v \Delta x \Delta z + \rho w \Delta x \Delta y$	$(\rho u + \Delta \rho u) \Delta y \Delta z + (\rho v + \Delta \rho v) \Delta x \Delta z + (\rho w + \Delta \rho w) \Delta x \Delta y$	$(-\Delta \rho u \Delta x - \Delta \rho v \Delta y - \Delta \rho w \Delta z) \Delta x \Delta y \Delta z + \rho q \Delta x \Delta y \Delta z$

During the solution process of a CFD program, the Navier-Stokes equations (eq. (9) to (12)) are applied to mesh cells in the same manner as the calculations shown in Table 3.1. However, the Navier-Stokes equations need some modification before they are used. The Navier-Stokes equations in the most useful form for the development of the finite volume method are given by the equations below (Versteeg & Malalasekera, 2007).

x – momentum component

$$\rho \frac{Du}{Dt} = -\frac{dP}{dx} + \text{div}(\mu \text{grad } \vec{u}) + S_{M_x} \quad (9)$$

y – momentum component

$$\rho \frac{Dv}{Dt} = -\frac{dP}{dy} + \text{div}(\mu \text{grad } \vec{v}) + S_{M_y} \quad (10)$$

z – momentum component

$$\rho \frac{Dw}{Dt} = -\frac{dP}{dz} + \text{div}(\mu \text{grad } \vec{w}) + S_{M_z} \quad (11)$$

Internal energy

$$\rho \frac{Di}{Dt} = -P \text{div } \vec{u} + \text{div}(k \text{grad } T_s) + \Psi + S_i \quad (12)$$

where,

$$S_M = S_M + s_M \quad (13)$$

$$S_i = S_E - u \cdot S_M \quad (14)$$

$$\Psi = \mu \left\{ 2 \left[\left(\frac{du}{dx} \right)^2 + \left(\frac{dv}{dy} \right)^2 + \left(\frac{dw}{dz} \right)^2 \right] + \left(\frac{du}{dy} + \frac{dv}{dx} \right)^2 + \left(\frac{du}{dz} + \frac{dw}{dx} \right)^2 + \left(\frac{dv}{dz} + \frac{dw}{dy} \right)^2 \right\} + \lambda (\text{div } \vec{u})^2 \quad (15)$$

i Internal energy

P Pressure

S_E Energy source term (including effects from potential energy changes)

S_M Momentum source term (including the effects from surface stresses)

s_M Viscous stress terms

T_s Substance temperature

$\vec{u}, \vec{v}, \vec{w}$ Velocity vectors in Cartesian coordinate system

u, v, w Velocity components in Cartesian coordinate system

x, y, z Control volume side lengths

λ Secondary viscosity (relates stresses to the volumetric deformation)

Ψ Internal energy source due to viscosity effects

The more conservative form of the general equations for Newtonian compressible fluids is given below for time-dependant three-dimensional flow (Versteeg & Malalasekera, 2007).

Continuity equation

$$\frac{d\rho}{dt} + \text{div}(\rho \vec{u}) = 0 \quad (16)$$

x – momentum component

$$\frac{d(\rho u)}{dt} + \text{div}(\rho u \vec{u}) = -\frac{dP}{dx} + \text{div}(\mu \text{grad } u) + S_{M_x} \quad (17)$$

y – momentum component

$$\frac{d(\rho v)}{dt} + \text{div}(\rho v \vec{u}) = -\frac{dP}{dy} + \text{div}(\mu \text{grad } v) + S_{M_y} \quad (18)$$

z – momentum component

$$\frac{d(\rho w)}{dt} + \text{div}(\rho w \vec{u}) = -\frac{dP}{dz} + \text{div}(\mu \text{grad } w) + S_{M_z} \quad (19)$$

Internal energy $\frac{d(\rho i)}{dt} + \text{div}(\rho i \vec{u}) = -P \text{div}(\vec{u}) + \text{div}(k \text{grad } T_s) + \Psi + S_i$ (20)

Equations of state $P = P(\rho, T_s) \text{ and } i = i(\rho, T_s)$ (21)

$$P = \rho R T_s \text{ and } i = c_v T_s \quad (22)$$

c_v Specific heat capacity at constant volume

R Specific gas constant

Due to the commonalities of the above listed equations (eq. (17) to (20)); it is possible to insert a common property ϕ , which may represent any property such as: temperature, pressure etc.

The equations above may be rewritten into the following form.

$$\frac{d(\rho \phi)}{dt} + \text{div}(\rho \phi \vec{u}) = \text{div}(\Gamma \text{grad } \phi) + S_\phi \quad (23)$$

S_ϕ Source term of property ϕ

Γ Diffusion coefficient

ϕ Property to be calculated

The above equation (eq. (23)) is known as the general transport equation for property ϕ . The first term is the rate of change term of property ϕ , the second term from the left is the convective term, the first term after the equal sign is the diffusion term and the last term on the right is the source term (Versteeg & Malalasekera, 2007).

By applying Gauss's theorem (Ferziger & Peric, 2002) to the convective and diffusive terms of the steady state transport equation, the following algebraic formulation is obtained.

$$\sum_{f=1}^n \rho_f \phi_f \vec{u}_f \cdot \vec{A}_f = \sum_{f=1}^n \Gamma_f (\nabla \phi)_f \cdot \vec{A}_f + S_{\phi_P} V_P \quad (24)$$

$\vec{A}_f$ Outward pointing face area vector (refer to figure 3.4b)

S_{ϕ_P} Volumetric source term of property ϕ

$\vec{u}_f$ Velocity component in direction of the face area vector

V_P Volume of cell P

$\vec{\nabla}$ Del, geometric specific gradient operator

Γ_f Diffusion coefficient in the direction of the face area vector

The transport equation may be integrated over non-overlapping control volumes with n number of faces, where the number of faces depends on the individual control volume shape.

For the control volume presented in figure 3.3 the transport equation is integrated over $n = 6$ flat faces for the hexagonal control volume.

Assumptions applied during the derivation of the above formulation given in equation (24) include the following: the fluid properties, property gradients and/or sources are calculated at the centre of the control volumes and assumed to represent the whole volume. The values of any property, gradient or source strength at a connecting face between two adjacent control volumes are determined by interpolation between cell centres (Ferziger & Peric, 2002). There exist a number of interpolation techniques to acquire the value of the property in question.

Depending on the conditions under investigation, different interpolation techniques may produce more suitable solutions.

STAR-CCM+ can make use of upwind differencing, central differencing or quadratic upwind differencing. By default STAR-CCM+ makes use of a second order differencing scheme since second order schemes have been proven to be more accurate for most conditions of flow (Versteeg & Malalasekera, 2007).

STAR-CCM+ gives the user an option to select a different interpolation scheme during the pre-processing stage of the simulation.

Consider the one-dimensional case of a control volume P and its adjacent neighbours denoted as W and E , with connecting faces w and e (shown in figure 3.4c).

In the absence of a source term, only the diffusive and convective terms of the transport equation (eq. 23) remain.

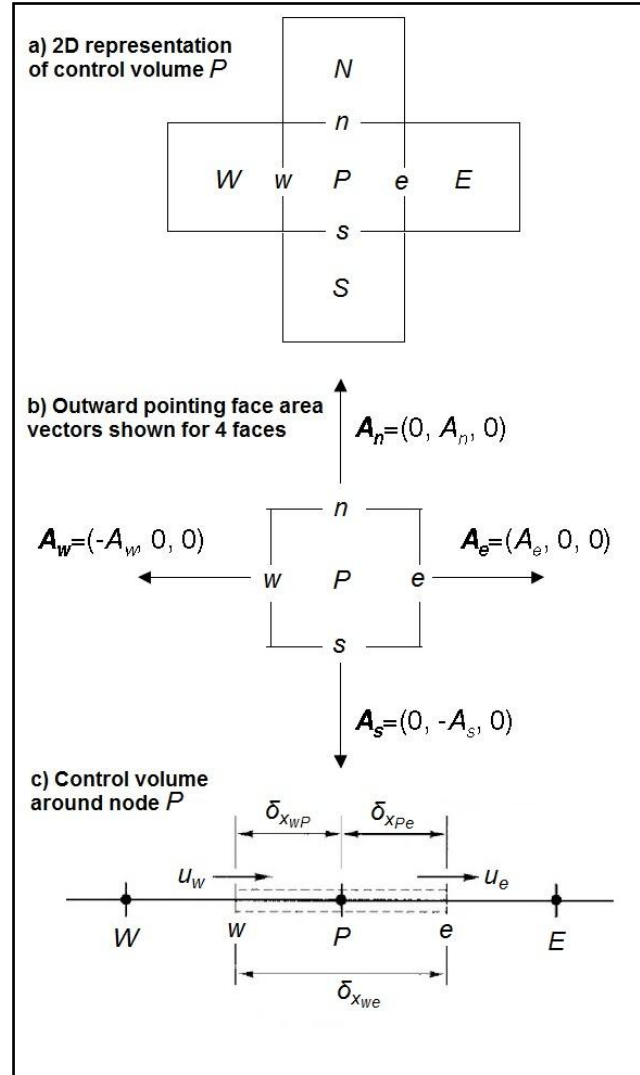


Figure 3.4 - In a) the schematic shows a 2D illustration of node P and its adjacent 2D neighbouring cell nodes (N , S , W and E). In b) the schematic shows the four outward pointing face area vectors for each side of cell P . n , s , w and e represent the neighbouring faces in the 2D directions of node P . The illustration in c) shows the 1D control volume at node P and its adjacent neighbours, W and E . The notations w and e represent the connecting faces of node P with node W and E respectively. δ_x is the distance between two relevant nodes or faces.

Similar to the conservation of mass applied to the control volume in figure 3.3, the conservation of property ϕ can be applied. The transport equation (eq. (23)) is integrated over the control volume P to solve for property ϕ .

$$(\rho u A \phi)_e - (\rho u A \phi)_w = \left(\Gamma A \frac{d\phi}{dx} \right)_e - \left(\Gamma A \frac{d\phi}{dx} \right)_w \quad (25)$$

Integration of the continuity equation (eq. 16) gives the following.

$$(\rho u A)_e - (\rho u A)_w = 0 \quad (26)$$

Similarly the transport equation is applied to the other nodes in the model (W and E in figure 3.4c). By making some approximations, simplifications, assumptions and applying a differencing scheme, a matrix is formed consisting of n -equations each corresponding to a node in the model. At this point a solution procedure is needed to solve the matrix.

From equation (25), the transport of property ϕ is dependent on the direction and the magnitude of the velocity field within the computational domain. No property distribution can be solved without knowing the velocity field.

The velocity field is however not known and needs to be solved just like the rest of the flow variables. In order to solve for all the flow variables, the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm can be used (Versteeg & Malalasekera, 2007).

SIMPLE is an iterative algorithm for calculating the velocities and pressures in a flow domain. This is done because the pressure and velocity fields in the flow domain are strongly linked to each other; therefore it needs to be solved in a coupled manner. The SIMPLE algorithm for the Navier-Stokes equations is a stepwise process, which is explained as follows.

- 1) Guess a realistic pressure field P^* .
- 2) Insert the guessed pressure values and solve the corresponding system of velocities.

$$u_P^* = \frac{\sum_{nb} a_{nb} u_{nb}^*}{a_P} + \frac{S_U}{a_P} - \frac{V_P}{a_P} \left(\frac{dP^*}{dx} \right)_P \quad (27)$$

a	Coefficient describing convective and diffusive effects
nb (subscript)	Refers to neighbouring cell faces
P (subscript)	Refers to node P
S_U	Momentum source term for the U cell
u^*	Velocity guess value
V	Volume of cell

- 3) Solve the Pressure-correction equation, (derived from the mass conservation equations) given by equation (28) and equation (29).

$$(\rho_e d_e A_e + \rho_w d_w A_w) P'_P = \rho_e d_e A_e P'_E + \rho_w d_w A_w P'_W + \left[\rho_w u_w^* A_w - \rho_e u_e^* A_e + \rho_w G_w^* A_w - \rho_e G_e^* A_e \right] \quad (28)$$

$$a_P P_P = a_E P_E + a_W P_W + S_P \quad (29)$$

where,

$$a_E = \rho_e d_e A_e, \quad a_W = \rho_w d_w A_w, \quad a_P = a_W + a_E,$$

$$\text{and } S_P = \left[\rho_w u_w^* A_w - \rho_e u_e^* A_e + \rho_w G_w^* A_w - \rho_e G_e^* A_e \right]$$

- 4) Do the velocity correction with equation (30), the pressure correction with equation (31), the face velocity corrections with equation (32) and equation (33), and finally calculate the updated mass flux with equation (34) and equation (35):

$$u_P = u_P^* - \frac{V_P}{a_{PP}} \left(\frac{dP'}{dx} \right)_P \quad (30)$$

$$P = P^* + P' \Rightarrow P' = P - P^* \quad (31)$$

$$u_w = u_w^* + G_w^* - d_w (P'_P - P'_W) \quad (32)$$

$$u_e = u_e^* + G_e^* - d_e (P'_E - P'_P) \quad (33)$$

$$F_e = \rho_e u_e A_e \quad (34)$$

$$F_w = \rho_w u_w A_w \quad (35)$$

where,

$$d = \frac{A}{a} \quad (36)$$

F	Mass flux
G^*	Guessed Rhie and Chow correction term
P'	Difference between the final pressure P and the guessed pressure P^*
P^*	Pressure guess value

- 5) Using the updated mass flux (F), solve all the transport equations for the variable ϕ .
- 6) Insert the corrected pressure obtained from equation (31), as the guessed pressure P^* in equation (27), and repeat the steps defined for the SIMPLE algorithm until convergence is reached.

When the pressure P is correct then the correct velocity for mass conservation is obtained. By making use of the SIMPLE method all the variables are solved iteratively in a coupled system.

Given sufficient boundary conditions, the $n \times n$ matrix can be solved (Versteeg & Malalasekera, 2007). The larger the size of the matrix, the more time it takes to solve since the SIMPLE algorithm has to be applied to more cells. Dividing a flow domain into a large number of CVs (small CVs) causes the algorithm to solve slower. The upside to using smaller CVs is that the simulation can produce a more accurate solution.

3.1.3 Meshing

The first step in creating an accurate CFD model is to create a mesh of adequate resolution. For internal flow, the surface extruder, polyhedral- and prism-layer mesher of STAR-CCM+ can be used. Prism layers are located on the outer surface of the domain where the velocity- and thermal boundary layers are present. It is important for internal flow models to have a sufficient number of cells within the region closest to the wall; otherwise the calculation of the boundary layer may be inaccurate.

A number of important parameters need to be set in order to create a mesh for sufficient accuracy of the results. The most important parameters for the automatic, unstructured mesh generator is given below in the order in which they are discussed:

- Base size
- Prism layer stretching
- Prism layer thickness
- Number of prism layers
- Surface size - Relative cell target size
- Surface size – Relative minimum cell size

The base size accepts a numerical value which is preferably in the same size range as the smallest significant model features. For example in this investigation the channel diameter was 2 mm and the length was longer than 100 mm. It is then necessary to choose a base size of less than 2 mm to fit enough cells on the cross-section face. The base size has no direct effect on the cell size but rather through the relative cell target size and relative minimum cell size which is assigned as percentage values of the base size. To fit approximately 10 cells on a cross-section face of the channel, a base size of 0.2 mm would be assigned with a relative target size of 100% the base size.

The prism layer-stretching factor is a useful tool if greater accuracy is required near the wall of the channel. The parameter determines the thickness of each prism layer.

If the factor was set to 1.3, it meant that when looking at the adjacent layers in the radial direction of the channel from the boundary to the centre, any layer going to the centre had a thickness of 1.3 times thicker than its preceding neighbor. Unfortunately, small changes of the stretching factor could considerably increase the simulation time if cells become to “thin”. The stretching factor was set to a value of one in this study since it was essential to keep the size of the cells uniform within the velocity- and thermal boundary layer.

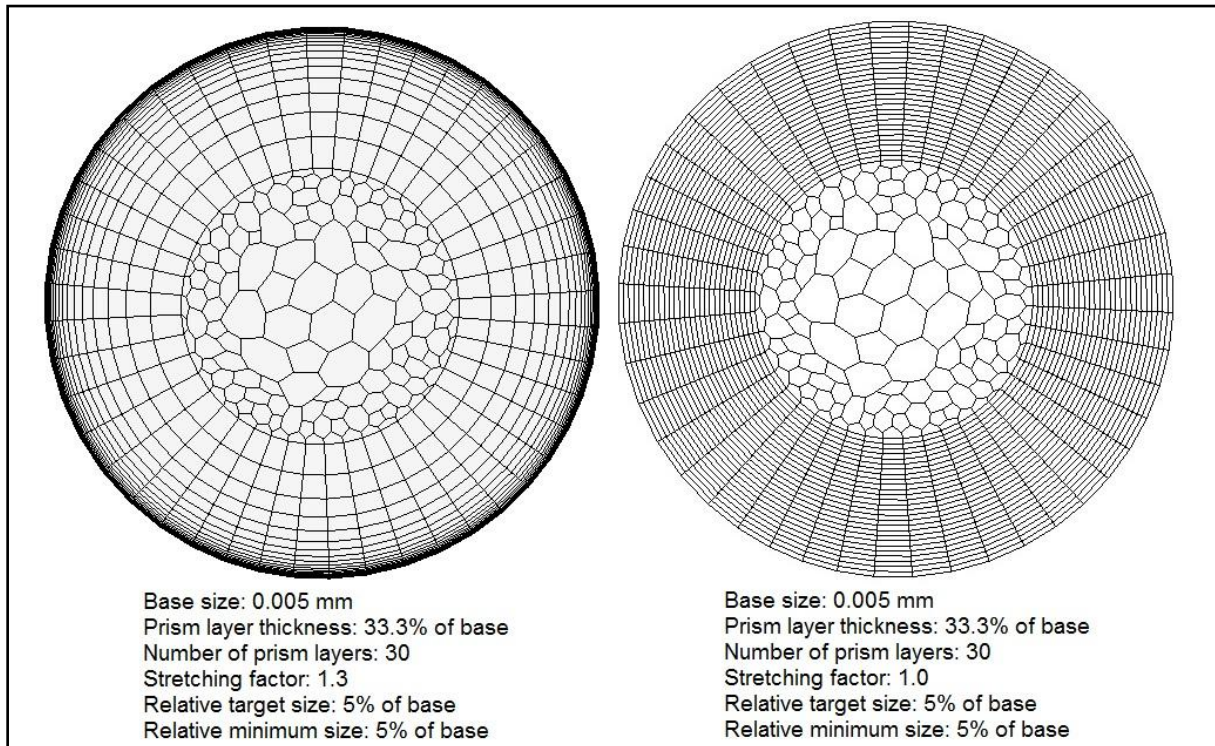


Figure 3.5 – Schematics showing differences within the prism layer section for meshes having stretching factors of 1.3 (left) and 1 (right). Note that these are not the exact models used during the simulations.

Figure 3.5 shows an exaggerated prism layer region to demonstrate how the cell sizes change from the wall boundary toward the centre when the stretching factor is adjusted. For a stretching factor higher than 1, the prism layers closest to the centre of the channel became too thick and inaccurate. It can be seen in the left schematic of figure 3.5, how large the PL cells nearest to the centre become for the mesh with the stretching factor set to 1.3.

The most important parameter is the overall prism layer thickness. If the thickness is set too small, the velocity and temperature boundary layers are not accurately resolved. In such cases, the models frequently give inaccurate and non-converging results. Of all the parameters, the thickness proved to be the most influential in providing accurate results for the Nusselt number calculations.

A uniform cell size was desired in the prism layer region, thus the thickness parameter was set to the highest value corresponding to a smooth and uniform prism layers region, as shown in figure 3.6a.

A problem experienced with the meshing of the semi-circular and triangular cross-sections was the distortion of the face cells when the prism layer thickness was set too large (shown in figure 3.6b). The problem occurred mainly due to the sharp corners in the geometry. Compared to figure 3.5, the same problem is not present in the circular channel.

It is important to note that the models shown in figure 3.5 and figure 3.6 are not the actual models used in simulations. The models have only been set up for demonstrative purposes.

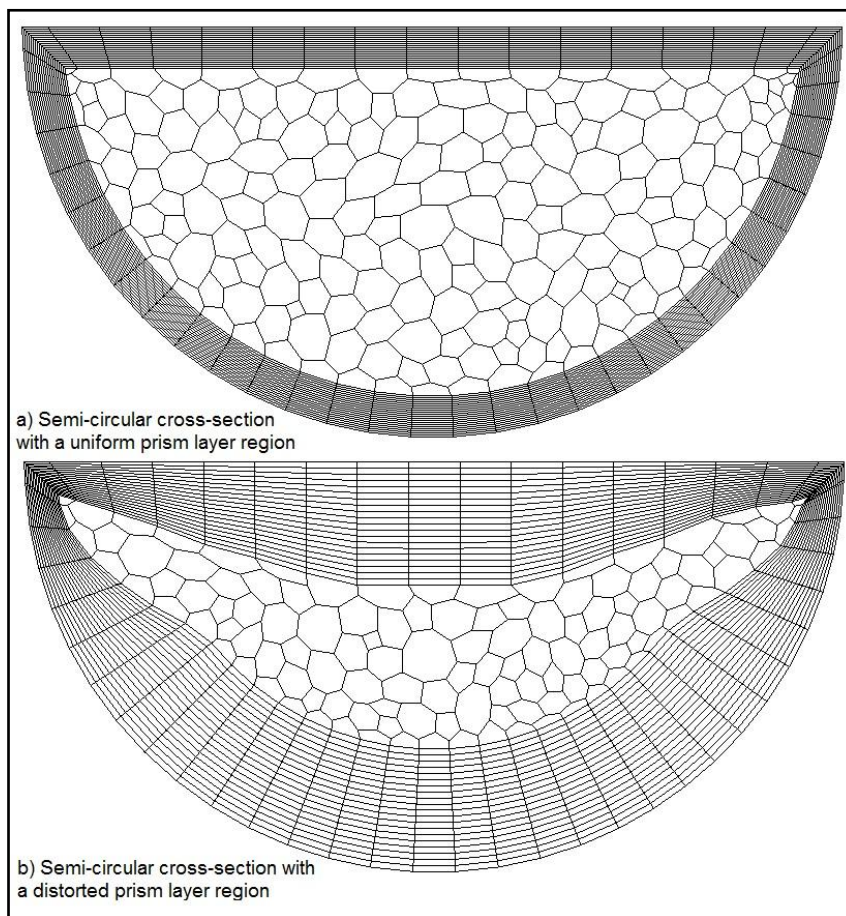


Figure 3.6 - Snapshot of the distortion occurring in b) when the prism layer thickness is set too high for the corresponding cell target size. a) shows a cross-section with a uniform prism layer thickness. Note that these are not the exact models used during the simulations.

The next important mesh parameter in STAR-CCM+ is the number of prism layers created in the prism layer region. This parameter determines the number of cells in the prism layer region but not the overall thickness of the prism layer region.

Increasing the number of prism layers decreased the thickness of those cells in the prism layer region. Too many prism layers negatively influenced the simulation time because increasing the number of layers slightly, dramatically increased the total number of cells. Using about 20 prism layers seemed to be an optimal number. The simulation time remained within practical limits and the Nusselt number calculations, exhibited acceptable grid independence qualities.

The relative target size determined the size of the polyhedral cells, which were situated in the central region of the channel. The Nusselt number variations obtained by varying the relative target size seemed to converge without requiring too many cells. The results seemed to be influenced much more by the prism layer parameters, which directly affected the boundary layer calculations.

The first step in both parts of the numerical investigation was to obtain mesh independence for the models.

To establish mesh independence, an iterative process was used where the difference between results obtained during consecutive runs of different mesh quality (one mesh being a finer quality mesh than its predecessor) indicated the degree of refinement necessary to achieve accurate results. The mesh quality was improved by making small changes to the mesh parameters until the difference in results between runs was negligibly small. Refinement beyond such a point was not practical since no useful improvement in the results would be obtained.

3.1.4 Physics

For all STAR-CCM+ simulations done during the investigation, the same physics model was used. The physics options were set to the following:

- Space – Three dimensional
- Motion – Stationary
- Time – Steady
- Material – Liquid (Water)
- Flow – Segregated
- Equation of state – Constant density
- Viscous regime – Laminar
- Optional physics models – Segregated fluid temperature

STAR-CCM+ was extended to apply the Nusselt number equation (eq. (6)) from Du Toit (2003) as a user defined field function (eq. (37)).

A report is a calculation done during individual iterations in the solution process and is one of the mechanisms in the software which is used to extend functionality.

For a particular property (e.g. temperature, pressure, etc) specified by the user, the report uses a pre-defined formula to calculate the property. To explain how the equations (eq. (6) and eq. (7)) were implemented, the following is helpful.

$$User\ field\ function_{Nusselt\ number} = \left(\frac{Report_{Boundary\ heat\ flux}}{Report_{Bulk\ fluid\ temperature} - T_w} \right) \left(\frac{d_h}{k_f} \right) \quad (37)$$

A mass-flow averaged report (eq. (38)) was used to calculate the bulk fluid temperature (ϕ_f) for a cross-sectional area at a specific axial location. The report is formulated in the Star-CCM+ manual as follows (CD-adapco, 2009).

Report_{Bulk fluid temperature}

$$Mass\ Flow\ Averaged = \frac{\sum_f \rho_f \phi_f / \vec{V}_f \cdot \vec{A}_f}{\sum_f \rho_f / \vec{V}_f \cdot \vec{A}_f} \quad (38)$$

$\vec{V}_f$ Face velocity vector

ρ_f Face density

ϕ_f Scalar property (bulk fluid temperature)

The final report was an area averaged report (eq. (39)) applied at the wall. The boundary heat flux (ϕ_f) was supplied to the report as the scalar property to be averaged.

The report is formulated in the Star-CCM+ User Guide as follows (CD-adapco, 2009).

Report_{Boundary heat flux}

$$Area\ Average = \frac{\sum_f \phi_f / A_f}{\sum_f 1 / A_f} \quad (39)$$

The user specifies the wall temperature (T_w), hydraulic diameter and the fluid thermal conductivity (k_f).

The input part to the mass-flow averaged report (eq. (38)) is a cross-section plane through the fluid computational domain, perpendicular to the flow direction. The input part to the user field function (eq. (37)) to calculate the Nusselt number, and the report to calculate the

boundary heat-flux (eq. 39), is the intersection perimeter of a cross-section plane and the channel wall.

Both above mentioned planes were positioned at the same axial location where the Nusselt number was calculated. It was further possible to calculate the Nusselt numbers at multiple axial locations in the channel by creating cross-section planes at those locations.

3.2 Scope

Now follows the scope section, primarily defining the study conditions. The first section states all the relevant assumptions applying to the study. Similarly in the next section, the boundary conditions applied to the simulations are listed. Then follows the procedure whereby the straight channels were investigated and in the final section the same is discussed about the serpentine channel investigations.

3.2.1 Assumptions

Some assumptions had been made for the simulations done on straight- and serpentine channels. The assumptions are as follows:

- All methods treated the effects of viscous dissipation and variations in heat conduction within the fluid as negligible.
- All fluid properties were assumed to remain constant.
- The value applied for the fluid's thermal conductivity was estimated as the average value for the fluid inlet temperature and a fluid exit temperature that corresponded to 80% of the wall temperature. The approximation stemmed from the fact that most heat exchangers have an effectiveness of about 80%, which yielded a fluid exit temperature of 340 K in this study.

3.2.2 Boundary conditions

The following boundary conditions were applied to the straight- and serpentine channels:

- Heating in all models occurred from the wall to the fluid, with a uniform wall surface temperature present from the inlet to the outlet. The wall was maintained at a temperature (T_w) of 350 K, while the fluid entered the channel at 300 K for the simulations done.
- The inlet plane interface of the straight- and serpentine channels was set to a velocity inlet mode and the appropriate velocity was then assigned. For the straight channels, the relevant velocities are listed in Table 3.2 for the particular cross-section shapes and in Table 3.3 and Table 3.4 for the serpentine channels.

- The exit plane interface was specified to the mass flow split mode. The pressure outlet option could not be used because the pressure drop across the channel section was unknown. These settings were selected in STAR-CCM+ at the boundary type option for the particular part. The wall part of the channel was set to wall mode with the uniform surface temperature option activated and the abovementioned wall temperature assigned to the boundary.

3.2.3 Straight channel

Planes were positioned at various distances downstream of the inlet. The Nusselt number at each of these planes was calculated. If convergence did not occur, the model length was increased and the run repeated. This procedure was repeated until the Nusselt number value converged.

The channel lengths had a direct effect on the number of mesh cells and thus also the simulation time. For this reason it was not possible to make the channels too long in order to obtain definitive convergence. A realistic definition of convergence was needed, and it was determined that convergence would be assumed when changes in Nusselt number values were below 0.1%

The detail about the investigations done on the straight channels has been summarized in Table 3.2. The investigation was done at $Re = 200$ for all the cross-section shapes using water as the working fluid.

Table 3.2 - The investigations done for straight channels with circular, semi-circular and triangular cross-sections.

Cross-section	Channel length	Hydraulic diameter	Working fluid	Reynolds number	Flow velocity
Circular	800 mm	3 mm	Water	200	0.0594 m/s
Semi-circular	450 mm	1.222 mm	Water	200	0.146 m/s
Triangular	450 mm	1.1547 mm	Water	200	0.1544 m/s

3.2.4 Serpentine channel

The cross-sections under investigation had semi-circular and equilateral triangular shapes.

All the results for the investigation were obtained from the second unit of the flow channel (refer to figure 2.2), to ensure that the entrance and exit effects did not influence the results. The thermal performance for the uniform wall surface temperature boundary condition (T) was investigated by varying the radius ratio $\frac{R_c}{d}$ and the Reynolds number. An enhancement factor was used to present the results, showing an expected gain in heat transfer performance. It was an effective way to compare the differences in performance for varying conditions and geometric parameters.

For the serpentine path, there were two geometric ratios of interest, $\frac{R_c}{d}$ and $\frac{L}{d}$ and their values were set as follows:

- The half wavelength (L) was constant and fixed at 9 mm throughout the investigation.
- The ratio $\frac{L}{d}$ was set equal to 4.5, and gave the diameter (d) a value of 2 mm.
- Values for the radius ratio $\frac{R_c}{d}$ were varied incrementally between 1 and 1.8 using increments of 0.2

The flow conditions were set within the following limits:

- Laminar flow with Reynolds numbers varying between 50 and 400 using finer increment sizes for Re where improved resolution was desired.
- Water was used as the modeling fluid ($Pr = 6.13$).

Results were presented in the form of a parametric table, generated to summarize the thermal performance of the variations in geometric ratios and Reynolds numbers. Graphs clearly demonstrated the variations in the enhancement factors between the different conditions.

The effects of various parameters are discussed in the following subsections.

3.2.4.1 Reynolds numbers

Firstly, the effect from the variation in Reynolds number was investigated by maintaining all other influencing parameters at constant values and simulating flow for different bulk flow velocities (v_{ave}) corresponding to that particular Reynolds numbers. The radius ratio $\frac{R_c}{d}$ remained fixed at a value of one.

Table 3.3 - Flow velocities and Reynolds numbers for the semi-circular serpentine channel.

Semi-circular velocity (v_{ave}) (m/s)	Reynolds number (Re)
0.036447312	50
0.072894625	100
0.109341937	150
0.14578925	200
0.218683875	300
0.2915785	400

Table 3.4 - Flow velocities and Reynolds numbers for the triangular serpentine channel.

Triangular velocity (v_{ave}) (m/s)	Reynolds number (Re)
0.038598084	50
0.077196168	100
0.115794252	150
0.154392336	200
0.231588504	300
0.308784672	400

The Reynolds numbers with their corresponding flow velocities are tabulated for the semi-circular cross-section in Table 3.3 and for the triangular cross-section in Table 3.4.

The Nusselt number values were calculated at 20 different planes during each simulation using the equations defined earlier (eq. (6) and (7)), and used to calculate the Nusselt number values for the straight channels.

The Nusselt numbers obtained from the 20 planes were then averaged to find the overall channel Nusselt number. From the work of Rosaguti et al. (2006) on semi-circular serpentine channels, for flow characterized by $Re = 200$, the Nusselt number should be approximately equal to $Nu_{avg} \approx 2.8$.

For each bend region (shown as the black and white regions in figure 3.7), 5 planes were used to calculate the local Nusselt number values. Figure 3.7 gives a visual interpretation of the plane locations. The locations of the planes for each bend region were as follows:

- Inlet of bend (1, 6, 11 and 16)
- Centre of bend (2, 7, 12 and 17)
- Exit of bend (3, 8, 13 and 18)
- 1 diameter length downstream of each bend (4, 9, 14 and 19)
- Centre of straight sections (5, 10, 15 and 20)

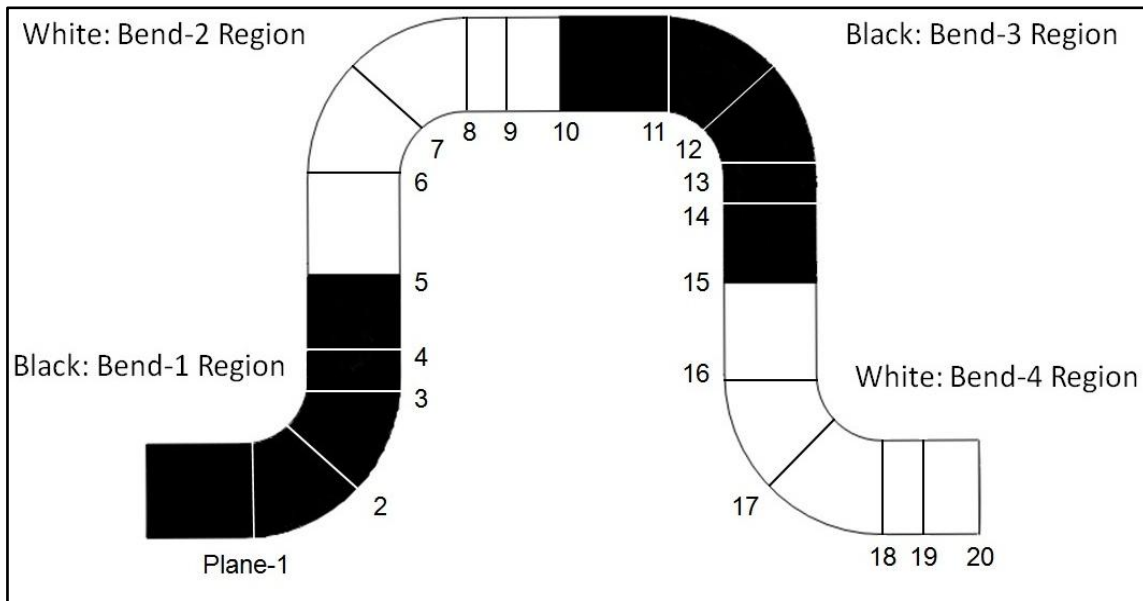


Figure 3.7 - Visual representation of the plane locations at which the Nusselt numbers were calculated for the Reynolds number investigations.

3.2.4.2 Radius ratio, $\frac{R_c}{d}$

The same procedure was followed to investigate the effect of the radius ratio. The Reynolds number was set to a value of 200 and the radius ratio simulated for $1.0 \leq \frac{R_c}{d} \leq 1.8$ having incremental values of 0.2. For $\frac{R_c}{d} > 1.8$, the shape of the unit transforms into a bell-like shape (shown in figure 3.8), characterised by different flow phenomena since the straight sections disappear and the layout is not classified as serpentine anymore.

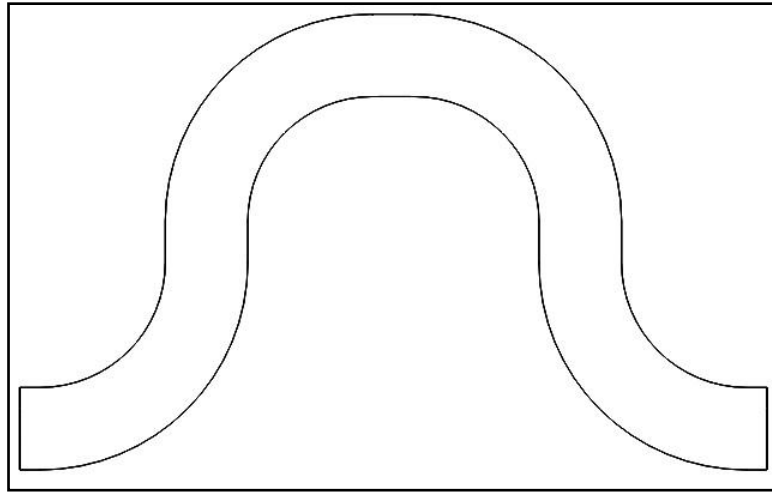


Figure 3.8 - Schematic shows a channel where $\frac{R_c}{d} = 2$. It is clear that the straight sections have almost disappeared, forming a bell-like shape. The flow exits a bend flowing directly into the next bend.

Inserting the same number of above-mentioned planes into such a geometry to calculate Nusselt number values produced inconsistent results compared to the more clearly defined serpentine shaped channels.

Regarding the radius ratio investigation, the Nusselt number values were thus calculated and averaged at 16 positions along the path. Note that the number of planes used for the radius ratio investigation differed from the number of planes used for the Reynolds number investigation.

The 4 planes positioned one diameter length downstream of each bend (in figure 3.7 they are numbered as 4, 9, 14 and 19) were removed because when the bend radius became too large, the plane's position moved into the next bends' region, which means that the calculated Nusselt number at these plane positions were representative of the results for the next bend region (shown as the black and white regions of figure 3.9).

The calculation planes used for the radius ratio investigation were located as shown in figure 3.9. The locations of the planes for each bend region were:

- Inlet of bend (1, 5, 9 and 13)
- Centre of bend (2, 6, 10 and 14)
- Exit of bend (3, 7, 11 and 15)
- Centre of straight sections (4, 8, 12 and 16)

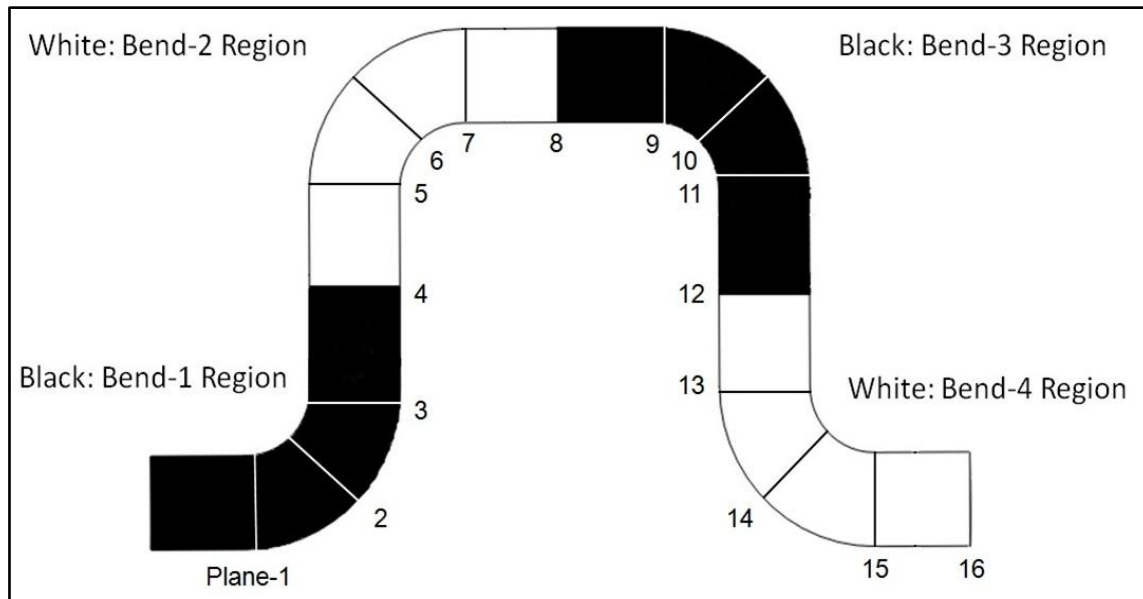


Figure 3.9 - Visual interpretation of the plane locations at which the Nusselt numbers were calculated for the radius ratio investigations.

3.3 Chapter summary

The numerical investigation was divided into two parts: first the straight sections having circular, semi-circular and triangular cross-sections were modeled to the point where the theoretical Nusselt number value was obtained. This confirms the accuracy of STAR-CCM+ and for simulating micro-channel flow conditions. The second part consisted of simulating the serpentine channel with semi-circular and triangular cross-section shapes.

A methodology for the Nusselt number calculations was given including some important equations. The Nusselt number equation (eq. (6)) was defined in STAR-CCM+ as a user field function since the application of this equation accurately predicted the Nusselt number obtained in a previous study by Du Toit (2003). Details of exactly how the equation was implemented in STAR-CCM+ was discussed in section 3.1.4.

A basic summary was given of how the CFD code works. Some equations were listed, including the Navier-Stokes equations and the more conservative form of the general equations which were simplified to obtain the so-called general transport equation. The transport equation was used to solve for properties in the CFD code. The SIMPLE algorithm was stated to show how the solution process of a CFD code works. Examples were shown of meshes and the different cell types were discussed.

The meshing process in STAR-CCM+ was discussed in some detail. The most important meshing parameters were listed and individually discussed. The most influential parameter in the current investigation was the overall prism layer thickness. This parameter determined the size of cells in the outer boundary region of the flow channel. Since the thermal and hydraulic boundary layers formed on the boundary surface it was important to have sufficient resolution of cells in the region to clearly define the velocity and temperature profiles.

In the scope section, the assumptions and boundary conditions were discussed for the two-part investigation. For both parts, the uniform wall surface temperature (T) condition was applied. The wall surface was held at 350 K and the fluid entered the channel at 300 K. Further details were included, such as the channel lengths, flow velocities and channel hydraulic diameters.

Values were given for geometric parameters such as the half wavelength and channel diameter. The range, for which the Reynolds number and radius ratio were investigated, was also defined in the scope section.

The investigation on the Reynolds numbers and the radius ratio was discussed and the positions inside the serpentine channel where cross-sectional planes were located were shown and discussed. The planes were located where the Nusselt numbers were calculated and averaged for each bend region. An overall average Nusselt number was also calculated to represent the Nusselt number of the complete channel.

Chapter 4 – Results and Discussion

The chapter gives the results from the study and a discussion thereof. Firstly the straight channel is discussed as far as the meshes used and the convergence of the Nusselt numbers for various cross-sections. Thereafter follows a section about the meshes and Nusselt number convergence of the serpentine channel simulations. The final section illustrates and discusses the flow phenomena found during this investigation in more detail.

4.1 Straight channels

In this section the circular simulations is discussed. It starts with the meshes used for accurate simulations and the next section describes results from the circular cross-section. The following section describes results from the semi-circular cross-section and the second last section describes the results from the triangular cross-section. In the final section a conclusion is given to summarize the straight channels section.

4.1.1 Numerical mesh

The surface extruder and poly mesher combination of Star-CCM+ yielded the most accurate results in modeling the straight sections. The surface extruder used far less cells when simulating internal flow than the pure polyhedral mesher did. Meshes created using the surface extruder were also found to converge quicker.

One reason was that once the velocity and temperature profiles are fully developed, the flow does not change much over short distances in the axial direction of flow and one can use more elongated cells to define the fully developed regions of flow. A short discussion follows.

In figure 4.1 and figure 4.2, the velocity and temperature profiles are shown respectively.

It becomes clear from figure 4.1 that as flow entered, the velocity profile started to form on the boundary and it kept growing toward the channel centre as the fluid flowed through the length of the channel up to a point where it was fully developed and no more change was noticed.

In figure 4.2, the same happened with the temperature profile as for the velocity profile, but it developed at a slower rate.

Unlike the velocity profile where the velocity magnitude for each layer remained constant along the length, the temperature changed throughout the length of the channel since heating to the fluid occurred. The change is not noticeable in figure 4.2, since the change occurred slowly along the length of the channel.

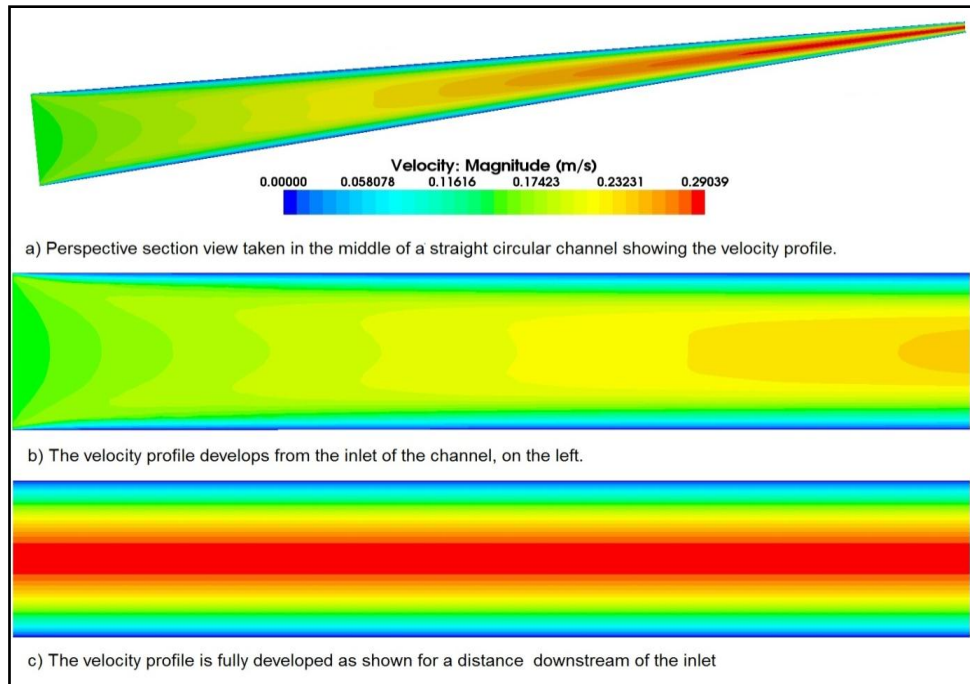


Figure 4.1 - Schematic shows the velocity boundary layer development for internal flow. a) is a perspective section view that shows how the velocity boundary layers develop from the inlet of the circular straight channel. b) shows how the velocity boundary layers develop from the inlet and c) shows the fully developed velocity profile a distance downstream of the inlet.

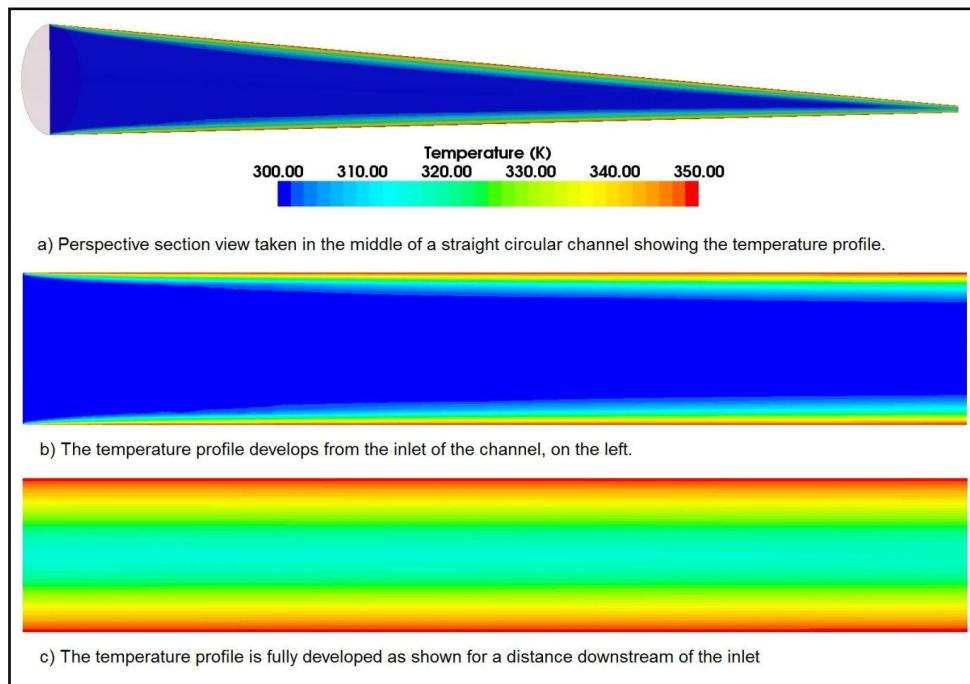


Figure 4.2 - Schematic shows the temperature boundary layer development for internal flow. a) is a perspective section view that shows how the temperature boundary layers develop from the inlet of the circular straight channel. b) shows how the temperature boundary layers develop from the inlet and c) shows the fully developed temperature profile a distance downstream of the inlet.

Heat was transferred from the surface to the fluid layers adjacent to the boundary and then towards the layers in the centre of the channel.

Models for the straight channels utilized the mesh extruder function since these channels were required to be much longer than the serpentine channels, and using the polyhedral cells, lead to problems with insufficient computer memory.

One example was where the polyhedral mesher divided the channel into more than 4 million cells, while the surface extruder divided the same geometry into little more than one million cells.

4.1.2 Circular straight channel

Two slightly different mesh models having similar meshing parameter settings, converged close to the theoretical value of $Nu_x = 3.657$. Details regarding the meshes can be found in Appendix A.1.1.

The results from the more refined mesh (Mesh 1), with almost double the number of cells than in the other mesh (Mesh 2), differed by only 0.22%, thus mesh independence could be assumed.

The most accurate result yielded a Nusselt number value equal to 3.632, which was within 0.7% of the established value for a circular cross-section (given in figure 2.1). For the circular cross-section, the Nusselt numbers required more than double the channel length to converge compared to the other cross-sections. This might be because of the larger hydraulic diameter and slower development of boundary layers.

Figure 4.3 and figure 4.4 shows the results for the three different cross-section shapes for which the investigation was done. The graph in figure 4.3 is comparable to the schematic from Shah & London (1978), shown earlier in figure 3.1.

As expected the Nusselt number value within the entry region was very large compared to the fully developed flow value, but it steadily decreased throughout the length of the channel until convergence occurred at a dimensionless axial distance value of approximately $x^* = 0.1$.

The data points for the circular cross-section that was used to construct figure 4.3 and figure 4.4 are available in Appendix A.1.1 for the straight sections.

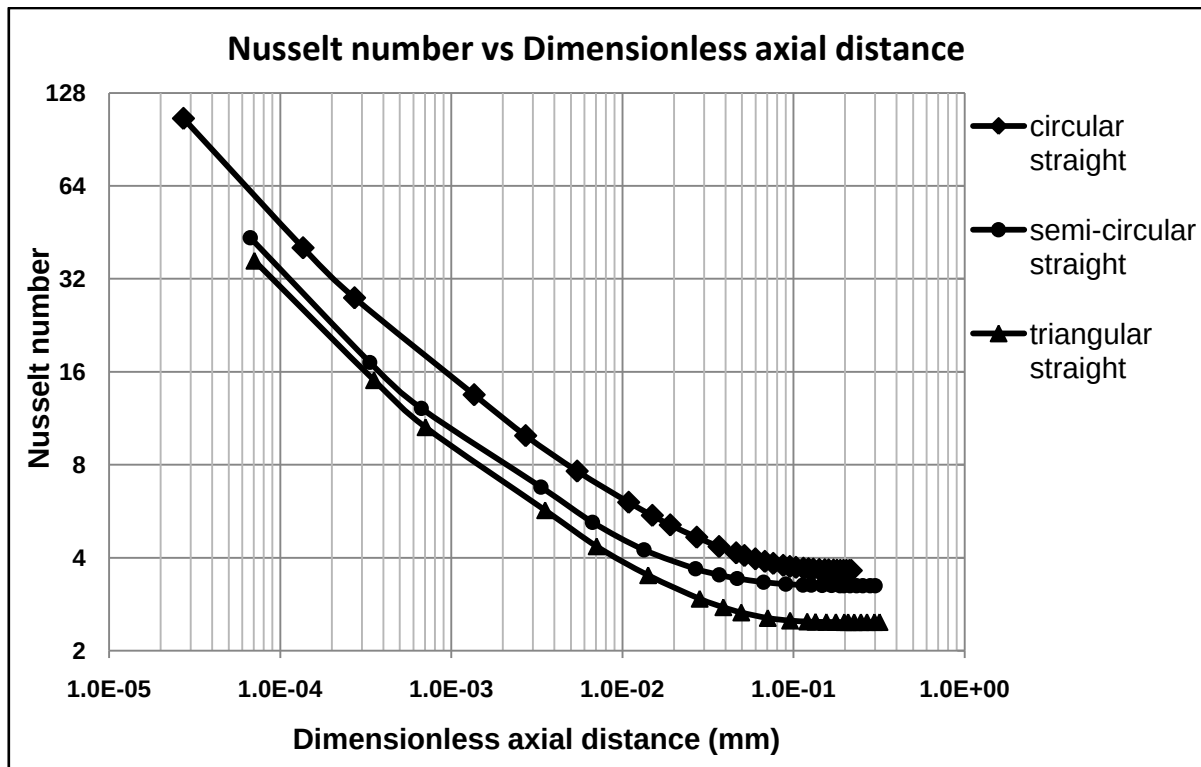


Figure 4.3 - Schematic showing the obtained Nusselt number versus the dimensionless axial distance on a logarithmic scale.

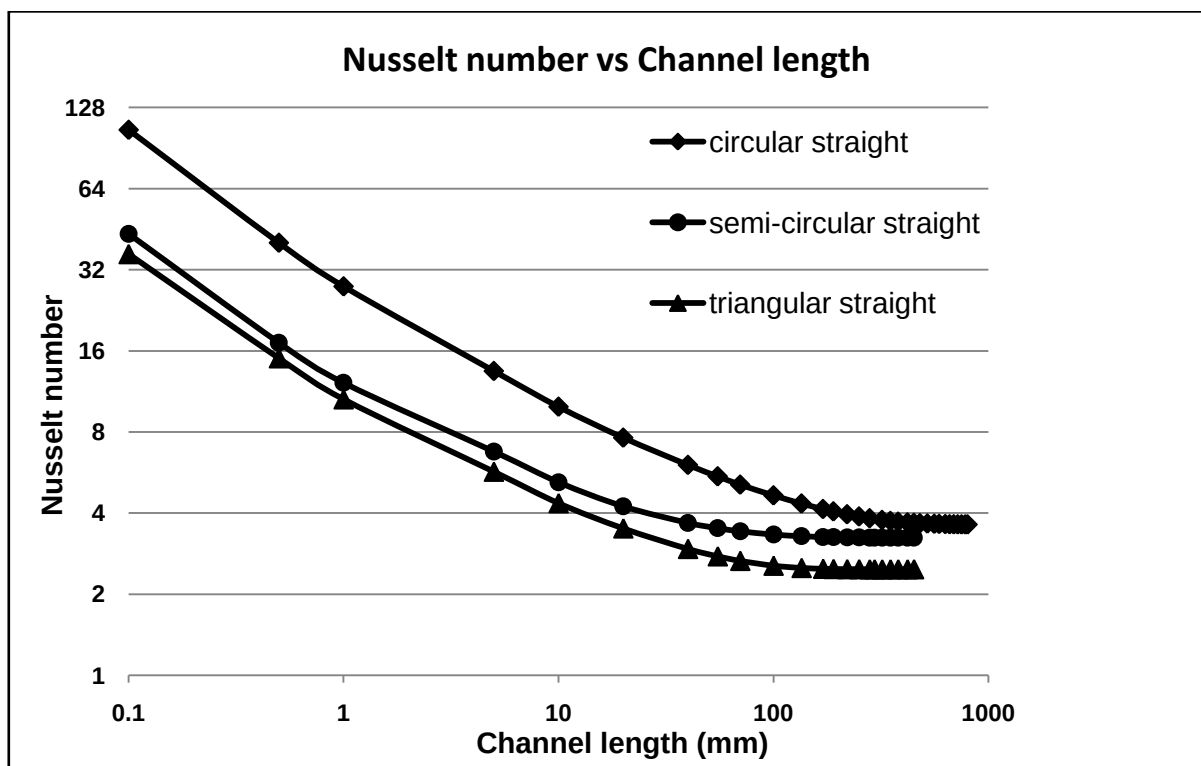


Figure 4.4 - Schematic showing the convergence of the Nusselt number versus the distance from the inlet on a logarithmic scale.

4.1.3 Semi-circular straight channel

The same methodology was applied to the semi-circular cross-section. Meshing parameters were the same as for the circular models except for the overall prism layer (PL) thickness. When reducing the polyhedral cell size in the case of the more refined mesh (Mesh2), the prism layer thickness should also be reduced in order for the size of the prism layer cells to stay uniform.

The two mesh models, set at different cell sizes and PL thickness values, seemed to converge close to the theoretical Nusselt number value for a semi-circular cross-section, again indicating mesh independence. The results obtained for the two meshes, differed by about 1.2%. Details regarding the meshes can be found in Appendix A.1.2.

The most accurate result was a Nusselt number value equal to 3.285, which is within 1.2% of the empirical value of $Nu_x = 3.326$. The data points for the semi-circular cross-section that was used to construct figure 4.3 and figure 4.4 are available in Appendix A.1.2 for the straight sections.

4.1.4 Triangular straight channel

For the triangular cross-section the PL thickness also had to be reduced to assure a uniform thickness of the cells in the PL region. Further, the same polyhedral cell sizes were used as for the two semi-circular meshes. Two meshes again were used, the one mesh being more refined than the other. Details regarding the most accurate mesh can be found in Appendix A.1.3.

The Nusselt number values for the triangular shape converged at the shortest axial distance of all the cross-sections. This can be explained by the triangular shape's small hydraulic diameter and resultant smaller volume flow with lower momentum and thermal inertia and faster boundary development.

The most accurate mesh exhibited convergence at a Nusselt number value equal to 2.476, which is within 0.54% of the empirical value of $Nu_x = 2.49$ for an equilateral triangle. The data points for the triangular cross-section that was used to construct figure 4.3 and figure 4.4 are available in Appendix A.1.3 for the straight sections.

4.1.5 Conclusion

The investigation of the straight channels, having matched the empirical values for the Nusselt number for three different cross-sectional shapes, validated the methodology using the equations from Du Toit (2003). The same methodology could therefore be applied to the investigation of the serpentine channels.

4.2 Serpentine channel

Similar to the previous straight channel section, the first section gives a description of the meshes used for simulations. The next section states and discusses the results for the Reynolds number investigation for the two cross-section shapes and in the second final section the radius ratio investigation results are stated and discussed. Finally a conclusion section gives a short synopsis of the results and discussions.

4.2.1 Numerical mesh

4.2.1.1 Semi-circular serpentine channel

The polyhedral mesher in combination with the prism layer mesher was used for the simulations conducted on the serpentine channels. Polyhedral cells provided more optimized unstructured cell volumes for complex geometries. Flow was by far more swirly in the serpentine channels than in the straight channels and the polyhedral cells provided a more accurate description of the flow in the bend regions at lower cell counts.

The mesh parameter settings used in simulating the straight channels were also used to mesh the serpentine channel. A semi-circular serpentine channel model (figure 2.2) was set up and refined until grid independence was obtained. The changes in total average Nusselt numbers for the middle unit of the domain were compared between meshes having different cell sizes. After every run a meshing parameter was refined until the effect from the particular parameter on the result was no longer visible. As previously mentioned some parameters proved to be more influential than others.

Table 4.1 - Specific sets of meshing parameters for serpentine simulations testing grid independence.

Model number	1	2	3	4
Base size	0.01 mm	0.005 mm	<u>0.004 mm</u>	0.003 mm
Number of layers	20	20	<u>20</u>	20
Stretching factor	1	1	<u>1</u>	1
Prism layer thickness	0.0004 mm	0.0002 mm	<u>0.0002 mm</u>	0.0002 mm
Cell target size	5% of base	5% of base	<u>5% of base</u>	5% of base
Cell minimum size	5% of base	5% of base	<u>5% of base</u>	5% of base
Nusselt number	9.654539	9.333523	<u>9.332624</u>	9.335753

In Table 4.1 a summary is given of Nusselt numbers obtained by varying the base size. One notices that the Nusselt numbers obtained from model numbers 2, 3 and 4 were closely matched, which was not the case for number 1.

This is because model number 1 has a much larger base size than the other three. Remember that the base size determines the size of the polyhedral cells. One notices that the prism layer thickness is also larger.

This was done simply to keep the prism layer region uniform as discussed in Chapter 3 and shown in figure 3.6.

The Nusselt number values from model number 2, 3 and 4, converged on a value of 9.33. The conclusion was made that these meshes have reached grid independence.

Model number 4 had the most cells corresponding to over 600 000 cells, making simulating a large number of models, a time consuming process. For this reason, the second most refined simulation (number 3 in Table 4.1) was used for simulating the serpentine channels having semi-circular and triangular cross-sections. The mesh corresponded to approximately 391 000 cells and exhibited acceptable convergence for the Nusselt number.

The Nusselt number value at which the different mesh models converged ($Nu = 9.33$), corresponded to an enhancement factor value of $e_{Nu} = 2.8$. This value matched the enhancement factor value obtained by Rosaguti et al. (2006), which was in the order of $e_{Nu} \approx 2.8$. This was for the same radius ratio and flow conditions corresponding to $Re = 200$ and uniform wall surface temperature boundary condition (T).

4.2.1.2 Triangular serpentine channel

The geometric dimensions for the equilateral triangle were very similar to that of the semi-circular cross-section. The same prism layer thickness also provided a uniform cell-size distribution in the cross-section region using the same cell size as discussed above. For this reason, grid independence was tacitly assumed for the triangular shaped models of the same mesh settings as for the semi-circular cross-section.

4.2.2 Reynolds number

A parametric table is shown in Table 4.2 to give a summary of the Reynolds number results.

Table 4.2 - Summary of the results obtained from investigating the Reynolds number.

Reynolds number	Average heat transfer enhancement factor (e_{Nu})	
	Semi-circular	Triangular
50	1.341	1.694
100	1.767	2.366
150	2.306	2.966
200	2.792	3.463
300	3.508	4.243
400	4.054	4.857

The results for the investigation of the effect of the variation of the Reynolds number are shown in figure 4.5 for the semi-circular- and the triangular cross-sections.

The results for the semi-circular cross-section agreed very well with the results obtained by Rosaguti et al. (2006).

There appeared to be little difference in the trends between the results obtained for the triangular- and semi-circular cross-sections, except that the overall enhancement factors at all values of the Reynolds numbers were higher for the triangular cross-section. This meant that the heat transfer enhancement due to using serpentine channels seemed to be more beneficial when using the equilateral triangular geometry.

However, this does not mean that the triangular cross-section exhibited the best overall heat transfer behaviour; it only showed the triangular shape had the best enhancement of heat transfer compared to the straight section of same shape. At a Reynolds number value of 400, the Nusselt number obtained for the triangular cross-section was $Nu = 12.093$, while the semi-circular cross-section value obtained was $Nu = 13.485$.

Calculating the convective heat transfer coefficient (h) by using equation 1 for the two cases, gave $h = 7.3188 \frac{\text{kJ}}{\text{m}^2\cdot\text{K}}$ for the semi-circular case and $h = 6.202 \frac{\text{kJ}}{\text{m}^2\cdot\text{K}}$, for the triangular cross-section, showing that convective heat transfer for the semi-circular cross-section serpentine channel was better.

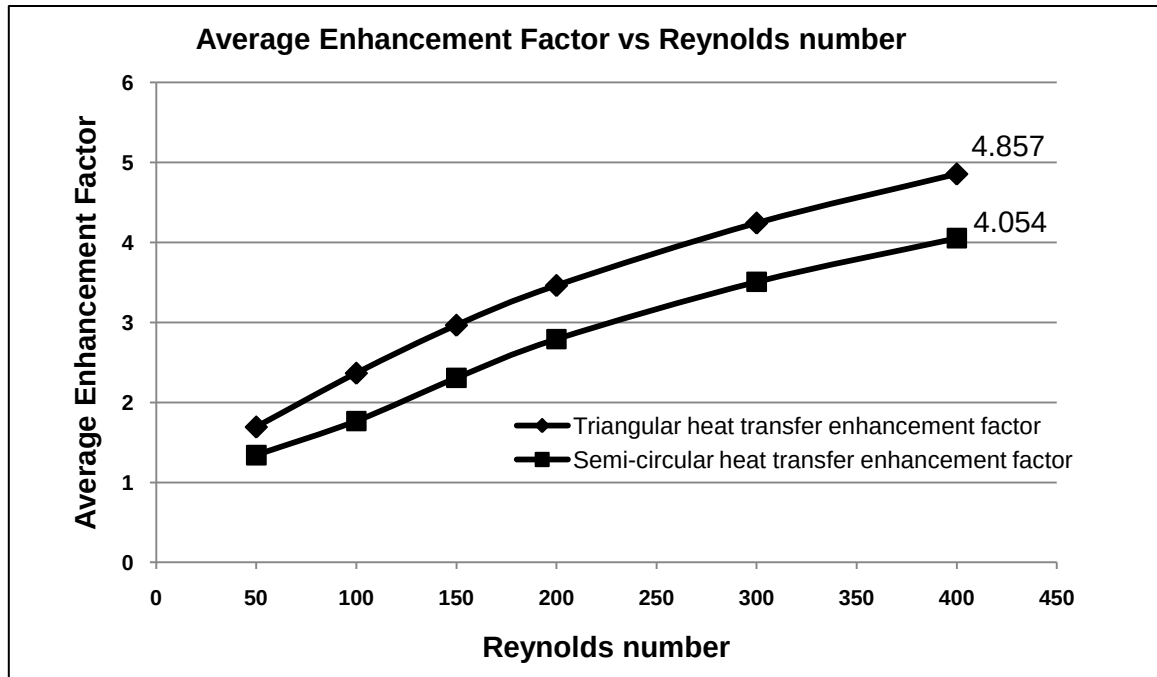


Figure 4.5 - Graph of heat transfer enhancement factor (e_{Nu}) versus Reynolds number for semi-circular and triangular serpentine channels using $\frac{R_c}{d} = 1$ and $\frac{L}{d} = 4.5$ for uniform wall surface temperature boundary condition (T).

Examining figure 4.6, the enhancement factors peaked at the exit of all the bends and then rapidly decreased in the middle sections until flow entered the next bend, and the process repeated itself in the next bend region.

The enhancements in bend-2 and bend-4 were a little lower than in bend-1 and bend-3. In section 4.3, the flow phenomena are discussed and a possible explanation is given to the above mentioned variations in heat transfer enhancement. To obtain a better understanding of the results shown in figure 4.6 and figure 4.7, the reader is referred to the schematic shown in figure 3.8 which shows all the plane positions where the Nusselt number were calculated.

Figure 4.8 and figure 4.9 show the velocity profiles within the central unit of the serpentine model for both cross-sections.

By inspecting the velocity contours, one sees that the high velocity regions before and after bend-2 and bend-4 are on opposite sides of the cross-section. This is due to the inertial force being the most dominant force within the bend region.

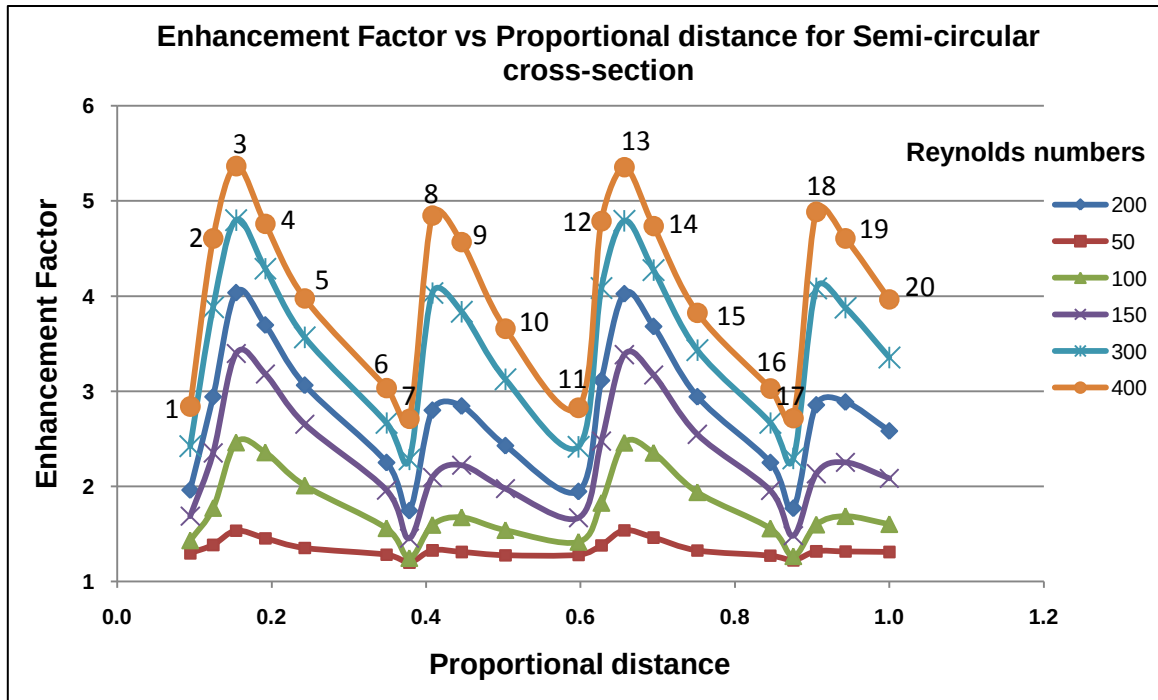


Figure 4.6 - Schematic showing the enhancement factor (e_{Nu}) for the proportional distances in the semi-circular serpentine channel for different values of Reynolds number. Applies to $\frac{R_c}{d} = 1$ and $\frac{L}{d} = 4.5$ for uniform wall surface temperature boundary condition (T).

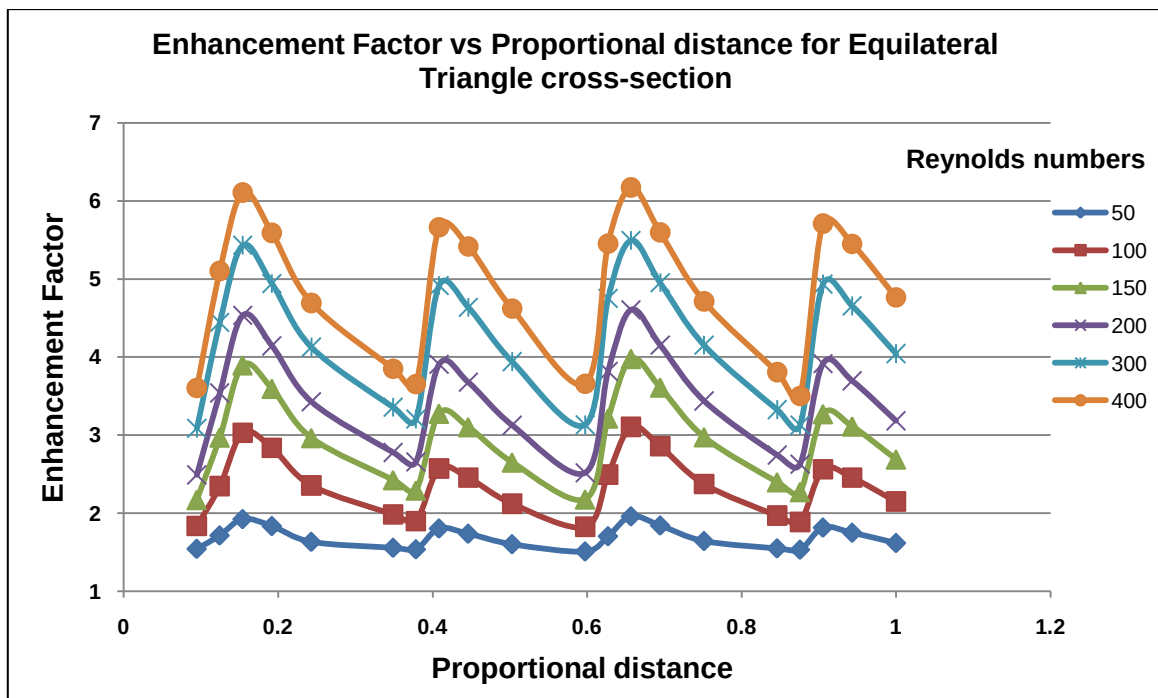


Figure 4.7 - Schematic showing the enhancement factor (e_{Nu}) versus the proportional distances in the triangular serpentine channel for different values of Reynolds number. Applies to $\frac{R_c}{d} = 1$ and $\frac{L}{d} = 4.5$ for uniform wall surface temperature boundary condition (T).

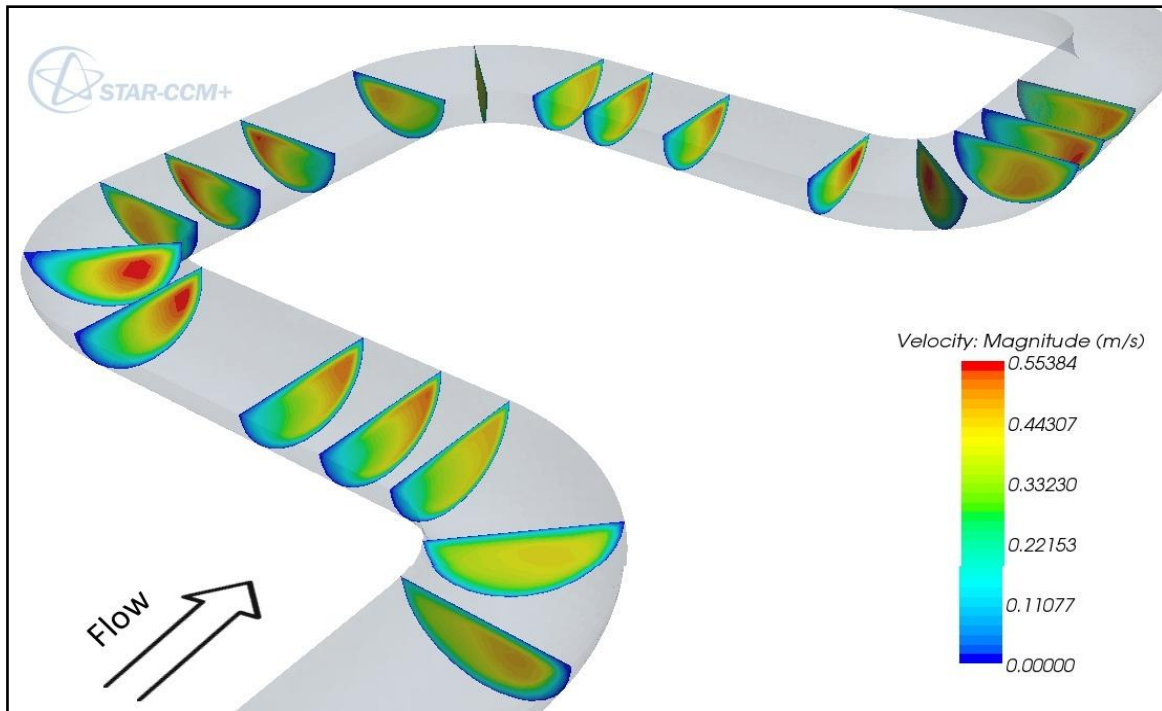


Figure 4.8 - Cross-sectional velocity profiles at 20 plane locations inside a semi-circular serpentine channel, indicating high velocity regions within the flow domain. $Re = 400$, $\frac{R_c}{d} = 1$ and $\frac{L}{d} = 4.5$ for uniform wall surface temperature boundary condition (T).

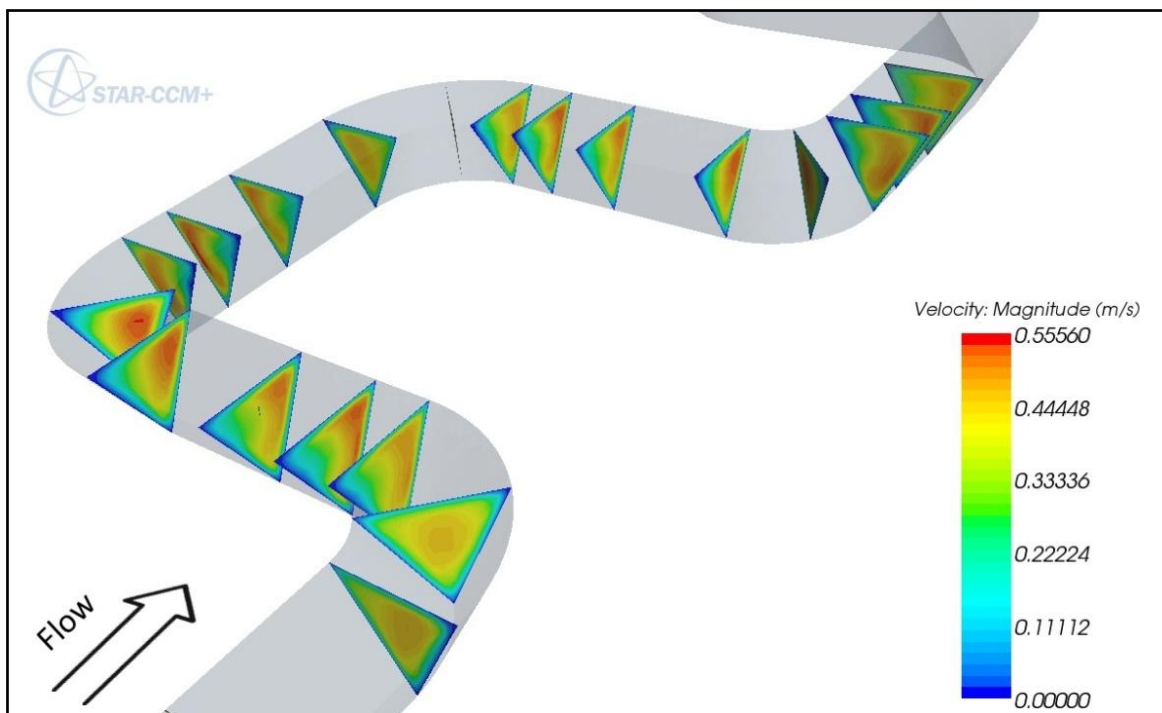


Figure 4.9 - Cross-sectional velocity profiles at 20 plane locations inside a triangular serpentine channel, indicating high velocity regions within the flow domain. $Re = 400$, $\frac{R_c}{d} = 1$ and $\frac{L}{d} = 4.5$ for uniform wall surface temperature boundary condition (T).

4.2.3 Radius ratio, $\frac{R_c}{d}$

Results for the study investigating the effect of the variation of the radius ratio have shown no significant variation on the thermal performance for either of the cross-sections. Figure 4.10 shows a slight variation in the enhancement factor, yet too small to be significant compared to the enhancement due to the variation in Reynolds number.

A parametric table is shown in Table 4.3 to give a summary of the radius ratio results.

Table 4.3 - Summary of the results obtained from investigating the radius ratio.

Radius ratio	Average heat transfer enhancement factor (e_{Nu})	
	Semi-circular	Triangular
1.0	2.792	3.341
1.2	2.695	3.123
1.4	2.652	3.181
1.6	2.620	3.149
1.8	2.654	3.176

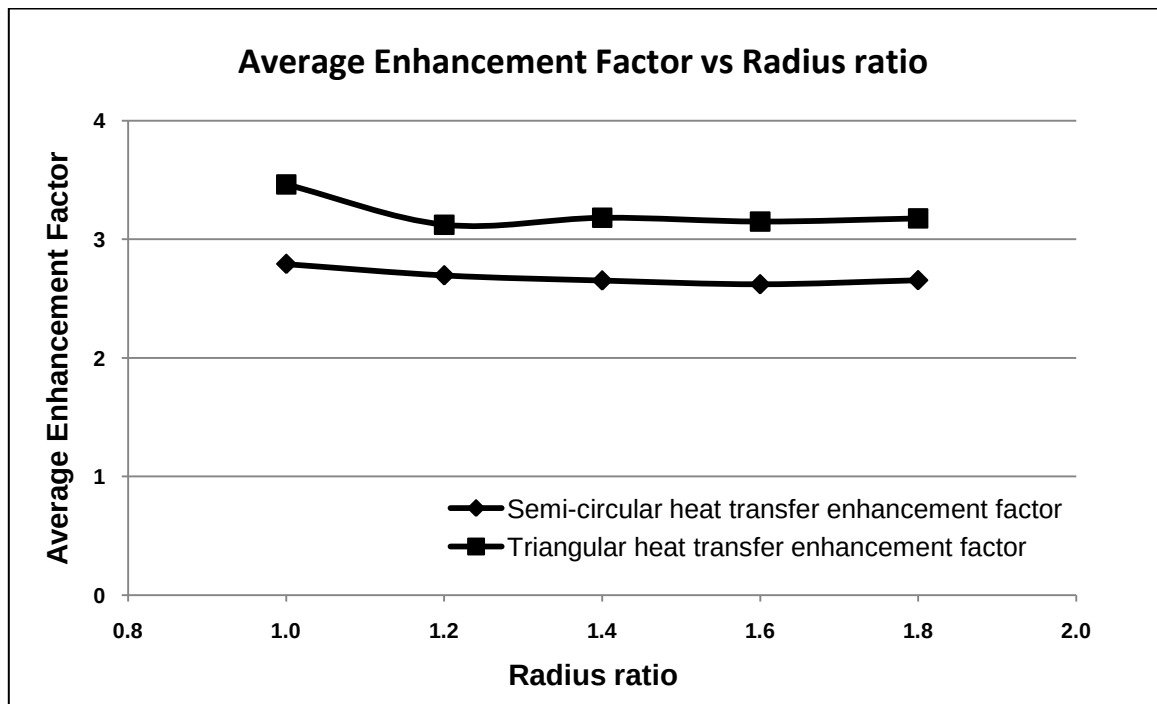


Figure 4.10 – Graph of enhancement factor (e_{Nu}) versus the radius ratio for semi-circular and triangular serpentine channels. $\frac{R_c}{d}$ for $Re = 200$ and $\frac{L}{d} = 4.5$ for uniform wall surface temperature boundary condition (T).

From figure 4.10, a slight decrease in enhancement factor was noted for both cross-sections. As the radius ratio increased for a constant diameter, the radius of curvature of the bend increased. A logical interpretation would be that the fluid momentum was not shifted as rapidly as in the case of a small radius resulting in less effective mixing.

Figure 4.11 and figure 4.12 show the enhancement factors obtained for the two cross-sections. The planes positioned at the exit of each bend (3, 7, 11 and 15), showed peak values in the enhancement factor for their particular bend regions. The peaks at plane 7 and 15 corresponded to bend-2 and bend-4. They were lower peak values when compared to bend-1 and bend-3.

One may notice from looking at figure 4.11 that as the radius ratio increased, the peaks became flatter up to the point where for the radius ratio $\frac{R_c}{d} = 1.8$, the peak became more of a camel-hump looking shape.

The reason for this was when the radius ratio increased, the change in direction through the bend was not as sudden, but more gradual. It meant the fluid was not forced to mix as vigorously as for a small radius geometry. The same effect was noticed when looking at the investigation for the triangular cross-section shown in figure 4.12.

To obtain a better understanding of the results shown in figure 4.11 and figure 4.12, also refer to the schematic shown in figure 3.9 which shows all the plane positions where the Nusselt numbers were calculated.

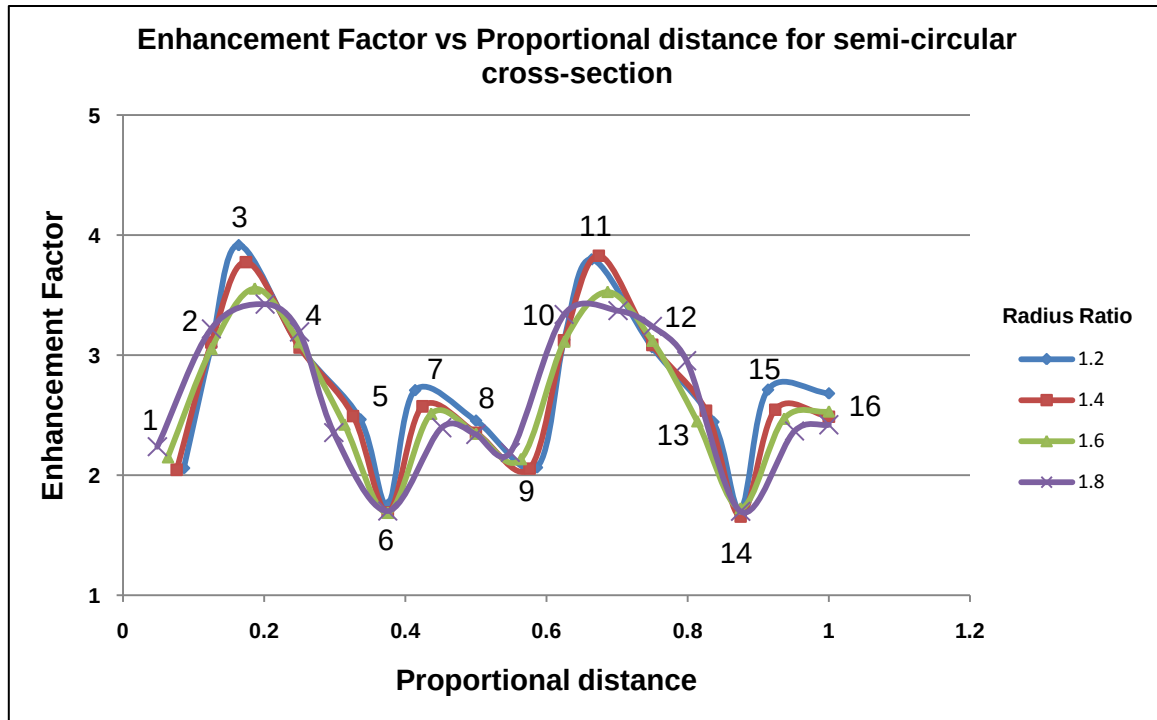


Figure 4.11 – The enhancement Factor (e_{Nu}) versus proportional distance for a semi-circular serpentine channel. $Re = 200$, $\frac{L}{d} = 4.5$ for uniform wall surface temperature boundary condition (T).

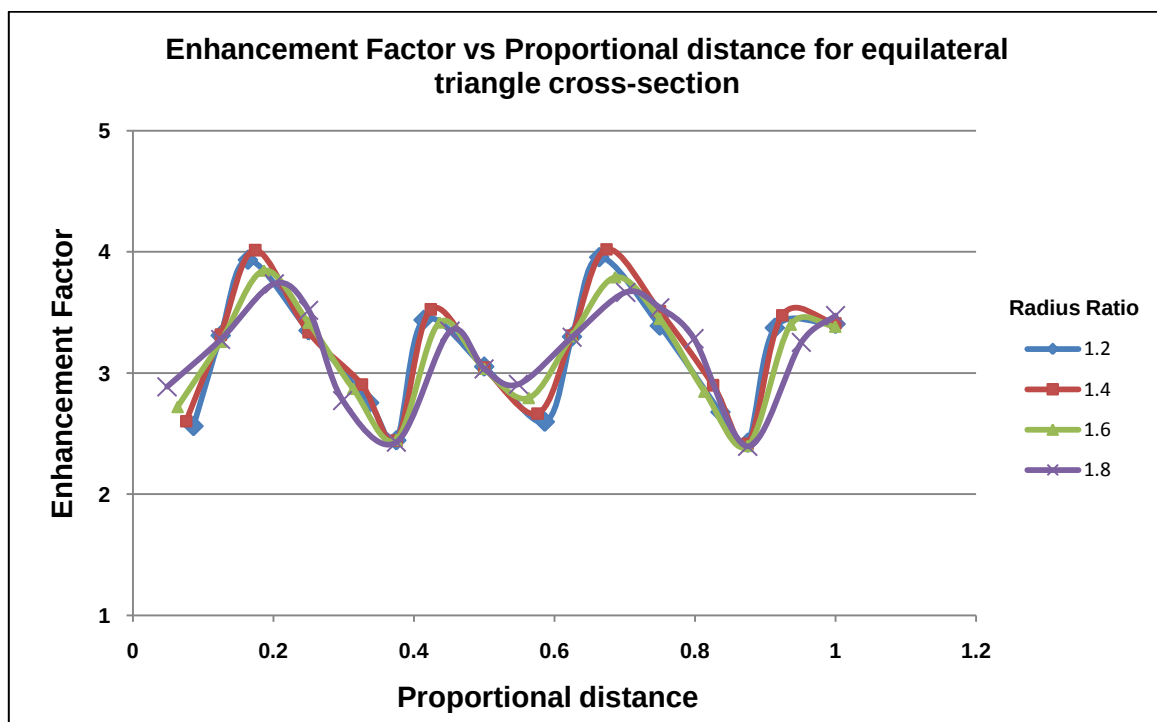


Figure 4.12 – The enhancement Factor (e_{Nu}) versus the proportional distance for equilateral triangular serpentine channel. $Re = 190$, $\frac{L}{d} = 4.5$ for uniform wall surface temperature boundary condition (T).

4.2.4 Conclusion

From the investigations the most influential factor was the Reynolds number. A clear increase was noted when the Reynolds number was increased. This corresponded to more effective mixing induced by the bends due to the more influential inertial effects in the flow. The radius ratio investigations did not show any significant enhancement in heat transfer. This is due to weaker inertial effects in the bend regions corresponding to a larger bend radius of curvature.

4.3 Flow characteristics

In this sections the flow behaviour from the simulations are given and discussed. Firstly some of the results from the Rosaguti et al. (2006) simulations were compared to the simulations from the current investigation.

Figure 4.13 represent the velocity profiles for the semi-circular serpentine cross-section investigated by Rosaguti et al. (2006). The contours are for $Re = 200$. Comparing it to the velocity contours obtained from the current investigation at $Re = 200$, it was clear that the fluid did not have the same momentum transfer through the bends. The velocity contours from the current study is shown in figure 4.14.

Comparing the velocity contours from the two simulations showed some differences in the shapes of the velocity contours. Both simulations were performed at exactly the same flow conditions, $Re = 200$, and radius ratio, $\frac{R_c}{d} = 1$.

From the profiles shown in the four schematics of figure 4.13, the study of Rosaguti et al. (2006) concluded that the enhanced heat transfer behaviour of the fluid flowing in the tortuous channels was primarily due to vortices induced by the bends.

The current study did not show the same contour shapes for the high velocity regions in the corners of the post bend regions. In figure 4.13 the velocity contours shows the fluid was forced into the corners while in figure 4.14 the fluid only seemed to be shifted from one side of the channel to the other, not nearly as vigorously as in figure 4.13.

Another comparison to the Rosaguti et al. (2006) study was done. The velocity contours at $Re = 200$ of the Rosaguti et al. (2006) study, was compared to $Re = 400$ of the current investigation.

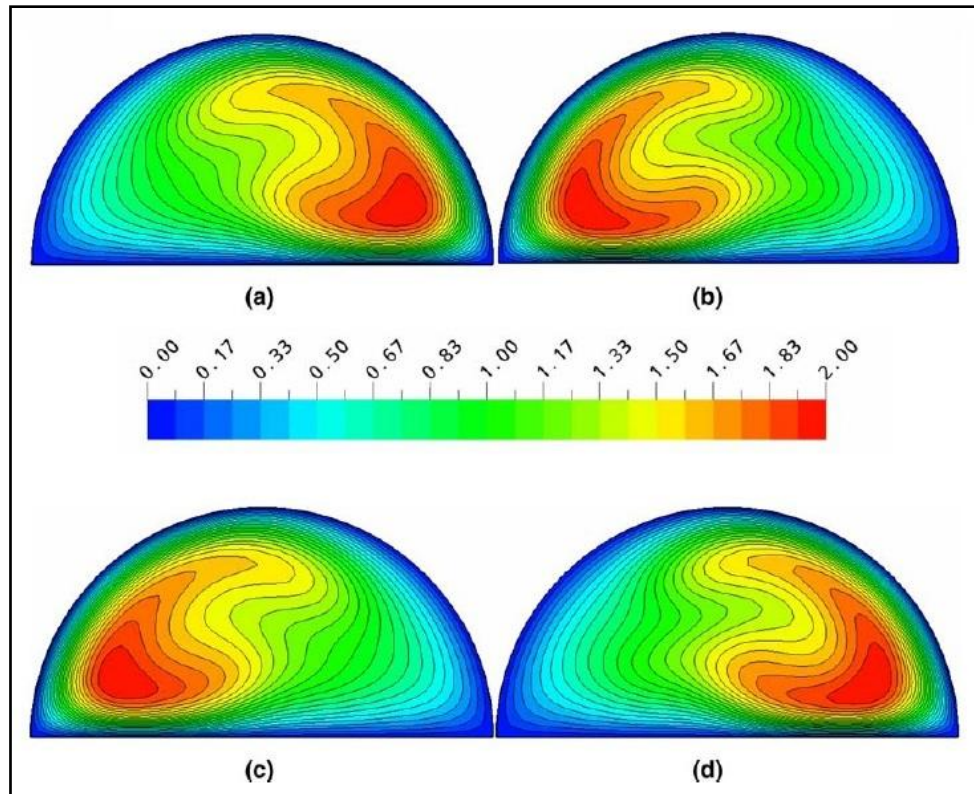


Figure 4.13 - Velocity contours from the Rosaguti et al. (2006) investigation at locations of 1 diameter length downstream of the bends. The scale shown is $\frac{v}{v_m}$, where v_m is the mean velocity of a serpentine channel. Conditions are $\frac{L}{d} = 4.5$, $\frac{R_c}{d} = 1$ and $Re = 200$. (a) is 1d downstream of bend-1, (b) is 1d downstream of bend-2, (c) is 1d downstream of bend-3, (d) 1d downstream of bend-4. Modified from Rosaguti et al. (2006).

Comparing the velocity contours at $Re = 400$ of the current investigation, shown in figure 4.15, to the Rosaguti et al. (2006) velocity contours shown in figure 4.13, it is clear that the contours for these two cases corresponded more closely as for $Re = 200$ of the current investigation.

In figure 4.15 the fluid seems to have been forced into the corners of the channel in a similar way to the Rosaguti et al. (2006) study. This behaviour assumed to prove the existence of Dean vortices in the post-bend regions.

To investigate the presence of vortices, streamlines were inserted into the bulk flow domain of current models. However, no significant vortices were found to be present in the flow that may have caused any major heat transfer enhancement. Figure 4.16 shows the streamlines inserted into the channel in the post bend-1 region.

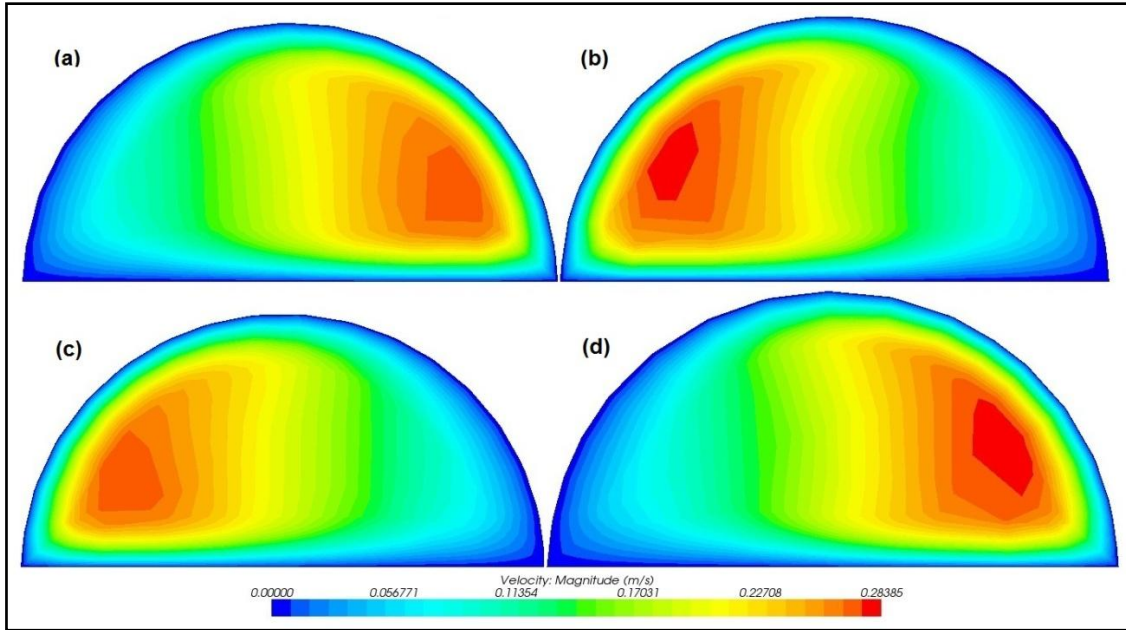


Figure 4.14 - Velocity contours from the current investigation at the location of 1 diameter length downstream of the bends. Conditions are $\frac{L}{d} = 4.5$, $\frac{R_c}{d} = 1$ and $Re = 200$. (a) is 1d downstream of bend-1, (b) is 1d downstream of bend-2, (c) is 1d downstream of bend-3, (d) 1d downstream of bend-4.

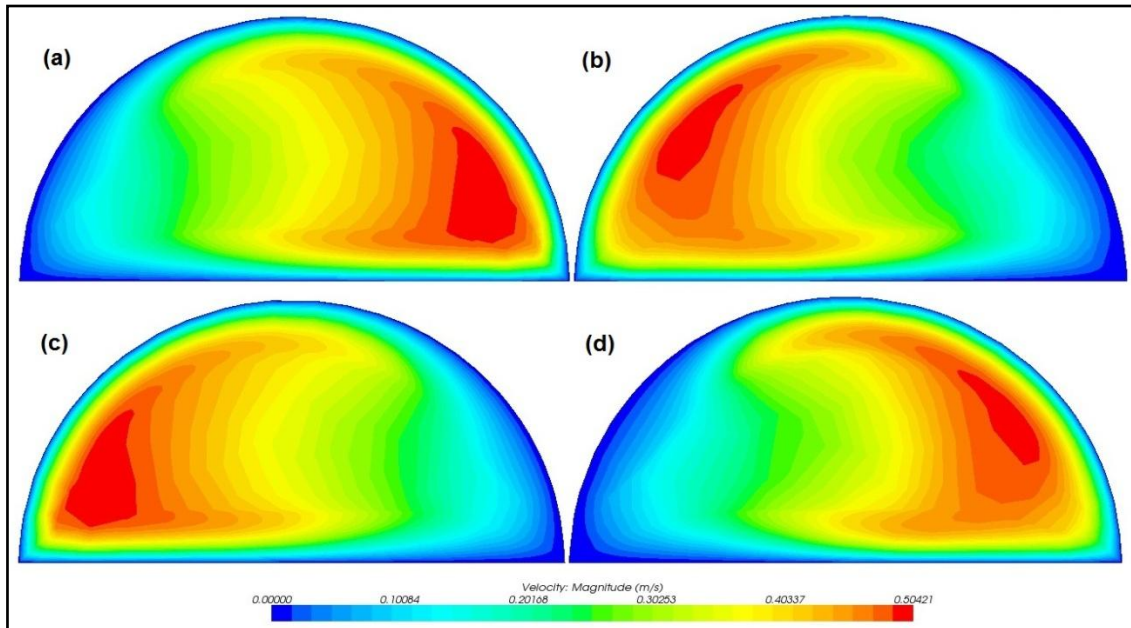


Figure 4.15 - Velocity contours from the current investigation at the location of 1 diameter length downstream of the bends. Conditions are $\frac{L}{d} = 4.5$, $\frac{R_c}{d} = 1$ and $Re = 400$. (a) is 1d downstream of bend-1, (b) is 1d downstream of bend-2, (c) is 1d downstream of bend-3, (d) 1d downstream of bend-4.

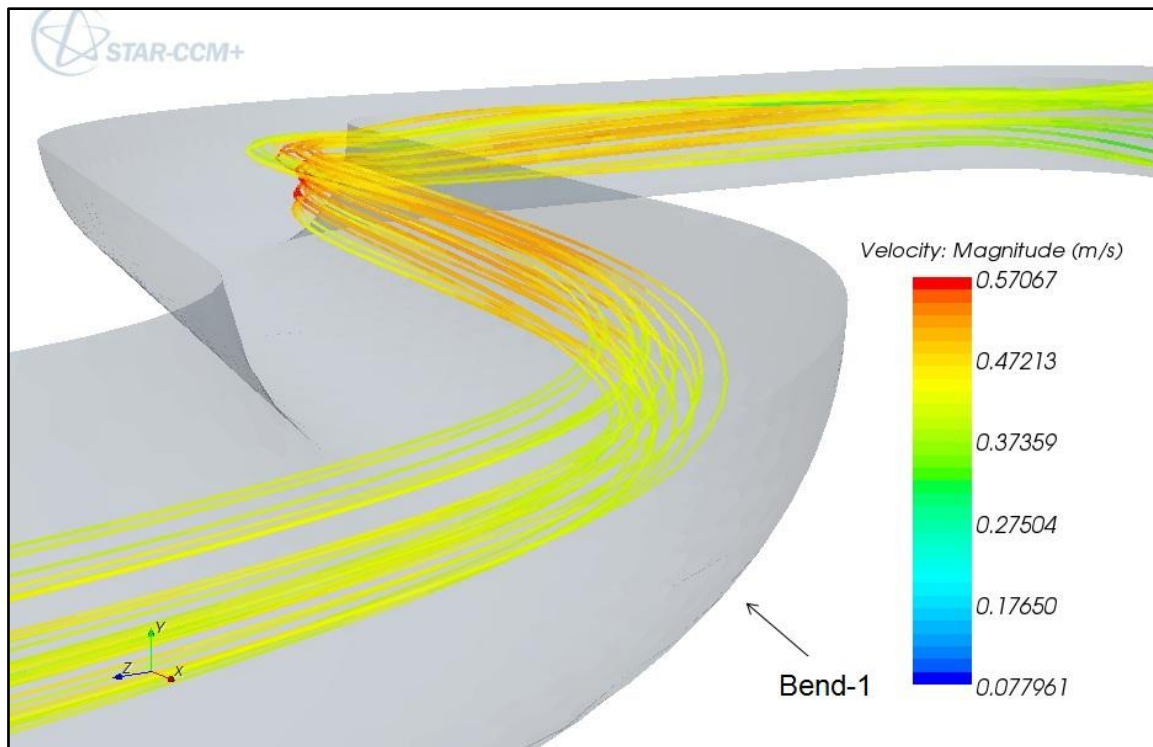


Figure 4.16 - Schematic shows the streamlines for the semi-circular serpentine channel with flow of $Re = 400$. The schematic shows the post bend-1 region.

The streamlines in figure 4.16 showed no vortices in the bulk fluid flowing in the channel. Filling up the whole diameter of the channel with streamlines produced the same results. Similarly the streamlines following bends- 2, 3 and 4 is shown in figure 4.17, 4.18 and 4.19.

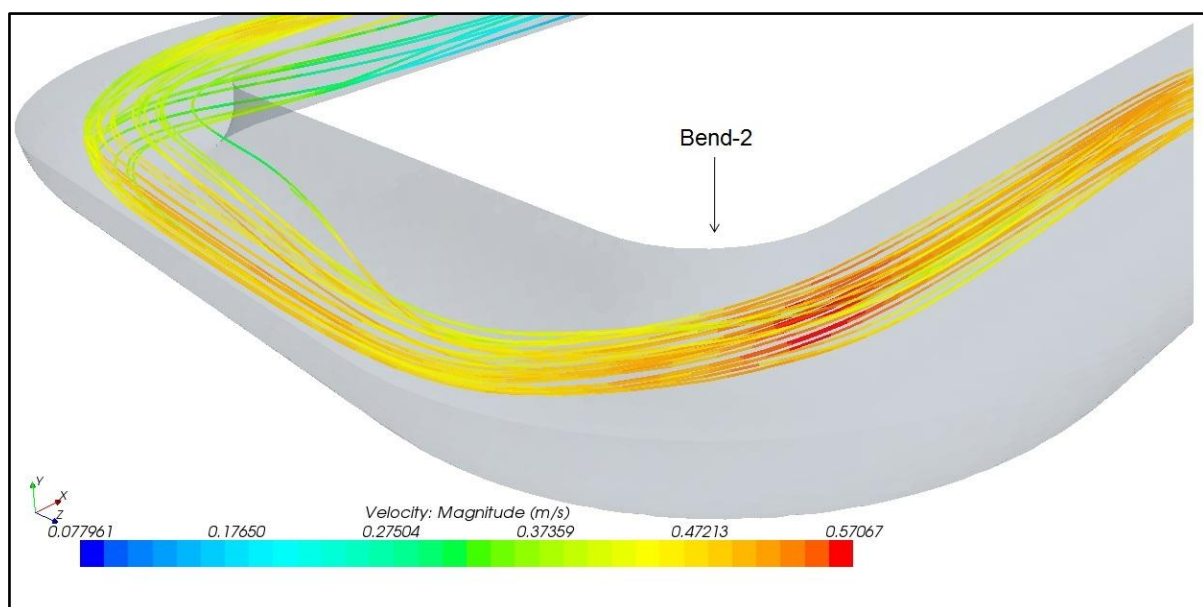


Figure 4.17 - Schematic shows the streamlines for the semi-circular serpentine channel with flow of $Re = 400$. The schematic shows the post bend-2 region.

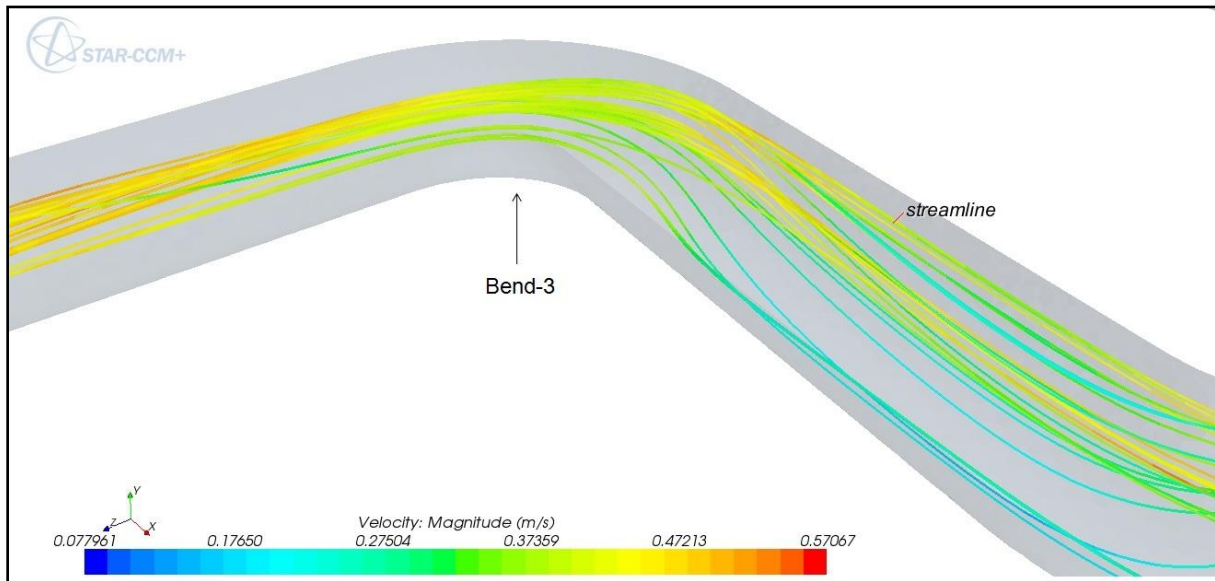


Figure 4.18 - Schematic shows the streamlines for the semi-circular serpentine channel with flow of $Re = 400$. The schematic shows the post bend-3 region.

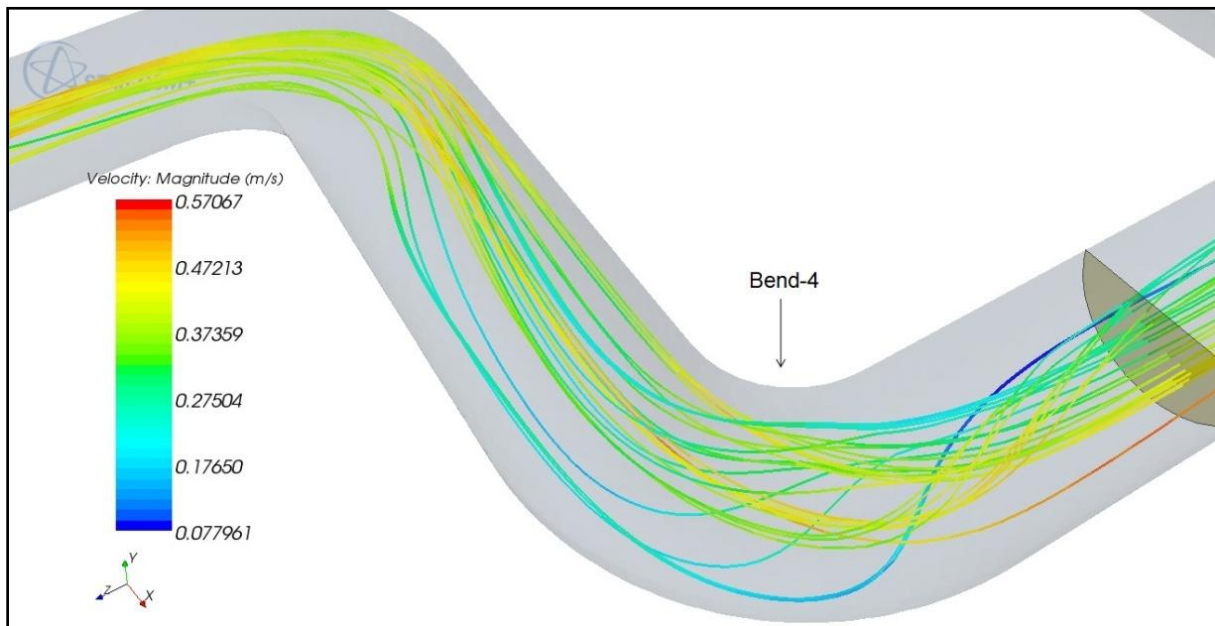


Figure 4.19 - Schematic shows the streamlines for the semi-circular serpentine channel with flow of $Re = 400$. The schematic shows the post bend-4 region.

Again by looking at figure 4.17 and figure 4.18, the streamlines showed no sign of vortices in the bend regions of bend-2 and bend-3.

As a further example the velocity vector function was enabled for the streamlines. Figure 4.20 shows the results from the simulation.

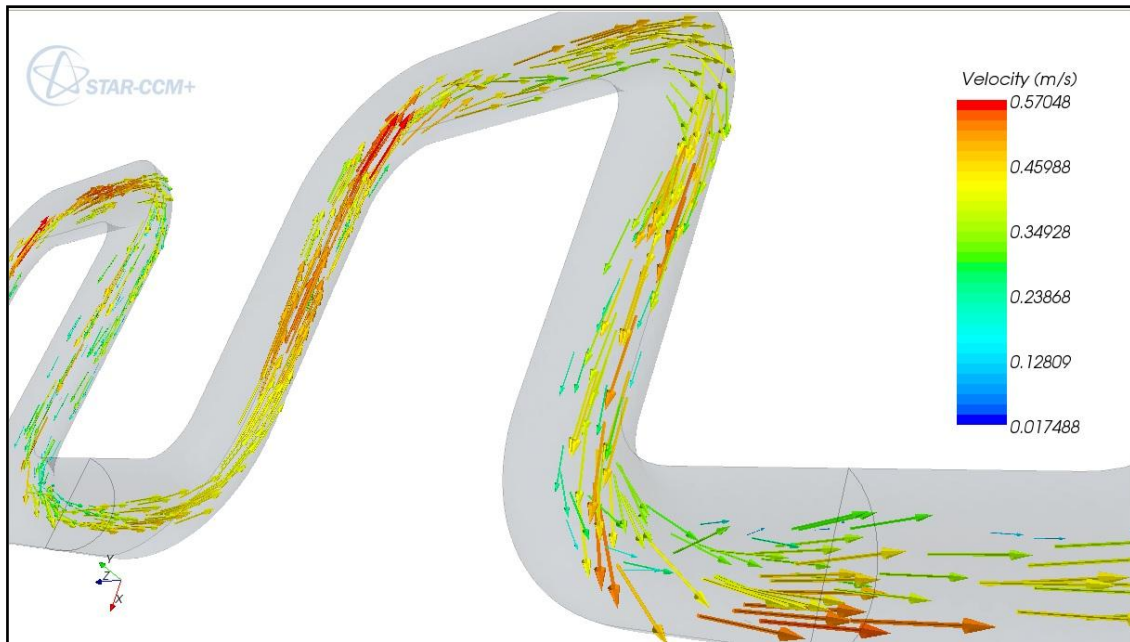


Figure 4.20 - Schematic shows the streamlines as velocity vectors, indicating the flow direction. The magnitude is represented by the colour.

From the schematic it is clear that the velocity vector arrows show no sign of vortices in the bulk flow region. For vortices one may expect the velocity vector arrows to point in a rotational direction about an axis parallel to the flow direction. This clearly was not the case.

Looking at figure 4.19 presenting the post bend-4 region, it was noticed that some of the streamlines showed signs of vortices. On closer inspection it became clear that a small vortex formed above the bulk fluid flowing in the centre of the channel. The widths of the vortices were small compared to the bulk of the fluid which smoothly flowed through the bend. It looked like the bulk fluid collided against the bend wall and then struggled to settle in the straight section following the bend. The flow behaviour could rather more clearly be described by mixing than vortex formation.

To explain the flow phenomena it was important to note that there were two regions with different flow phenomena.

The first was the bulk fluid region called the primary flow region, where the majority of the mass flow was observed. In this region the flow seemed to only pass from one bend to the next without incurring any rotational movement or vortices. It only seemed to follow the shape of the channel, thus no flow separation was visible. The flow patterns that was seen in figures 4.16 to 4.19 showed examples of the primary flow that mixed in the post bend regions.

The other region contained small vortex like rotational movement in the top-middle regions of the semi-circular channel. The vortex movement was referred to as secondary flow and were noticed as flow exited the bends. It was observed more prominently in the exit regions of bend-2 and bend-4 as seen from the channel top view in figure 4.21.

As example to explain the small vortices, a schematic is presented in figure 4.21 which shows the flow flipping over the centre of the channel. This behaviour was mostly noticeable in the regions exiting the bends.

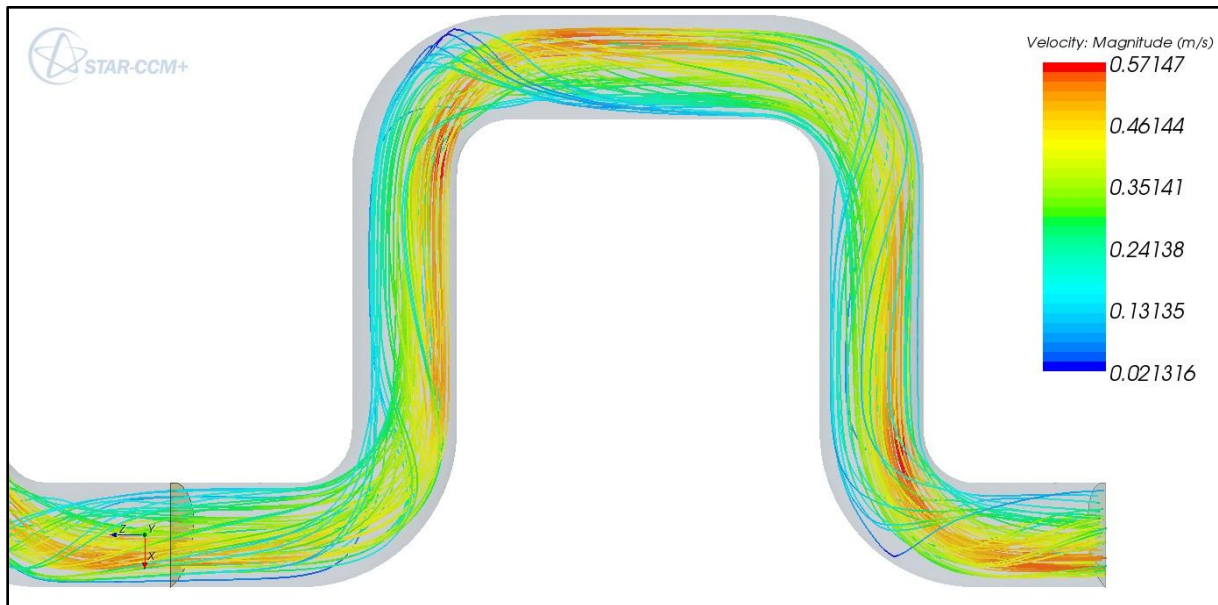


Figure 4.21 - Top view of a semi-circular serpentine channel. The streamlines show radial movement in the post bend regions.

As flow entered a bend, it seemed that the fluid adjacent to the side walls of the channel was flipped over to the other side of the channel as soon as it was forced from its position by the primary flow.

In a circular cross-section the primary flow might have followed the circular shape of the channel roof, and rolled over to the other side of the channel. In a semi-circular and triangular cross-section, the same was not possible. The roof of the semi-circular- and triangular shape was flat, and prevented the bulk flow to rotate, rather forcing the secondary flow into the corner and to flip over to the other side of the channel.

Figure 4.22 shows a schematic of a vortex as formed in the post bend-4 region. The anti-clockwise vortex was located above the primary flow which did not have any rotation. A similar distinct vortex was visible in the post bend-2 region.

These minor vortices should not be confused with Dean's vortices, where the bulk flow actually forms the vortex movement.

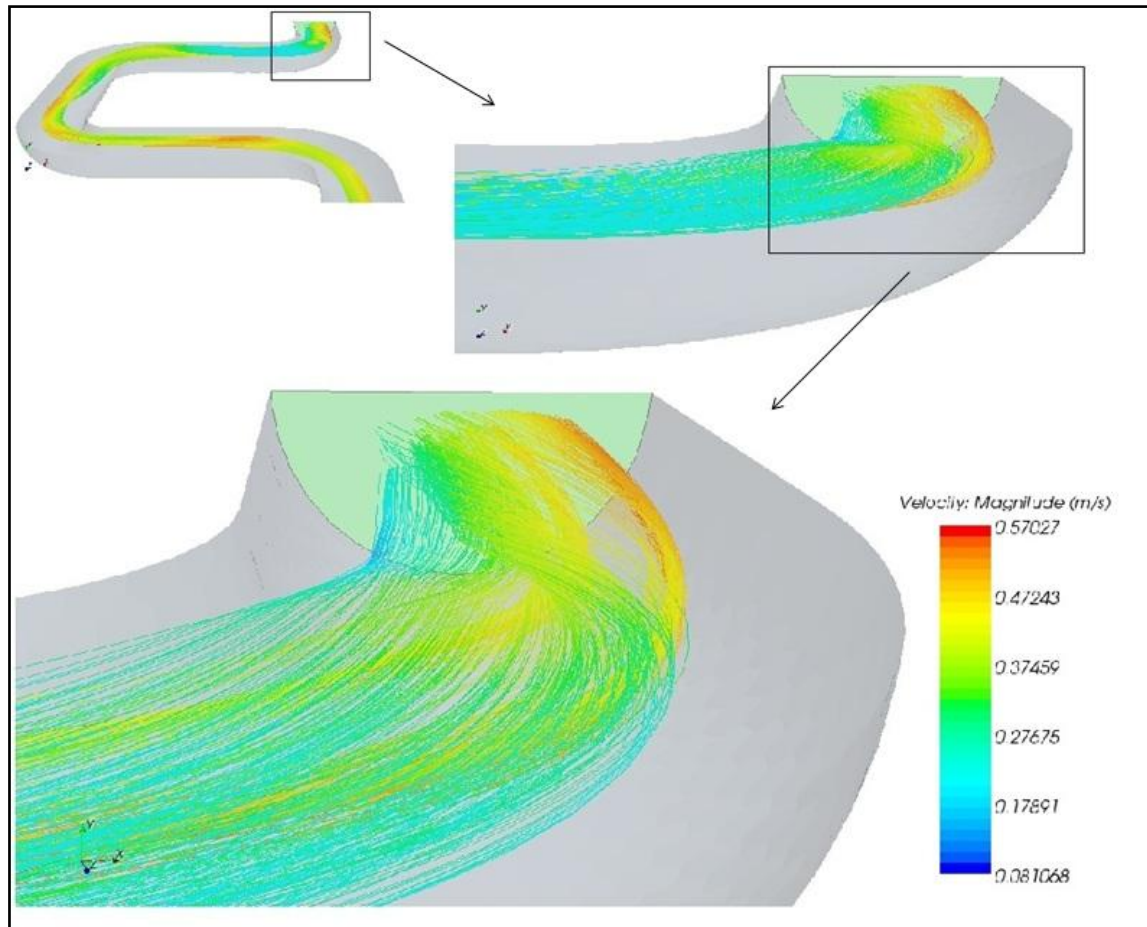


Figure 4.22 - Schematic shows a vortex adjacent to the roof of bend-4 in a serpentine channel with semi-circular cross-section.

Almost the same behaviour was noticed for the triangular geometry. The rotational flow movement is shown in figure 4.23 and figure 4.24 for the triangular cross-section.

The presence of minor vortices did not seem to be a significant cause for enhancement in heat transfer since the vortices were present in the bend regions (bend-2 and bend-4) but with lower heat transfer enhancement compared to bend-1 and bend-3. The strength of the vortices did not seem to be very strong, and the majority of the enhancement was more likely to come from the random mixing induced by the flow shifting in the bends.

The bends ensured that the thermal boundary formed on the wall surface (figure 4.2) was disturbed frequently for effective mixing. The spikes in heat transfer in the regions of bend-1 and bend-3 (shown in figure 4.6 and figure 4.7) were due to the effective mixing in combination with the small vortices forming in the post bend region of bend-2 and bend-4.

The minor vortices forming after bend-2 and bend-4 might have helped to amplify the mixing taking place in the next bends (bend-1 and bend-3). These vortices could have caused higher inertial effects in the regions of bend-1 and bend-3, and by increasing the inertial forces in the region, mixing was enhanced.

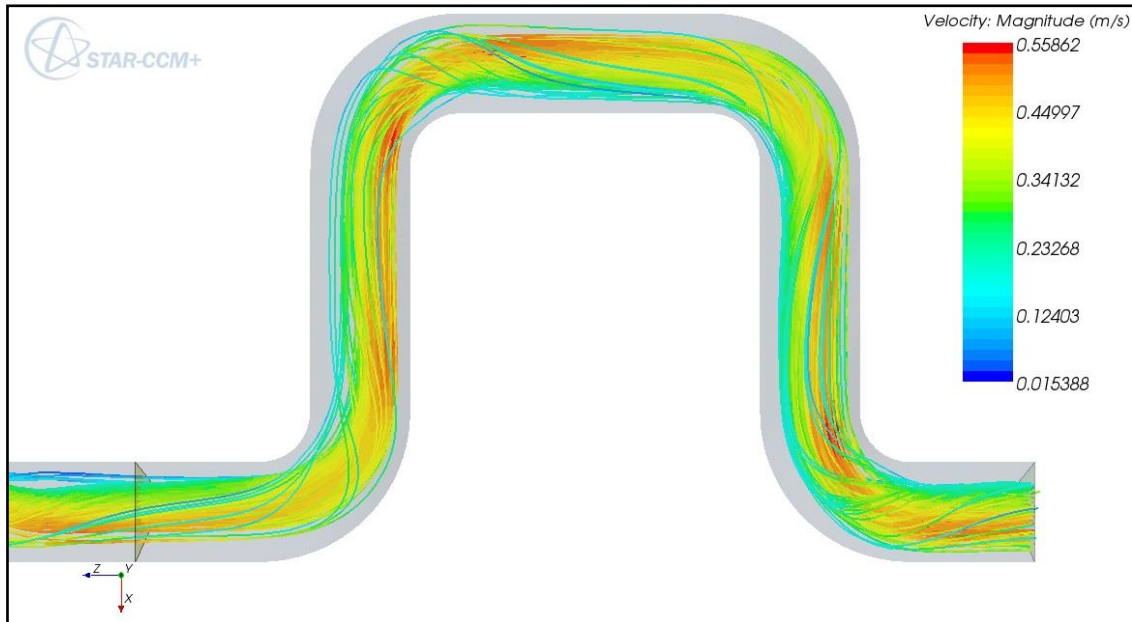


Figure 4.23 - Top view of a triangular serpentine channel. The streamlines show rotational movement in the post bend regions.

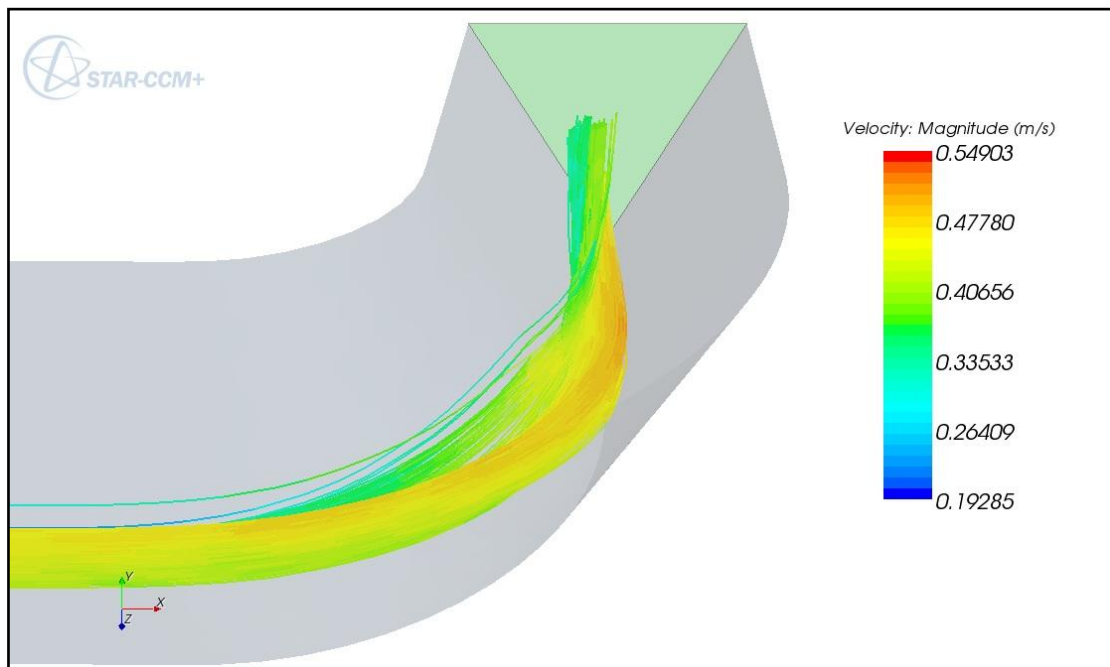


Figure 4.24 - Schematic shows a small vortex adjacent to the roof of bend-4 in a serpentine channel with triangular cross-section.

4.4 Chapter summary

The straight channel simulation results correlated within acceptable tolerances to the published values of the Nusselt number for the circular, semi-circular and triangular cross-section. This provided verification of STAR-CCM+ and the general methodology, in order to continue with the serpentine channel investigation.

The Reynolds number proved to be a dominant factor for heat transfer enhancement in serpentine channels. This is evident by looking at figure 4.5. As the flow sped up, the inertial effects started to dominate the flow, resulting in more effective fluid mixing.

The radius ratio investigation did not have the same dominant effect on heat transfer as the Reynolds number (refer to figure 4.10) A slight decrease was noted as the radius ratio was increased, this was due to the inertial effects becoming less influential.

The flow phenomena was investigated using the streamlines feature of STAR-CCM+. The streamlines confirmed the presence of some vortices in two of the channel bends, yet no Dean vortices were observed as the literature suggested. Flow rather seemed defined by the primary flow passing through the bends with inertial effects causing good mixing and the presence of vortices in bend-2 and bend-4. Mixing thus seemed to be the most dominant flow feature responsible for heat transfer enhancement.

Chapter 5 – Conclusions

The heat transfer performance of serpentine channels with semi-circular- and triangular cross-sections was investigated using CFD. The simulations were conducted for test cases characterized by varying both the Reynolds numbers and radius ratio independently. The heat transfer enhancement factor was used to compare the Nusselt numbers of serpentine channels to the Nusselt numbers of straight channels of the same channel profile.

All the objectives stated in the project objectives were obtained. The objectives were:

- To establish a relationship between the flow and heat transfer for different geometric configurations and flow conditions.
- Investigate flow phenomena for serpentine channel flow and identify the flow structures with the most dominant influence on heat transfer.
- Identify local variations in heat transfer within a serpentine unit and explain their presence.
- Find a geometric configuration and the flow conditions that have the best heat transfer behaviour and benefits most from using tortuous channels compared to straight channels.

Flow was characterized by a combination of primary- and secondary flow patterns. This was confirmed by streamlines that indicated the presence of the flow patterns within the post bend regions. The primary flow was found to be the bulk of the fluid moving through the bends without rotating. The strongest cause of heat transfer enhancement could be assigned to effective mixing.

The secondary flow showed up as vortices forming in the post bend-2 and bend-4 regions but not as prominent as Dean's vortices. The vortices seemed to enhance the mixing effect in the upcoming bends, corresponding to higher heat transfer enhancements in bend-1 and bend-3. The effective mixing was primarily responsible for the high values of heat transfer enhancements, due to the enhanced inertial effects introduced into flow by bends.

Heat transfer was much more dominant in the bend regions. From figure 4.6 and figure 4.7, the enhancement factors were seen to peak at locations exiting the bends, especially in the exit regions of bend-1 and bend-3. It was clearly visible in the cases where flow had a high Reynolds number that the enhancement factor at the end of the straight section rapidly

decreased to almost half the peak value at the beginning of the straight section. The rapid decrease was due to the stabilizing effect of straight sections following the mixing sections.

The major influencing factor in heat transfer was the flow velocity. By increasing the flow velocity and thus the Reynolds number, significant enhancements in the Nusselt number were observed. For the semi-circular cross-section, local enhancements more than four times higher than that possible in a straight channel for a Reynolds number value of 400, were seen. Local enhancements in the triangular cross-section were almost five times higher.

Higher Reynolds number flow corresponded to higher-pressure drops in the channels, which is a very undesirable effect in heat transfer engineering. Mention is only made of this fact since pressure drop was not a focus point within this investigation, but could be a deciding factor during actual heat exchanger design.

The radius ratio did not influence the heat transfer performance by much. A slight decrease in enhancement factor was noticed as the radius ratio increase (figure 4.10). A larger radius ratio corresponds to a larger bend radius. As the bend radius increased, the enhancement reduced due to weaker mixing strength.

For this investigation the best heat transfer enhancement compared to a straight channel was the equilateral triangular cross-section with an overall average enhancement factor $e_{Nu} = 4.856$ compared to that for the semi-circular cross-section of $e_{Nu} = 4.054$. Both values were for a Reynolds number value of 400, and the geometric factors set to $\frac{R_c}{d} = 1.0$ and $\frac{L}{d} = 4.5$.

The best overall heat transfer was found for a semi-circular serpentine channel with flow conditions set to $Re = 400$ and geometric factors set to $\frac{R_c}{d} = 1.0$ and $\frac{L}{d} = 4.5$. For this configuration, a Nusselt number value of 13.5 was observed compared to the observed triangular cross-section Nusselt number of 12.093.

The convective heat transfer coefficient (h) for the semi-circular cross-section was 18% better than for the triangular shape, even if the enhancement was better for the triangular cross-section. The enhancement factor of the triangular shape was 19.8% higher than in the semi-circular cross-section but the Nusselt number of the semi-circular shape was 11.5% higher than for the triangular shape.

Chapter 6 - Recommendations

During the investigation conducted on the straight channels, a problem was encountered using the polyhedral mesher whereby results from the investigation did not match values for the published Nusselt numbers.

To confirm the existence of the problem, a model mesh was build using both the surface extruder and the polyhedral mesher. Figure 6.1 shows the results obtained in the investigation. The first 255 mm of the channel consisted of cells generated by the surface extruder and the final 50 mm consisted only of polyhedral cells.

As expected from the surface extruder cells, the Nusselt number converged close to the theoretical value, but as soon as the flow reached the section comprised of polyhedral cells, the results rapidly changed.

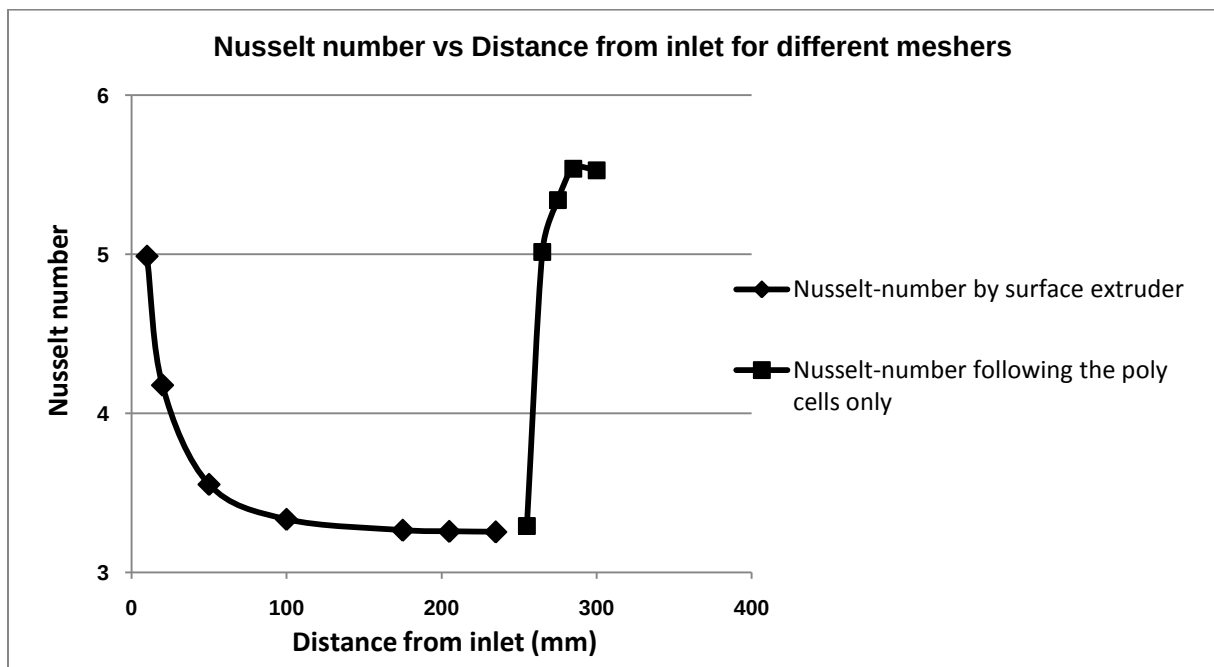


Figure 6.1 - Results for surface extruder mesher versus polyhedral cell mesher of a semi-circular straight channel with a length of 300 mm. The surface extruder mesher is used for the first 250 mm and the polyhedral mesher is used from 250 mm to 300 mm.

An inquiry of the problem produced no explanation. It was speculated that the problem might involve the interpolation approach employed by STAR-CCM+ during calculation in the post-processing stage. The planes used as input parts to the reports and user defined field functions (discussed in section 3.1.3), intersect the polyhedral cells at random locations. Since the properties are stored at the centre of the cells, linear interpolation is used to estimate the value at the positions where the plane intersected the cell. The calculated

values for the planes were representations of values for each cell which the plane intersected. The problem might lie in the technique implemented where the individual cell values were processed to produce the final value for the post-processor.

Another suggestion was that an investigation be done on the finite volume coefficients used in the matrix, or sparse matrix solution method implemented by STAR-CCM+. This however could be a more difficult route to investigate, since the same solver algorithm produced partly acceptable results using the same model containing different cell types. Coefficients are also calculated using generic face-based algorithms, independent of cell topology, which does not suggest an obvious source of errors.

A large amount of time was dedicated to solving the problem but produced no solution and until this inconsistency is cleared up, it is not recommended to use STAR-CCM+'s polyhedral cell mesher for modeling under similar conditions i.e. laminar flow in exclusively straight channels without proper verification.

When employing user defined field functions in STAR-CCM+, a problem arose while using the functions to calculate and plot the property using a single derived part such as a probe. This resulted in a time consuming operation where the Nusselt number needed to be calculated at each individual plane.

After completing this investigation, it is recommended to use STAR-CCM+ with caution for the particular internal flow problem due to difficulties experienced with the polyhedral mesher and due to the fact that STAR-CCM+'s built in Nusselt number calculator function appears to be inconsistent and values failed to converge, even for a very long flow length. The built-in function calculated the Nusselt number from values of the local heat transfer coefficient, and had difficulty in matching the published benchmark values.

Results obtained from the investigation proved a definite heat transfer enhancement. It is however, recommended that a study be done on the pressure and friction factor variation for the same conditions as for this study. In the heat exchanger industry, designs are also based on these properties and the final design would involve an optimal balance between heat transfer and pressure drop.

References

CD-adapco. (2009). User Guide: STAR-CCM+ Version 4.06.011. 3232, 3234.

du Toit, C. G. (2003). Friction Factor and Nusselt number for fully developed laminar flow in a semi-circular duct. (*In* Meyer, J.P., ed. HEFAT 2003 Heat Transfer, Fluid Mechanics, and Thermodynamics : 2nd International Conference on Heat Transfer, Fluid Mechanics, and Thermodynamics held in Livingstone, Victoria Falls, Zambia on 24-26 June 2003. Centurion : HEFAT, [CD-ROM] Paper DC2.)

Ferziger, J. H., & Peric, M. (2002). Computational Methods for Fluid Dynamics. 3rd ed. New York: Springer-Verlag.

Geyer, P. E., Fletcher, D. F., & Haynes, B. S. (2007). Laminar flow and heat transfer in a periodic trapezoidal channel with semi-circular cross-section. *International Journal of Heat and Mass Transfer*, 50:3471–3480.

Gupta, R., Geyer, P. E., Fletcher, D. F., & Haynes, B. S. (2008). Thermohydraulic performance of a periodic trapezoidal channel with a triangular cross-section. *International Journal of mass and heat transfer*, 51:2925-2929.

Heatric - Compact Heat Exchanger solutions. (2010).

http://www.heatric.com/compactness_heat_exchanger.html (Date of access 18 April 2010)

Hesselgreaves, J. E. (2001). Compact Heat Exchangers: Selection, Design and Operation. 1st ed. Oxford: Pergamon.

Incropera, F. P., Dewitt, D. P., Bergman, T. L., & Lavine, A. S. (2007). Fundamentals of Heat and Mass Transfer. 6th ed. Hoboken: Wiley.

Kalb, C. E., & Seader, J. D. (1972). Heat and mass transfer phenomena for viscous flow in curved circular tubes. *International journal of heat and mass transfer*, 15(4):801-817.

Pra, F., Tochon, P., Mauget, C., Fokkens, J., & Willemsen, S. (2008). Promising Design of compact heat exchangers for modular HTRs using Brayton cycle. *Nuclear Engineering and Design*, 238:3160-3173.

Rosaguti, N. R., Fletcher, D. F., & Haynes, B. S. (2005). Laminar Flow and Heat transfer in a Periodic Serpentine Channel. *Chemical & engineering technology: industrial chemistry, plant equipment, process engineering, biotechnology*, 28(3):353-361.

Rosaguti, N. R., Fletcher, D. F., & Haynes, B. S. (2007). Low Reynolds number heat transfer enhancement in sinusoidal channels. *Chemical Engineering Science*, 62(3):694-702.

Rosaguti, N., Fletcher, D. F., & Haynes, B. S. (2006). Laminar flow and heat transfer in a periodic serpentine channel with semi-circular cross-section. *International Journal of mass and heat transfer*, 49(17-18):2912-2923.

Rousseau, P. G. (2010). Thermal-fluid Systems Modeling - Course Notes (MGII 885). Potchefstroom: NWU.

Schönfeld, F., & Hardt, S. (2004). Simulation of helical flow in microchannels. *AIChE Journal*, 50(4):771-778.

Shah, R. K., & London, A. L. (1978). Advances in Heat Transfer, Supplement 1. Laminar Flow Forced Convection in Ducts. New York: Academic Press.

Southall, D., Le Pierres, R., & Dewson, S. J. (2008). Design Considerations for Compact Heat Exchangers. (*In Proceedings of ICAPP: Conference held in Anaheim, CA, USA on 8-12 June 2008.*)

Versteeg, H. K., & Malalasekera, W. (2007). An introduction to computational fluid dynamics: The finite volume method. Prentice Hall.

Appendix A

A.1 Results for straight channels

A.1.1 Circular straight channel

The following results apply to the circular cross-section.

Table A.1 - Results from the straight channel with circular cross-section.

Distance from inlet (mm)	Dimensionless axial distance (x^*)	Local Nusselt number (Nu_x) mesh1- 2206766 cells	Local Nusselt number (Nu_x) mesh2- 1228766 cells
0.1	2.71887E-05	105.717	151.1424
0.5	0.000135943	40.290	40.230
1	0.000271887	27.752	27.680
5	0.001359434	13.473	13.305
10	0.002718869	9.930	9.905
20	0.005437738	7.623	7.587
40	0.010875476	6.041	6.026
55	0.014953779	5.480	5.466
70	0.019032083	5.103	5.089
100	0.02718869	4.656	4.641
135	0.036704731	4.344	4.330
170	0.046220772	4.144	4.131
190	0.05165851	4.061	4.049
220	0.059815117	3.961	3.951
250	0.067971724	3.888	3.878
280	0.076128331	3.830	3.821
320	0.087003806	3.774	3.765
350	0.095160413	3.743	3.734
380	0.10331702	3.718	3.710
420	0.114192496	3.694	3.686
450	0.122349103	3.680	3.672
480	0.13050571	3.670	3.661
520	0.141381185	3.660	3.650
560	0.152256661	3.652	3.642
590	0.160413268	3.648	3.638
630	0.171288744	3.643	3.633
660	0.179445351	3.637	3.630
690	0.187601958	3.635	3.628
720	0.195758564	3.634	3.627
750	0.203915171	3.633	3.625
780	0.212071778	3.632	3.624
800	0.217509516	3.632	3.624

The models, for which above results were extracted, used the following mesh conditions.

mesh 1:

- 2206766 cells
- Base size – 3 mm
- 20 prism layers
- Stretching factor - 1
- Prism layer region thickness - 0.2 mm
- Relative target size - 5% of base size
- Relative minimum size - 5% of base size

mesh 2:

- 1228766 cells
- Base size – 5 mm
- 15 prism layers
- Stretching factor - 1
- Prism layer region thickness - 0.2 mm
- Relative target size - 5% of base size
- Relative minimum size - 4% of base size

A.1.2 Semi-circular straight channel

The following results apply to the semi-circular cross-section.

Table A.2 - Results from the straight channel with semi-circular cross-section.

Distance from inlet (mm)	Dimensionless axial distance (x^*)	Local Nusselt number (Nu_x) mesh2 – 2249574 cells	Local Nusselt number (Nu_x) mesh1 – 1611160 cells
0.1	6.6748E-05	46.397	43.463
0.5	0.00033374	16.681	17.153
1	0.00066748	12.082	12.180
5	0.003337401	7.544	6.773
10	0.006674801	5.357	5.211
20	0.013349602	4.327	4.241
40	0.026699204	3.729	3.683
55	0.036711406	3.558	3.520
70	0.046723607	3.463	3.427
100	0.06674801	3.368	3.332
135	0.090109814	3.322	3.285
170	0.113471617	3.302	3.265
190	0.126821219	3.296	3.259
220	0.146845623	3.291	3.253
250	0.166870026	3.288	3.251
280	0.186894429	3.286	3.248
295	0.19690663	3.286	3.248
320	0.213593633	3.284	3.247
350	0.233618036	3.285	3.246
380	0.253642439	3.286	3.246
420	0.280341643	3.287	3.247
450	0.300366046	3.285	3.247

The models, for which above results were extracted, used the following mesh conditions.

mesh 1:

- 1611160 cells
- Base size – 3 mm
- 20 prism layers
- Stretching factor - 1
- Prism layer region thickness - 0.15 mm
- Relative target size – 5.5% of base size
- Relative minimum size - 5% of base size

mesh 2:

- 2249574 cells
- Base size – 2 mm
- 20 prism layers
- Stretching factor - 1
- Prism layer region thickness - 0.1 mm
- Relative target size - 5% of base size
- Relative minimum size - 5% of base size

A.1.3 Equilateral triangular straight channel

The following results apply to the triangular cross-section.

Table A.3 - Results from the straight channel with triangular cross-section.

Distance from inlet (mm)	Dimensionless axial distance (x^*)	Local Nusselt number (Nu_x) mesh1 – 1656385 cells
0.1	7.06383E-05	36.639
0.5	0.000353192	15.035
1	0.000706383	10.586
5	0.003531916	5.701
10	0.007063832	4.355
20	0.014127664	3.511
40	0.028255328	2.946
55	0.038851076	2.766
70	0.049446824	2.660
100	0.07063832	2.552
135	0.095361732	2.504
170	0.120085145	2.486
190	0.134212809	2.481
220	0.155404305	2.478
250	0.176595801	2.477
280	0.197787297	2.475
295	0.208383045	2.475
320	0.226042625	2.474
350	0.247234121	2.475
380	0.268425617	2.476
420	0.296680945	2.476
450	0.317872442	2.476

The models, for which above results were extracted, used the following mesh conditions.

The mesh parameters are:

- 1656385 cells
- Base size – 3 mm
- 15 prism layers
- Stretching factor - 1
- Prism layer region thickness - 0.07 mm
- Relative target size - 5% of base size
- Relative minimum size - 5% of base size