

Multi-objective optimization on fluid flow and heat transfer of 3D micro-fin surfaces

by

Shima Soleimani

B.S., University of Tehran, 2013

M.S., University of Tehran, 2015

AN ABSTRACT OF A DISSERTATION

submitted in partial fulfillment of the requirements for the degree

DOCTOR OF PHILOSOPHY

Alan Levin Department of Mechanical and Nuclear Engineering
Carl R. Ice College of Engineering

KANSAS STATE UNIVERSITY
Manhattan, Kansas

2022

Abstract

Convective heat transfer enhancement (HTE) techniques push to increase the removal of excess heat from a solid area to the working fluid. In passive HTE techniques, the convective heat transfer is enhanced through geometric means by improving the flow and inserting attentively designed roughness — often termed fins, dimples, twisted tapes, rib tabulators, and wire coils — on the solid bodies throughout the flow passages. Enhanced surfaces, either macro or micro-textured, are commonly applied in commercial applications such as heat exchangers, cooling systems, chillers, refrigerants, power plants, air conditioners. Micro-fins (< 0.5 mm tall) have the advantage of reducing thermal resistance and the disadvantage of increased pressure drop when applied in heat-exchange applications.

Turbulence and heat transfer are two critical factors in the chain of world energy consumption. Hence, the development of more efficient energy transfer techniques in engineering applications through control and optimization of the thermal turbulent boundary layer and turbulence continues to be a subject of intense interest. This study reviews a passive flow control technique with a focus on unique characteristics of heat transfer in devices called riblets. These surfaces are considered beneficial to many commercial applications as they reduce drag skin friction between a surface and fluid and, in some cases, can enhance heat transfer in the turbulent flow regime. Hence, riblets are potential candidates to improve the efficiency of HTE equipment as technology pushes to revolutionize the design and manufacturing of heat exchanger equipment.

Studies of flow fields around the fins have appeared in the literature but often do not include conduction analysis in the fin, leaving a key optimization parameter unexplored. In addition, while 2 dimensional (2D) helical micro-fin tubes are the most widely used in heat exchanger applications, experimental results have demonstrated that many 3D micro-fin geometries

outperform 2D fins. Hence, one of the goals of this study is to fill these gaps by finding out the effects of 2D and 3D micro-fin design variables on heat-transfer enhancement and pressure drop.

Another goal of this study is to evaluate the potential of a data-mining model as a surrogate of computational fluid dynamic (CFD) models in 2D and 3D micro-fin tubes. Data-mining is a procedure of extracting and recognizing useful information and succeeding knowledge from databases using mathematical, statistical, artificial intelligence, and machine learning technique, and is still not often used in heat transfer studies. Two different least squares-support vector regression (LS-SVR) models are employed to estimate the Colburn j factor (j) and the Fanning friction factor (f) as functions of micro-fin geometric variables. Results of the parametric study showed that the best 2D micro-fin tube can enhance the thermo-hydraulic efficiency index (η) up to 1.18. Results of the LS-SVR model showed that the percentage of average absolute error (AAE) between simulated and estimated values of the Colburn j factor and the Fanning friction factor is 2.05% and 2.93% for 3D micro-fin tubes, respectively. The statistical results of this study proved that LS-SVR model can be an accurate surrogate for the CFD-based model.

Because of the great potential for optimization of micro-fin surfaces and the lack of any optimization work encompassing 3D geometries in this field, this study proposes a robust methodology to solve the multi-objective optimization problem for this class of heat exchanger surfaces with an accurate, first-principles numerical model and thereby determine the flow physics that characterize the optimal geometries. This study provides a representation of the geometry of 3D micro-fins with enough degrees of freedom to encompass a wide range of physically realizable surface geometries. To reduce computational time, a fluid flow channel is presented with a novel computational domain. The channel contains two-sided periodic domains and micro-fins on the top and bottom walls having an opposite angle. An accurate and parallelizable numerical model is

developed based on Reynolds average Navier-Stokes equations (RANS) and a realistic $k-\varepsilon$ model. The simulation procedure applies ANSYS Fluent 19.1 to solve convective heat transfer and turbulent flows over micro-fins and conductive heat transfer for solid conduction to maintain robustness for geometries that do not yield unity fin efficiencies. A boundary condition of constant temperature is defined at the channel walls. This study provides a robust methodology to integrate the numerical model into a multi-objective optimization algorithm named non-dominated sorting genetic algorithm (NSGA-II) to generate the best trade-off (Pareto front) solutions between enhancement of the Nusselt number and the Fanning friction factor of a micro-fin surface to a smooth surface at Reynold number (Re) of 49,000. Information from the Pareto front solutions will be used to explore the thermal and flow fields of the optimal solutions. The result of the Pareto front showed the significant effects of micro-fin height and pitch on objective values of optimal design geometries. This study presents a unique intersection of the drag reduction approach and heat transfer enhancement by finding unique operating geometries. For example, a particular optimal design geometry reduced drag by 8% but was also beneficial to heat transfer increasing by 21% compared to a smooth surface.

Multi-objective optimization on fluid flow and heat transfer of 3D micro-fin surfaces

by

Shima Soleimani

B.S., University of Tehran, 2013

M.S., University of Tehran, 2015

A DISSERTATION

submitted in partial fulfillment of the requirements for the degree

DOCTOR OF PHILOSOPHY

Alan Levin Department of Mechanical and Nuclear Engineering
Carl R. Ice College of Engineering

KANSAS STATE UNIVERSITY
Manhattan, Kansas

2022

Approved by:

Major Professor
Steven Eckels

Copyright

© Shima Soleimani 2022.

Abstract

Convective heat transfer enhancement (HTE) techniques push to increase the removal of excess heat from a solid area to the working fluid. In passive HTE techniques, the convective heat transfer is enhanced through geometric means by improving the flow and inserting attentively designed roughness — often termed fins, dimples, twisted tapes, rib tabulators, and wire coils — on the solid bodies throughout the flow passages. Enhanced surfaces, either macro or micro-textured, are commonly applied in commercial applications such as heat exchangers, cooling systems, chillers, refrigerants, power plants, air conditioners. Micro-fins (< 0.5 mm tall) have the advantage of reducing thermal resistance and the disadvantage of increased pressure drop when applied in heat-exchange applications.

Turbulence and heat transfer are two critical factors in the chain of world energy consumption. Hence, the development of more efficient energy transfer techniques in engineering applications through control and optimization of the thermal turbulent boundary layer and turbulence continues to be a subject of intense interest. This study reviews a passive flow control technique with a focus on unique characteristics of heat transfer in devices called riblets. These surfaces are considered beneficial to many commercial applications as they reduce drag skin friction between a surface and fluid and, in some cases, can enhance heat transfer in the turbulent flow regime. Hence, riblets are potential candidates to improve the efficiency of HTE equipment as technology pushes to revolutionize the design and manufacturing of heat exchanger equipment.

Studies of flow fields around the fins have appeared in the literature but often do not include conduction analysis in the fin, leaving a key optimization parameter unexplored. In addition, while 2 dimensional (2D) helical micro-fin tubes are the most widely used in heat exchanger applications, experimental results have demonstrated that many 3D micro-fin geometries

outperform 2D fins. Hence, one of the goals of this study is to fill these gaps by finding out the effects of 2D and 3D micro-fin design variables on heat-transfer enhancement and pressure drop.

Another goal of this study is to evaluate the potential of a data-mining model as a surrogate of computational fluid dynamic (CFD) models in 2D and 3D micro-fin tubes. Data-mining is a procedure of extracting and recognizing useful information and succeeding knowledge from databases using mathematical, statistical, artificial intelligence, and machine learning technique, and is still not often used in heat transfer studies. Two different least squares-support vector regression (LS-SVR) models are employed to estimate the Colburn j factor (j) and the Fanning friction factor (f) as functions of micro-fin geometric variables. Results of the parametric study showed that the best 2D micro-fin tube can enhance the thermo-hydraulic efficiency index (η) up to 1.18. Results of the LS-SVR model showed that the percentage of average absolute error (AAE) between simulated and estimated values of the Colburn j factor and the Fanning friction factor is 2.05% and 2.93% for 3D micro-fin tubes, respectively. The statistical results of this study proved that LS-SVR model can be an accurate surrogate for the CFD-based model.

Because of the great potential for optimization of micro-fin surfaces and the lack of any optimization work encompassing 3D geometries in this field, this study proposes a robust methodology to solve the multi-objective optimization problem for this class of heat exchanger surfaces with an accurate, first-principles numerical model and thereby determine the flow physics that characterize the optimal geometries. This study provides a representation of the geometry of 3D micro-fins with enough degrees of freedom to encompass a wide range of physically realizable surface geometries. To reduce computational time, a fluid flow channel is presented with a novel computational domain. The channel contains two-sided periodic domains and micro-fins on the top and bottom walls having an opposite angle. An accurate and parallelizable numerical model is

developed based on Reynolds average Navier-Stokes equations (RANS) and a realistic $k-\varepsilon$ model. The simulation procedure applies ANSYS Fluent 19.1 to solve convective heat transfer and turbulent flows over micro-fins and conductive heat transfer for solid conduction to maintain robustness for geometries that do not yield unity fin efficiencies. A boundary condition of constant temperature is defined at the channel walls. This study provides a robust methodology to integrate the numerical model into a multi-objective optimization algorithm named non-dominated sorting genetic algorithm (NSGA-II) to generate the best trade-off (Pareto front) solutions between enhancement of the Nusselt number and the Fanning friction factor of a micro-fin surface to a smooth surface at Reynold number (Re) of 49,000. Information from the Pareto front solutions will be used to explore the thermal and flow fields of the optimal solutions. The result of the Pareto front showed the significant effects of micro-fin height and pitch on objective values of optimal design geometries. This study presents a unique intersection of the drag reduction approach and heat transfer enhancement by finding unique operating geometries. For example, a particular optimal design geometry reduced drag by 8% but was also beneficial to heat transfer increasing by 21% compared to a smooth surface.

Table of Contents

List of Figures	xiii
List of Tables	xviii
Nomenclature	xix
Acknowledgments.....	xxiii
1 Introduction.....	1
1.1 Heat-exchanger Surface-level Design.....	5
1.1.1 Experimental Studies	7
1.1.2 Analytical Studies	8
1.1.3 CFD-based Studies.....	8
1.1.3.1 Parametric Studies.....	9
1.1.3.2 Optimization Studies	9
1.1.3.3 Surrogate-based Optimization Studies	11
1.2 Study Outline	12
2 Physical Mechanism of Fluid Drag and Convective Heat Transfer	13
3 Drag Reduction and Heat Transfer Enhancement by Riblet Surfaces- A review.....	19
3.1 The Role of Riblet in Fluid Drag and Heat Transfer	22
3.2 Riblet Design and Configuration	26
3.3 Riblet Fabrication Techniques	28
3.4 Experimental Studies	31
3.4.1 Experimental Drag Reduction Studies.....	33
3.4.1.1 Micro-Sized Channel Studies.....	33
3.4.1.2 Macro-Sized Channel Studies	34

3.4.2	Experimental Heat Transfer Studies	36
3.5	Numerical Studies	38
3.5.1	Numerical Studies of Drag Reduction	41
3.5.2	Numerical Heat Transfer Studies.....	45
3.6	Riblet Optimization.....	46
4	Numerical Model Setup	54
4.1	Governing Equations and Parameters	54
4.2	Discretization and Convergence	55
4.3	Boundary Conditions	56
4.4	Data Reduction.....	58
5	Parametric Study of Transverse and Axial 2D Micro-fins on Fluid Flow and Heat Transfer	61
5.1	Scenario Description	62
5.2	Geometry and Boundary Condition	63
5.3	Meshing and Independence Study	64
5.4	Model Validation	66
5.5	Results and Discussion	68
5.5.1	Velocity and Thermal Fields Around the Micro-fins	68
5.5.2	Fin Performance.....	73
5.5.3	Pitch Control for Optimum Design.....	75
6	Parametric Study of 2D and 3D Micro-fin Tubes on Fluid Flow and Heat Transfer	79
6.1	Design Variables	79
6.2	Geometry and Boundary Condition	81
6.3	Meshing and Independence Study	83

6.4	Model Validation	85
6.5	Result and Discussion	85
6.5.1	Parametric Study on 2D Micro-fin Tubes.....	85
6.5.2	Parametric Study on 3D Micro-fin Tubes.....	87
6.5.3	Velocity and Thermal Fields for Smooth and Best Micro-fin Tubes	92
7	Performance Study of LS-SVR Model on 2D and 3D Micro-fin Tubes	95
7.1	Application of a Data-Mining Method in a Micro-Fin Study.....	95
7.1.1	LS-SVR Method	95
7.1.2	LS-SVR Performance Criteria	98
7.1.3	Implementation of the LS-SVR Model.....	98
7.2	Result and Discussion	100
8	Optimization Study of 3D Micro-fins on Fluid Flow and Heat Transfer	104
8.1	Problem Description	104
8.1.1	Design Variables.....	105
8.2	Geometry and Boundary Condition.....	107
8.3	Meshing and Independence Study	109
8.4	Optimization Study	110
8.4.1	NSGAI Framework.....	111
8.4.2	Simulation Framework.....	113
8.5	Result and Discussion	115
8.5.1	Velocity and Thermal Fields of the Optimal Design Points	120
9	Summary and Conclusion.....	125
	References.....	128

List of Figures

Figure 1.1. Photos of a (a) 2D and (b) 3D micro-fin tube	7
Figure 2.1. A velocity profile over a smooth surface	14
Figure 3.1. Typical designs of riblet cross-section profiles: (a) blade, (b) scalloped, (c) saw-tooth, and (d) trapezoidal	26
Figure 3.2. Catalog of riblet configurations: (a) continuous longitudinal, (b) continuous in yaw, (c) inline segmented, (d) staggered segmented, (e) herringbone, and (f) zigzag	27
Figure 3.3. Integration of overlying turbulent vorticity with (a) a smooth surface and blade riblets at two different (b) $s^+ = 25.3$ and $s^+ = 41.2$ at $Re = 4,180$ (Martin and Bushhan, 2016a)	43
Figure 3.4. Conceptual model of vorticity behavior in inline staggered configuration considering two different j_r^+ (Martin and Bushhan, 2016a).	44
Figure 3.5. Thermo-hydraulic efficiency index over riblet surfaces at geometrical parameters of (a) $h/s = 1$ and (b) $h/s = 0.5$ considering different molecular Prandtl and Reynolds numbers (data from Benhalilou and Kasagi (1999))	45
Figure 3.6. Optimization of riblet cross-section profile (data from Bechert et al. (1997)).....	47
Figure 3.7. Effect of s^+ on drag reduction at different (a) h/s and (b) t/s ratios for blade riblets in an open oil-flow channel (data from Bechert et al. (1997)).....	48
Figure 5.1. A schematic of a single channel with transverse micro-fins on the lower wall	62
Figure 5.2. Model geometry for a rectangular micro-fin profile	64
Figure 5.3. The grid system of (a) whole domain and (b) zoom-in view at the solid-fluid interface for a case with a rectangular micro-fin.	65
Figure 5.4. Total heat-transfer rate and pressure gradient for different mesh resolutions of the (a) smooth case and (b) a case with a transverse rectangular micro-fin.....	66

Figure 5.5. Comparison of numerical and experimental results of stream-wise mean-velocity profiles for (a) the smooth case and (b) a case with transverse fins.	67
Figure 5.6. Velocity streamlines around transverse (a) rectangular, (b) trapezoidal, (c) triangular, and (d) parabola micro-fins of Scenario 6 at $p_f = 2$ mm	69
Figure 5.7. Mean-velocity profiles of turbulent boundary layers, through line 1 for (a) transverse and (b) axial micro-fins of Scenario 1, 6, and the smooth case.	70
Figure 5.8. Function curves of C_r^+ in (a) transverse and (b) axial cases, and κ in (c) transverse and (d) axial cases for different e^+	71
Figure 5.9. Contours of temperature for transverse rectangular micro-fins of Scenarios (a) 1, (b) 3, and (c) 5 at $p_f = 2$ mm	72
Figure 5.10. Contours of solid body temperature in transverse (a) rectangular, (b) trapezoidal, (c) triangular, and (d) parabola micro-fins, and axial (e) rectangular, (f) trapezoidal, (g) triangular, and (h) parabola micro-fins of Scenario 6 at $p_f = 2$ mm	73
Figure 5.11. Performance characteristics: (a) Q/Q_s and (b) f/f_s for transverse and axial cases at $p_f = 2$ mm	74
Figure 5.12. Thermo-hydraulic efficiency index for all cases at $p_f = 2$ mm.....	75
Figure 5.13. Velocity vectors and contours of temperature for transverse rectangular micro-fins of Scenario 1 with p_f/e ratios of (a) 2, (b) 4, (c) 8, and (d) 12.....	76
Figure 5.14. Velocity contours for axial rectangular micro-fins of Scenario 6 with p_f/e ratios of (a) 0.6, (b) 1.5, (c) 3, and (d) 6.....	77

Figure 5.15. Comparison of Q/Q_s , f/f_s , and the thermo-hydraulic efficiency index for different p_f/e ratios of (a) transverse rectangular micro-fin of Scenario 1 and (b) axial rectangular micro-fin of Scenario 6	78
Figure 6.1. A schematic of (a) micro-fin and cutter profiles and (b) an unrolled tube with micro-fins and cutters (solid and dash lines represent micro-fins and cutters, respectively)	80
Figure 6.2. Geometrical representation of solid and fluid computational domains of a (a) 2D and (b) 3D micro-fin tube	82
Figure 6.3. Grid system of solid and fluid domains with a zoom-in view at the solid-fluid interface for a 3D micro-fin tube	83
Figure 6.4. Mesh independency analysis for (a) the smooth tube and (b) Tube 3.....	84
Figure 6.5. Effects of the micro-fin number of starts and height for different helix angles of (a) 25°, (b) 45°, and (c) 60° (square dot represents $e = 0.1$ mm, triangle dot represents $e = 0.3$ mm, and round dot represents $e = 0.5$ mm; and blue and red dots represent d -type and k -type micro-fins, respectively)	86
Figure 6.6. Effects of the micro-fin height and number of starts on f/f_s for different helix angles of (a) 25°, (b) 45°, and (c) 60° (square dot represents $e = 0.1$ mm, triangle dot represents $e = 0.3$ mm, and round dot represents $e = 0.5$ mm; and blue and red dots represent d -type and k -type micro-fins, respectively)	86
Figure 6.7. Effects of the micro-fin height and number of starts on the efficacy index for different helix angles of (a) 25°, (b) 45°, and (c) 60° (square dot represents $e = 0.1$ mm, triangle dot represents $e = 0.3$ mm, and round dot represents $e = 0.5$ mm; and blue and red dots represent d -type and k -type micro-fins, respectively)	87

Figure 6.8. Effect of micro-fin variables of micro-fin (a) height, (b) helix angle, and (c) number of start on j/j_s and f/f_s	88
Figure 6.9. Effects of the p_f/e ratio and axial length of discontinuity on the j/j_s ratio for different number of discontinuities of (a) 0.25, (b) 0.33, (c) 0.5, (d) 1, (e) 2, and (f) 3	89
Figure 6.10. Effects of the p_f/e ratio and axial length of discontinuity on the f/f_s ratio for different number of discontinuities of (a) 0.25, (b) 0.33, (c) 0.5, (d) 1, (e) 2, and (f) 3	90
Figure 6.11. Effects of the p_f/e ratio and axial length of discontinuity on the thermo-hydraulic efficiency index for different number of discontinuities of (a) 0.25, (b) 0.33, (c) 0.5, (d) 1, (e) 2, and (f) 3.....	91
Figure 6.12. Temperature contours of a fluid cross-section of the (a) smooth and (b) micro-fin tube; a solid cross-section of the (c) smooth and (d) micro-fin tube; and (f) a partial solid-fluid interface of the micro-fin.....	93
Figure 6.13. Velocity contours for the (a) smooth and (b) micro-fin tube and (c) velocity vectors above a partial surface of the micro-fin	94
Figure 7.1. Flowchart of the LS-SVR model implementation.....	99
Figure 7.2. Simulated and estimated (a) j and (b) f in 2D micro-fin tubes and simulated and estimated (c) j and (d) f in 3D micro-fin tubes.....	102
Figure 8.1. A schematic of (a) micro-fin and cutter profiles and (b) a bottom wall of a channel with micro-fins and cutters (solid and dash lines represent micro-fins and cutters, respectively)	107
Figure 8.2. Geometric representation of solid and fluid sections of a channel with 3D micro-fins	107

Figure 8.3. Mesh system of solid and fluid domains with a zoom-in view at the solid-fluid interface for a 3D micro-fin channel.....	109
Figure 8.4. Mesh independency study for a (a) smooth and (b) micro-fin channel.....	110
Figure 8.5. An optimization framework for implementing NSGAI 112	112
Figure 8.6. NSGAI flowchart at first iteration.....	113
Figure 8.7. Simulation framework	114
Figure 8.8. Scattering of micro-fin (a) height, angle, pitch and (b) width, length of discontinuity, and number of discontinuities per micro-fin pitch at the initial population (blue dots), 10th (canyon dots), 20th (green dots), 30th (yellow dots), 40th (unfilled black dots), and 52rd (red dots) iteration	116
Figure 8.9. Convergence of objective functions (a) Nu/Nu_s and (b) f/f_s over iteration using of NSGAI algorithm	117
Figure 8.10. NSGA-II Pareto front after 52 iterations	118
Figure 8.11. Parametrization of the Pareto Front.....	120
Figure 8.12. Velocity contours and stream-lines above the micro-fin surface at the cross-midsection of the channels with design points (a) A (b) B, and (c) C.....	121
Figure 8.13. Velocity vectors in the micro-fin valleys of design points (a) A, (b) B, and (c) C. 122	122
Figure 8.14. Temperature contours at a partial solid-fluid interface of design point (a) A (b) B, and (c) C	123
Figure 8.15. Temperature contours above the micro-fin surface at the cross-midsection of the channels with design points (a) A (b) B, and (c) C	124

List of Tables

Table 3.1. Review papers over drag reduction methods	20
Table 3.2. Experimental studies in closed and open channels by riblets	36
Table 3.3. Numerical studies in closed and open channels with riblets	42
Table 4.1. Thermodynamic parameters	55
Table 5.1. Scenario description for different micro-fin geometries.....	63
Table 5.2. Comparison of experimental and numerical results for the smooth case	67
Table 6.1. Results of length study (errors are in respect to six helix turns).....	83
Table 6.2. Numerical and experimental results comparison at $Re = 49,000$	85
Table 7.1. Input and output of LS-SVR method for training and testing datasets for 2D and 3D micro-fin tubes	100
Table 7.2. Results of the RMSE for different values of regularization parameters in 2D and 3D micro-fin tubes	101
Table 7.3. Calculated statistical results with LS-SVR model for 2D and 3D micro-fin tubes ...	103
Table 8.1. Definition of state and constant design variables of the study	106
Table 8.2. Results of domain independency study (errors are in respect to 20 repetitions for length and width domains).....	108
Table 8.3. Comparison of results of this study with other studies for the best micro-fin tube at Re $= 49,000$ and $D_i = 15.54$ mm with water flow	119

Nomenclature

l_g	a function of A_g
A_c	cross-section flow area, m ²
A_{en}	enhanced surface area, m ²
A_g	riblet valley cross-section area, m ²
A_s	smooth surface area, m ²
b	bias
C^+	a constant
C_r^+	Roughness function
C_t^+	a constant
c_p	specific heat at constant pressure, J kg ⁻¹ K ⁻¹
d	vorticity diameter, m
D_i	inner tube diameter/channel height, m
D_o	outer tube diameter/total channel height, m
e	micro-fin/high-fin height, m
f	Fanning friction drag
F	roughness factor
f_d	Darcy friction drag
Fr	NSGAI front
g	gravitational acceleration, m s ⁻²
h	riblet height, m
h_{co}	heat-transfer coefficient, W m ⁻² K ⁻¹
I	number of individual solutions at each iteration solved in a parallel platform
J	number of individual solutions at each iteration
j	Colburn j factor
J_l	loss function
j_r	riblet longitudinal spacing, m
k	turbulent kinetic energy generation rate, m ² s ⁻²
K	thermal conductivity, W m ⁻¹ K ⁻¹

K_f	Kernel Function
l	a Lagrange multiplier
L	periodic channel/tube length
LA	Lagrange function
L_a	axial length of the discontinuity, m
Le	vorticity length, m
l_g	a function of A_g
l_r	riblet longitudinal length, m
$\dot{m}$	mass flow, kg s^{-1}
n	input data counter
N	total number of input data
N_c	cutter number of start
N_f	fin number of start
Nu	Nusselt number
P	periodic spatially pressure term, Pa
p	pressure, Pa
p_c	cutter pitch, m
p_f	micro- or high-fin pitch, m
Pr	molecular Prandtl number
Pr_t	turbulent Prandtl number
Q	Heat flux rate, W
q	average heat flux, W m^{-2}
q_{tot}	total heat flux, W m^{-2}
R	parent population
r	correlation coefficient
R^2	coefficient of determination
Re	Reynolds number
R_r	Reynolds analogy factor
s	riblet lateral spacing, m
St	Stanton number
t	riblet thickness, m

T	temperature, K
t_c	cutter thickness or discontinuity length, m
t_f	fin thickness, m
T_{in}	inlet bulk temperature, K
T_o	outlet fluid temperature, K
T_s	channel/tube's outer wall temperature, K
T_w	solid-fluid interface temperature, K
T_∞	free-stream fluid temperature, K
T_τ	friction temperature, K
u	velocity in the x direction, m s ⁻¹
u_τ	friction velocity, m s ⁻¹
V	average flow velocity, m s ⁻¹
v	velocity in the y direction, m s ⁻¹
x	stream-wise direction
X	set of decision variables
y	wall-normal direction
z	span-wise direction
<i>Greek letters</i>	
α	helix or yaw angle, degree
η	thermo-hydraulic efficiency index
β	a linearly subtracting term
θ_t	non-dimension periodic temperature variable
θ	included angle between sides of micro-fin/riblet, degree
λ	output of training dataset
λ_t	ratio between inlet and outlet flow temperature
δ	number of discontinuities per fin pitch
γ	a regularization parameter
σ	RBF Kernel parameter
ϖ^T	transposed form of ϖ
ϖ	weighting vector

φ	mapping function
Ω	offspring population
χ	input of training dataset
αt	thermal diffusivity, $\text{m}^2 \text{s}^{-1}$
τ	wall shear stress, N m^{-2}
$\tau_{en,w}$	wall shear stress of the enhanced wall in a periodic channel with micro-fin on the bottom wall, N m^{-2}
$\tau_{en,s}$	wall shear stress of the smooth top wall in a periodic channel with micro-fin on the bottom wall, N m^{-2}
ρ	density, kg m^{-3}
ν	kinematic viscosity, $\text{m}^2 \text{s}^{-1}$
μ	dynamic viscosity, $\text{kg m}^{-1} \text{s}^{-1}$
κ	Von Karman constant
ε	turbulent kinetic energy dissipation rate, $\text{m}^2 \text{s}^{-3}$
ε_H	heat-transfer eddy diffusivity, $\text{m}^2 \text{s}^{-1}$
ε_M	momentum eddy diffusivity, $\text{m}^2 \text{s}^{-1}$

Superscripts

$()^+$	dimensionless variable
$()^-$	local time-mean value
$()'$	Fluctuating component
$()^{Fn}$	front number

Subscripts

$()_s$	smooth surface
$()_w$	wall
$()_{It}$	iteration number

Acknowledgments

I would like to express my deepest gratitude to my advisor, Prof. Steven Eckels, for giving me an opportunity to work on this dissertation topic and providing me unlimited guidance. His energy, kindness, and encouraging personality have deeply inspired me throughout my study. I extend my special thanks to his wife, Brenda Eckels, and his family.

I would like to thank my dear committee members: Prof. Mohammad Hosni, Prof. Amy Bets, Prof. Byron Jones, and Prof. Dong Lin, for their productive comments, inspiration, and support. I owe gratitude to Prof. Hosni for his advice and guidance to overcome the academic challenges during my Ph.D. years.

I am incredibly grateful to my mother, Nasrin Roohi, and my father, Asghar Soleimani, for their care, love, prayers, and sacrifices for being a constant source of support and encouragement. I would like to extend my gratitude to my husband, Mehmetcan Gursoy, for his love, care, and endless support for my success. Also, I express my thanks to my sister, Sheida, who reminds me of this quote “growth is painful, change is painful, but nothing is as painful as staying stuck somewhere you don’t belong.” I would like to say thanks to my mother-in-law, Lale Gursoy, and father-in-law, Hami Gursoy, for their strong interest and support in successfully completing my dissertation.

I would like to thank my dear friends and colleagues Rohan Amare, Matthew Campbell, Kaitlin Bachle, Garrett Mann, Zubieda Alali, Meredith Schlabach, Puxuan Li, Courtney Markle, Prof. Ning Zhang, and Prof. Hosein Ghasemi Baboly for their constant encouragement.

1 Introduction

A heat exchanger design problem is a high-dimensional multi-objective optimization problem. Any consideration of the formal optimization of heat exchangers inevitably leads to the problem of choosing the appropriate objective and constraints. The most common objectives for a heat exchanger design problem are maximizing heat transfer rate (Q), minimizing weight/size/material cost, and minimizing operational cost/pumping power.

Reducing the complexity of the multi-objective optimization heat exchanger design problem can be achieved in a number of ways: converting the multi-objective to a single-objective optimization problem using a linear weighted method (Meng et al., 2011; Liu et al., 2017; Kim and Kim 2004), focusing on one of the objectives and dealing with others as constraints (Park and Moon, 2005; Foli et al. 2006; Copiello and Fabbri, 2009), defining an economic objective function such as total annual cost (Xie et al., 2008; Sanaye and Hajabdollahi, 2010; Rao and Saroj, 2017; Rao and Patel, 2011; Mirzaei et al., 2017), using an objective function in form of exergy destruction or entropy generation minimization based on the second law of thermodynamics (Jasiński, 2011; Abu-Hamdeh et al, 2020; Wang et al., 2015; Hamadneh et al., 2020), using the thermo-hydraulic efficiency index, which is defined as the ratio of heat transfer increase to pressure drop increase compared to a smooth surface, for the objective function (Liu et al., 2017), and using a volume goodness factor for the objective function which aims at maximizing HTE for a similar pumping power (Campet et al., 2020).

Combining the objective functions into a single objective function is problematic since the objectives represent a trade-off. In problems with a single objective function, the relative importance of the two objectives must be chosen a priori; if this importance is changed after the

optimization is performed, the optimization procedure must be repeated. Because of this, the applicability of the optimal solution is limited to only the prior chosen objective importance.

A more general way of approaching the objective selection for heat exchanger optimization is to consider the problem explicitly as multi-objective optimization. Multi-objective optimization aims to generate a set of non-dominant Pareto fronts representing a set of optimal solutions and gives an understanding of the trade-offs between conflicting objectives. Compromise designs could be selected using the Pareto front for different performance evaluation criteria (PEC) discussed by Webb and Kim (2005).

An optimization solution is classified into gradient-based and direct search approaches, while the objective can be defined as single or multiple. In a gradient-based approach, search directions are determined by the gradient evaluation of objective functions at specified values of decision variables. These variables are the same as design variables that need to be optimized in a heat exchanger design problem, and they determine the value of the objective function. The methods that follow the gradient-based approach require the direct expression of the objective function with decision variables and assume objective functions are differentiable, making them unusable in most heat exchanger design optimizations. In contrast, the direct-search approach is a problem-independent technique and can overcome a wide range of optimization problems. They only require the value of objective functions and do not require any information about the gradient or higher derivatives to search for an optimal solution. Evolutionary and meta-heuristic algorithms are applicable optimization methods in heat exchanger design problems. These algorithms are based on computational and search methods influenced by artificial intelligence and natural systems that provide excellent approximating solutions to great ranges of problems. For a multi-objective design problem of a heat exchanger, for any particular desired heat transfer rate, the

optimal decision variables should be selected that lead to the least possible pressure drop. Examples of multi-objective optimization are applications of NSGAI (Mann and Eckels, 2019; Ranut et al., 2014), fast NSGAI (Ranut et al., 2014), multi-objective genetic algorithm (MOGA) (Juan and Qin, 2014; Cavazzuti and Corticelli, 2008; Juan et al., 2016), particle swarm optimization (PSO) (Liu et al., 2017), and strength Pareto evolutionary algorithm 2 (SPEA2) (Copiello and Fabbri, 2009).

Typically, a heat exchanger design problem can be categorized into device-level and surface-level optimization problems. Device-level design optimization includes designing an overall shape of a heat exchanger such as position optimization of fins and tubes and tube diameter in a plain fin-and-tube heat exchanger (Juan and Qin, 2014), shell and tube diameter, and fin dimensions in a fin and tube evaporator (Ozkol and Komurgoz, 2005); dimension optimization of fin and stream length in a plate-fin heat exchanger (Sanaye and Hajabdollahi, 2010); geometry optimization of a plate fin-tube heat exchanger body (Ozkol and Komurgoz, 2005); and optimization of baffle types, number of shells in series and parallel, the shell orientation and type, number of tube passes, tube length, tube diameter, and tube layout pattern in a shell and tube heat exchanger (Tayal et al., 1999). Researchers typically use empirical or semi-empirical correlations to evaluate the objective functions of the device-level optimization problems since CFD calculation is impractical to evaluate the objective functions in these large-scale multi-physical problems.

Surface-level design optimization focuses on optimal geometrical shapes of a solid surface in which fluid acts as the source or sink of energy. In contrast to device-level optimization, the scale of surface-level optimization is often small enough to apply advanced numerical calculations to evaluate the optimization objective functions. Passive methods of HTE are usually applied to

remove excess heat from the solid surface to the working fluid for a heat exchanger at the surface-level design. Passive methods include inserting attentively designed roughness, fins, dimples, twisted tapes, rib tabulators, and wire coils on the solid bodies through the flow passages (Dewan et al., 2004).

Surface-level design optimization using passive methods requires both conduction and convection heat transfer and turbulent flow analysis. The CFD methods that can solve momentum and energy equations are RANS, large eddy simulation (LES), and direct numerical simulations (DNS). Generally, RANS methods are preferred in design optimization problems because of the impractical computational burden of LES and DNS methods. Examples of applying RANS in surface-level optimization problems include passage shape optimization of a parallel plate heat exchanger (Cavazzuti and Corticelli, 2008; Nobile et al., 2006), dimension optimization of a micro-channel heat exchanger (Foli et al., 2006), dimension and configuration optimization of micro-fin in a tube heat exchanger (Mann and Eckels, 2019; Dastmalchi et al., 2017), and dimension optimization of semicircular grooves in double pipe heat exchangers (Güngör et al., 2020).

An optimization problem of surface-level design can be solved using one of the following: (1) CFD-based parametric study, (2) CFD-based optimization study, and (3) surrogate-based optimization study.

A parametric study is often a logical first step in optimization as it reveals general trends and allows control of computational time. The solution for the CFD-based simple or multi-objective optimization study can be found by coupling advanced optimization algorithms with CFD models. In a surrogate-based optimization approach, a data-mining model is applied as a surrogate of the CFD-based simulation model and then coupled with the advanced optimization

algorithm. In this approach, the CFD-based simulation model first evaluates the values of objective functions based on design variables. These initial data are used to train and test the data-mining model to learn and distinguish complex and non-linear relationships between decision variables and objective functions. Therefore, this approach reduces the computational time compared to the CFD-only optimization approach. The applied data-mining methods in various heat exchanger problems include a kriging model (KRG) (Koo et al., 2014; Wansaseub et al., 2017), radial basis neural network (RBNN) (Shi et al., 2017), a response surface model (RSM) (Li and Kim, 2008), support vector regression (SVR) (Peng and Ling, 2015), LS-SVR (Kahani et al., 2018), artificial neural network (ANN) (Beigzadeh et al., 2016), multilayer perceptron ANN (MLP-ANN) (Peng and Ling, 2015), and automated learning of algebraic models for optimization (ALAMO) (Lindqvist et al., 2018).

1.1 Heat-exchanger Surface-level Design

Although surface-level design in heat exchangers using the passive method are beneficial for thermal performance, the added mass of material leads to higher costs for bulk commercial products and increased drag forces. Drag in fluid dynamics is the force that keeps objects from moving within the fluid, such as water or air. The direction of the net drag force is opposite of the fluid flow velocity. Two basic types of drag are form (pressure) drag and skin friction drag. Pressure drag is strongly associated with the object's shape and is dominated by static pressure forces. Skin friction drag arises from the attraction and contact of a solid surface with adjacent fluid layers in a flow that leads to resistance (Dean, 2011). Drag force in either form plays a vital role in energy usage. It can present possible efficiency improvements in industrial and municipal fields such as aircraft, marine, gas/oil pipelines, oil well operations, firefighting, shipping, racing

swimsuits, water heating, cooling systems, sewer systems, and biomedical systems, including blood flow (Dean and Bhushan, 2010; Brostow, 2008).

In recent years, optimizing convective heat transfer and hydraulic performance enhancement in industrial applications has been a popular research topic. HTE techniques using enhanced surfaces have been extensively reviewed by Dewan et al. (2004) and Ji et al. (2015). The mentioned surfaces change the contact surface area between the solid and fluid, and local fluid flow physics in a complex way that leads to intricate changes in thermal and hydraulic performance (Orlandi et al., 2016; Leonardi et al., 2015; Stalio and Nobile, 2003). Although some researchers (Okamoto et al., 1993; Pozarlik et al 2008; Kazi et al., 2012) stated that the primary mechanism for enhancement is recirculation with flow separation and secondary flow, Peeters and Sandham (2019) indicated in fully rough conditions, the actual recirculation region act as thermal resistance. With flow separation and recirculation, fluid forms eddies and vortices. The generation of eddies and vortices leads to more mixing in the flow and some situations, contributes to heat transfer significantly enough to compensate for the lowered recirculation zone. Although this mechanism is advantageous to heat transfer, it typically increases form drag and apparent viscous effects leading to excess drag forces. Thus, one grand challenge for designing HTE equipment is finding an enhanced surface that can augment heat transfer with a minimum pressure drop penalty.

One area of particular importance for surface-level optimization is the design problem of internal micro-fins (< 0.5 mm tall). Internal fins and tube materials with high thermal conductivity (K), such as copper or aluminum, are often found in tube heat exchangers with water as the working fluid. Liquid chillers and water-cooled condensers are typical examples of this heat exchanger type. Historically 2D trapezoidal micro-fin tubes are employed, but 3D micro-fin configurations have recently gained popularity. Figure 1.1 shows examples of internal 2D and 3D micro-fin tube

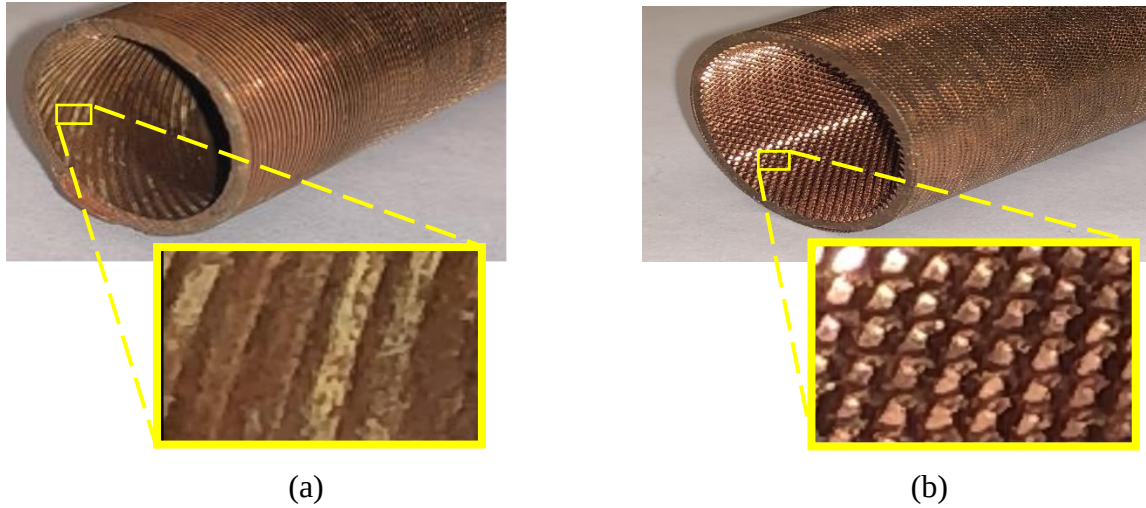


Figure 1.1. Photos of a (a) 2D and (b) 3D micro-fin tube

Different experimental, analytical, and CFD-based studies for performance evaluation of internally fin surfaces are reviewed in the following.

1.1.1 Experimental Studies

Takahashi et al. (1988) showed that a specific type of 3D micro-fin tube (A-8) enhanced heat transfer rate with a relatively low-pressure drop, resulting in 27% area reduction of a shell and tube evaporator not achievable using any other available 2D micro-fin tube. Webb (2009) studied the performance of three different types of 3D cone micro-fin tubes, and he compared the results with the best available commercially reported 2D micro-fin tube (Turbo BIII). He showed that two of the cone micro-fin tubes had 14 to 20% greater HTE, and they had nearly 5% greater thermo-hydraulic efficiency index than the Turbo BIII tube.

Very few works have been measured detailed characteristics of flow around the fins near the wall, and there is no work with temperature field measurement. Li et al. (2019) reported 2D flow-field measurement data using particle image velocimetry (PIV) in a flow channel with a micro-fin surface on the bottom wall at Reynolds number (Re) of 7,600, 14,000, and 28,000.

1.1.2 Analytical Studies

Analytical studies are limited to fin-shape optimization considering 1D or 2D conduction heat transfer analysis. The optimization of a fin profile and dimension have been studied with a constant (Ullmann and Kalman, 1989; Mikk, 1980; Fidanoglu et al., 2016) or a variable heat transfer coefficient (h_{co}) (Kundu and Das, 2002; Razelos and Imre, 1983; Kobus and Cavanaugh, 2006). These studies generally considered maximization of heat-transfer rate for a given fin mass or volume as a constraint (Kundu and Das, 2005), or minimization of fin mass or volume for a given heat-transfer rate as a constraint (Ullmann and Kalman, 1989; Mikk, 1980; Razelos and Imre, 1983; Kobus and Cavanaugh, 2006). Recently, Ayoobi et al. (2020) applied cylindrical and conical pin micro-fins and optimized fin dimension using MOGA considering two objectives of minimizing the temperature distribution function and maximizing heat flux function of the fins. Results reported the Pareto front for different thermo-physical properties. They showed that for an optimal cylindrical pin micro-fin, the temperature of the fin tips could reach the environmental temperature for different thermo-physical properties. Ghanadi Arab et al. (2013) optimized a transformative fin shape using a Bezier curve for four geometrical design points considering 2D conductive heat transfer analysis. They applied ANN as a surrogate of a finite-volume method to compute the temperature distribution function. The trained and tested ANN models combined with NSGAI for reporting two sets of Pareto fronts for heat transfer rate and each fin efficiency and fin surface area.

1.1.3 CFD-based Studies

RANS modeling is the most general CFD approach in industrial applications. Multiple RANS turbulence models have been applied in enhanced surfaces including Spalart-Allmaras, $k-\varepsilon$ of Goldberg, $k-\varepsilon$ of Lam-Bremhorst, shear stress transport (SST) of Menter (Kim et al., 2004), $k-\varepsilon$

of Norris and Reynolds (Liu and Jensen, 2001.), RNG $k-\varepsilon$ (Liu and Jensen, 2001), realizable $k-\varepsilon$, $k-kl-\omega$, Reynolds stress (RS) model, and SST $k-\omega$ (Rao and Saroj, 2017), and BSL (Nobile et al., 2006). Mann and Eckels (2019) proved that the realizable $k-\varepsilon$ model was superior to other turbulence models ($k-kl-\omega$, SST $k-\omega$, and RS) for calculating separated and complex flows with secondary circulation. The realizable $k-\varepsilon$ was able to predict the Nusselt number and Fanning friction factor at $Re = 49,012$ with a maximum deviation of -18.8 to +1.9% and -11.8 to -8.4%, respectively.

1.1.3.1 Parametric Studies

Recent publications have presented parametric studies on heat transfer and friction factors for internally 2D micro- or high-fin tubes with turbulent water flow. For example, Jamshed et al. (2019) studied the effects of axial pitch (p_f) with Reynolds number changing from 4,000 to 10,000. They obtained 33% enhancement at $Re = 10,000$ for one of the tubes. Liu et al. (2019) investigated the effects of fin variables of $e = 0.3 - 0.9$ mm, $\alpha = 10 - 40^\circ$, $N_f = 15 - 45$, thickness (t_f) = 0.5 - 0.9 mm, and different cross-section fin profiles of rectangular, circular, and triangular on heat transfer and friction factors at $Re = 44,430$. Their results showed helix angle and fin profile have the highest and lowest impact on the flow and heat transfer characteristics, respectively. Li et al. (2016) analyzed fin variables of $e = 0.3 - 0.6$ mm, $\alpha = 25 - 55^\circ$, $N_f = 10 - 50$ for an inner tube diameter (D_i) of 15.54 mm at $Re = 27,000$. They showed that p_f/e ratio is a significant factor that affects the location of separation and reattachment of flow in the boundary layer. They found variables of $e = 0.3 - 0.6$ mm, $\alpha = 25 - 35^\circ$, and $N_f = 40 - 50$ gave the best performance for the tube they studied.

1.1.3.2 Optimization Studies

An example of a single-objective optimization using PSO algorithm is the study of Dastmalchi et al. (2017), who optimized the thermo-hydraulic efficiency index for 2D helical

micro-fins inside a tube with water as the working fluid. They considered a fixed trapezoidal cross-section profile for micro-fins with design variables of micro-fin height ($0 \leq e \leq 0.5 \text{ mm}$), helix angle ($5^\circ \leq \alpha \leq 30^\circ$), and the number of starts ($10 \leq N_f \leq 50$) for $30,000 \leq Re \leq 100,000$ using three different inner tube diameters of 5, 10, 15 mm. In this study, water flow temperature was assumed greater than the ambient temperature and a constant convective heat transfer coefficient was selected as a boundary condition for the outer tube wall. They used a $k-\varepsilon$ model in the commercial software package ANSYS CFX v.15 for turbulent flow simulation and coupled the simulation model to PSO. Results showed the optimal number of micro-fin is 60 and optimal micro-fin height increases, and helix angle reduces by increasing Reynolds number for all three tube diameters. In the following, Mann and Eckels (2019) optimized internally 2D helical micro-fin geometry for a tube with an inner diameter of 15.54 at $Re = 49,000$. This study used water as the working flow and considered constant tube wall temperature for the boundary condition. They used a $k-\varepsilon$ model in Fluent for turbulent flow simulation and coupled the simulation model with NSGAI to achieve the Pareto front of the Nusselt number and the Fanning friction factor. The design variables were micro-fin height ($0.01 < e < 0.7 \text{ mm}$), helix angle ($0 < \alpha < 60^\circ$), and number of starts ($10 < N_f < 70$). Results showed flow characteristics change remarkably above and below the critical helix angle of 45° . In terms of 3D fin geometry optimization, Luo et al. (2019) optimized the geometry parameters in a pin-fin air channel with dimples/protrusions within the range of $8,500 \leq Re \leq 36,000$. The design variables were pin fin diameter, dimple/protrusion diameter, dimple/protrusion location, and dimple/protrusion depth. They applied the BSL model to simulate flow patterns and heat transfer in the ANSYS CFX. They coupled the simulation model with NSGA-II to achieve a Pareto front of heat transfer coefficient and the Fanning Friction factor. Results showed that in a pin-fin channel with dimples, the heat transfer coefficient is high at the

rear part of the dimple, but in a pin-fin channel with protrusions, the heat transfer coefficient is high at the leading edge of protrusion. They additionally observed that a pin-fin channel with protrusions leads to a higher pressure drop compared to a channel with dimples.

1.1.3.3 Surrogate-based Optimization Studies

Rout et al. (2015) optimized the dimension and arrangement of an internally 3D high-finned tube for laminar airflow, applying an RSM-assisted optimization approach. The RSM model was fed by a CFD-based database for the input variables of fin height, thickness, and number and output variables of Nusslet number and friction factor. They developed an RSM-NSGAI algorithm to maximize Nusslet number and minimize friction factor. Results showed that the RSM model was able to predict the output parameters within 5% of CFD-based data, which approved the excellent reliability of the surrogate model. Li and Kim studied shape optimization of 3D elliptic pin-fin arrays in turbulent air-channel flow using RSM and NSGAI. Results from the RSM model demonstrated the adjusted coefficient of determination (R^2) within the range of 0.9 to 1, which shows good accuracy for predicted data.

Recently, Campet et al. (2020) used surrogate-based optimization to reduce the computational cost for designing a 3D fin tube. They used wall-resolved LES to model a single start helical fin heat exchanger heated by a uniform heat flux at the tube wall. In their study, Gaussian process regression was applied as a surrogate model and combined with an advanced optimization algorithm to maximize the volume goodness factor. The decision variables were fin pitch, height, number of discontinuities per pitch (δ), and discontinuity length (t_c). Results showed that large and multiple discontinuities on fins lead to higher swirling motion in the near-wall region and enhance heat transfer to the maximum ratio of 2.1 while increasing pressure loss by a ratio of

2.7. They kept a p_f/e ratio greater than eight, limiting their decision space domain to only k -type roughness surfaces ($p_f/e > 4$), defined by Jiménez (2004).

1.2 Study Outline

The dissertation is outlined as follows. Chapter 2 presents the physical mechanism of fluid drag and convective heat transfer over smooth surfaces. Chapter 3 discusses momentum and heat transfer mechanisms over riblet surfaces. The chapter highlights riblet's design variables and configuration and the placement of riblets in closed channels. Riblet fabrication methods and materials are reviewed, including their advantages and disadvantages. The chapter separates drag reduction and heat transfer riblet studies based on numerical and experimental approaches, argues the effectiveness, and compares the superiority of the numerical methods. Different optimization viewpoints are also discussed to get minimum drag reduction and maximum HTE in thermal equipment using riblets. Chapter 4 is dedicated to the numerical model setup used in this study. This chapter explains governing equations, parameters, discretization methods, convergence criteria, boundary conditions, and data reduction manipulation that have been applied in the numerical models. Chapter 5 discusses the parametric study of transverse and axial 2D micro-fins on fluid flow and heat transfer while Chapter 6 reviews the parametric study of 2D and 3D micro-fin tubes on fluid flow and heat transfer. Performance of a data-mining model is (LS-SVR model) is evaluated on 2D and 3D micro-fin tubes to estimate fluid flow and heat transfer characteristics in Chapter 7 and Chapter 8 is dedicated to multi-objective optimization of 3D micro-fins to achieve the best trade-off (Pareto front) solutions between enhancement of Nusselt number and Fanning friction factor.

2 Physical Mechanism of Fluid Drag and Convective Heat Transfer

This section explores the physical description of friction drag and heat transfer over smooth surfaces. First, fluid flow is classified into laminar, turbulent, and transitional flow. In laminar flow, fluid particles move along smooth paths in parallel layers, generally when dealing with small pipes and low flow velocities. A turbulent flow regime is characterized by chaotic and random changes in time and space, and transitional flow is a mixture of laminar and turbulent flow.

Before the physical mechanism of drag force is discussed, a review of the characteristics of the turbulent boundary layer is helpful. Generally, three regions develop near the wall in turbulent flows. The closest region to the wall is the viscous sublayer where $y^+ < 5$, in which y^+ is the dimensionless distance from the wall ($y^+ = y\bar{u}_\tau/\nu$) and ν is kinematic viscosity. Following the calculation, the distance from the wall (y) is made dimensionless with the average skin friction velocity ($\bar{u}_\tau$) calculated from $(\bar{\tau}_s/\rho)^{0.5}$ while $\bar{\tau}_s$ is the average wall shear stress for a smooth surface and ρ is the fluid density. In this layer, the viscous forces are dominant and the flow damps fluctuations. The non-dimensional local time-mean velocity in the x -direction ($\bar{u}^+$) increases linearly as the distance from the wall increases ($\bar{u}^+ = y^+$), in which $\bar{u}^+ = \bar{u}/\bar{u}_\tau$ and $\bar{u}$ is the local time-mean velocity in the x -direction. The second region called the buffer layer is located between $5 < y^+ < 30$, in which neither viscous forces nor turbulent momentum forces are dominant. The third layer, the logarithmic layer, is located between $30 < y^+ < 100$ in which velocity and the wall distance follow a logarithmic relationship, is called the law of the wall ($\bar{u}^+ = (1/\kappa)\ln y^+ + C^+$),

where κ is the von Karman constant and C^+ is a constant (White, 2006). The flow from the wall to where the velocity follows the logarithmic law is defined as the inner region and the outside of this region is defined as the outer region. In the outer region, kinematic viscosity plays a minor role and in practical terms this region is inviscid for both internal and external flow. Figure 2.1 shows a velocity profile over a smooth surface.

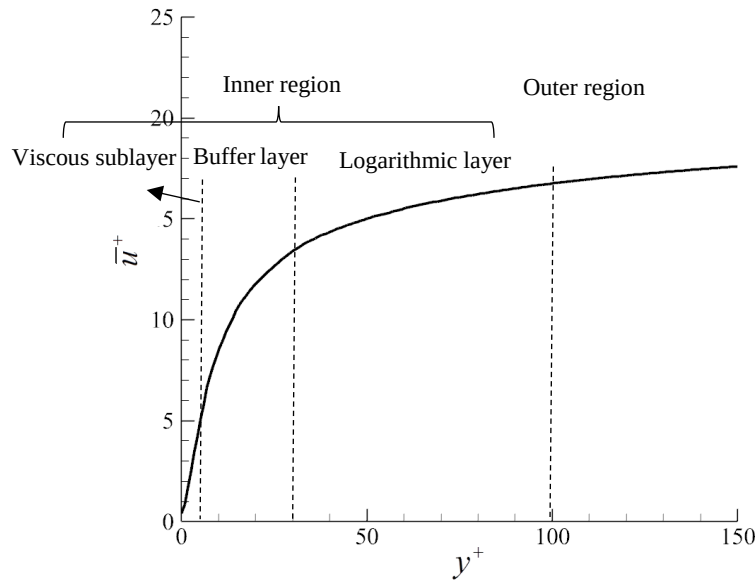


Figure 2.1. A velocity profile over a smooth surface

In a turbulent boundary layer, the wall produces and dissipates most turbulent coherent structures such as vortices, bursts (ejections), and sweeps associated with turbulent momentum transfer and drag forces (Hon and Walker, 1987). An essential driver for drag through the near-wall region is stream-wise vortices which spin along an axis line parallel to the mean flow velocity. Stream-wise vortices are dominant in the viscous sublayer and buffer layer, and move along the wall surface in the cross-flow direction (Kline et al., 1967). The interaction of the stream-wise vortices traveling through the mean flow velocity with the wall surface and adjacent vortices leads

to bursting motions, which significantly increase friction drag (Robinson, 1991). Vortices are characterized by vorticity diameter (d) and length (Le).

Heat convection is a heat transfer mechanism that occurs from the movement of the liquid or gas phase across a solid phase that removes energy. In force convection, the fluid is induced to move over the solid surface by an external source such as pumps or operated by pressure difference. The basic equation for heat transfer by convection is defined as $q = h_{co}(T_w - T_\infty)$, where q is the average heat flux along a wall; and T_w and T_∞ are the average wall temperature at the fluid-solid interface and free-stream fluid temperature, respectively. Convective heat transfer largely depends on flow properties, flow velocity, and surface geometry.

The temperature field that develops near a surface can be classified into the inner and outer regions. The inner thermal region follows the universal wall law for temperature within the turbulent boundary layer. The inner region is influenced by thermal boundary conditions and the molecular Prandtl number ($Pr = \nu/\alpha_t$) where α_t is thermal diffusivity and turbulent Prandtl number (Pr_t). To define the turbulent Prandtl number, first momentum eddy diffusivity and heat transfer eddy diffusivity must be specified. To model a turbulent boundary layer in two dimensions, information to evaluate $\overline{u'v'}$ in momentum equations of fluid flow are required. Due to the fluctuating velocity field, the term refers to the apparent turbulent shear stress (Reynolds stress) in RANS equations. Equation (2.1) can be used as a closure in momentum equations when Reynolds averaging is applied. And, Equation (2.2) shows a momentum differential equation for the constant-property turbulent boundary layer using the momentum eddy diffusivity concept (Kays and Crawford, 1993).

$$\overline{u'v'} = \varepsilon_M \frac{\partial \bar{u}}{\partial y} \quad (2.1)$$

$$\bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} - \frac{\partial}{\partial y} \left[(\nu + \varepsilon_M) \frac{\partial \bar{u}}{\partial y} \right] + \frac{1}{\rho} \frac{d\bar{p}}{dx} = 0 \quad (2.2)$$

where, u' and v' are the instantaneous value of the fluctuating component of velocity in the x and y direction, respectively; $\bar{v}$ is the local time-mean velocity in y -direction; ε_M is momentum eddy diffusivity; and $\bar{p}$ is pressure.

Subsequently, to model a thermal boundary layer in two dimensions, the energy equation of fluid flow is required. The term $\overline{v'T'}$ is defined as an apparent turbulent heat rate in the direction normal to the main flow and can be substituted in the energy equation of fluid flow using Equation (2.3). Equation (2.4) shows the differential equation for a constant-property thermal boundary layer using the heat transfer eddy diffusivity concept (Kays and Crawford, 1993).

$$\overline{v'T'} = \varepsilon_H \frac{\partial \bar{T}}{\partial y} \quad (2.3)$$

$$\bar{u} \frac{\partial \bar{T}}{\partial x} + \bar{v} \frac{\partial \bar{T}}{\partial y} - \frac{\partial}{\partial y} \left[(\alpha_t + \varepsilon_H) \frac{\partial \bar{T}}{\partial y} \right] = 0 \quad (2.4)$$

where T' is the fluctuating temperature component; $\bar{T}$ is local time-mean temperature; and ε_H is heat-transfer eddy diffusivity. Then, turbulent Prandtl number can be defined in the following equation:

$$Pr_t = \frac{\varepsilon_M}{\varepsilon_H} = \overline{u'v'} \frac{d\bar{T}}{dy} / \overline{v'T'} \frac{d\bar{u}}{dy}. \quad (2.5)$$

The mean temperature profile can be calculated using the concept of the law of the wall for temperature. Wall heat flux $\bar{q}_w = -K \left(\partial \bar{T} / \partial y \right)_w$ plays the physical role in the thermal boundary layer, same as wall shear stress in the turbulent boundary layer.

Total heat flux is defined as $\bar{q}_{tot} = -\rho c_p [\alpha_t + \varepsilon_H] \partial \bar{T} / \partial y$ which is a summation of molecular $(-K(\partial \bar{T} / \partial y))$ and turbulent heat flux $(-\rho c_p \overline{v' T'})$, where c_p is the specific heat at constant pressure. A friction temperature $\bar{T}_\tau = \bar{q}_w / (\rho c_p u_\tau)$ is introduced as a temperature scale with constant properties. Accordingly, a dimensionless mean-time temperature can be defined as $\bar{T}^+ = (T_w - \bar{T}) / \bar{T}_\tau$. Equation (2.6) shows an assumption in that wall heat flux is equal to total heat flux (Favre-Marinet and Tardu, 2013).

$$\frac{\bar{q}_{tot}}{\bar{q}_w} = \frac{\partial \bar{T}^+}{\partial y^+} \frac{\alpha_t + \varepsilon_H}{\nu} = \frac{\partial \bar{T}^+}{\partial y^+} \left(\frac{1}{Pr} + \frac{1}{Pr_t} \right) \kappa(y^+) = 1 \quad (2.6)$$

With the assumption in Equation (2.6), it is derived that the dimensionless mean-time temperature and wall distance follow a linear relation in the thermal-viscous sublayer and a logarithmic relation in the logarithmic layer in a thermal boundary layer. A thermal-viscous sublayer refers to the closest layer to the wall in which heat-transfer eddy diffusivity is negligible compared to thermal diffusivity and consequently $\bar{T}^+ = Pr y^+$. The second layer is called the thermal buffer layer in which none of the thermal diffusivity and heat-transfer diffusivity are dominated. The third layer is called the logarithmic layer in which thermal diffusivity is negligible compared to heat-transfer eddy diffusivity in a fully developed turbulent mixing region. Using Equation (2.6), it is derived that $\bar{T}^+ = (Pr_t / \kappa) \ln y^+ + C_t^+$, where the constant $C_t^+ = C_t^+(Pr)$ depends on molecular Prandtl number. A simplified procedure is to assume that the turbulent Prandtl number is constant. The consensual values found in the literature are $Pr_t = 0.9$ for turbulent wall flows and $Pr_t = 0.7$ in free shear layers (Favre-Marinet and Tardu, 2013).

The Reynolds analogy is a well-known concept with the primary assumption that turbulent momentum is analogous to heat transfer in a turbulent flow. In other words, the analogy is based

on the equality of momentum and heat-transfer eddy diffusivities that implies $Pr_t=1$. The analogy claims that the transport of momentum and heat transfer mainly relies on the same turbulent coherent structures. The Reynolds analogy factor is defined as $R_r = 2St/f$, where St is the Stanton number. (Stalio and Nobile, 2003). A Stanton number is a dimensionless number that measures the rate of the heat transfer coefficient to the thermal capacity of the fluid ($St = Nu/(Re.Pr)$).

3 Drag Reduction and Heat Transfer Enhancement by Riblet Surfaces- A review

This review focuses on drag reduction and subsequently on a unique intersection of drag and heat transfer research by emphasizing surfaces that reduce drag but may have a beneficial (or in some cases negative) effect on heat transfer. This combination is often not considered as most enhanced heat transfer surfaces increase drag in proportion to the heat transfer enhancement.

Drag reduction was accidentally discovered and reported by Toms (1949) and is the phenomenon of flow vorticity amelioration causing a reduction of skin friction. The net turbulent drag reduction methods include additive and non-additive techniques. The additive technique involves adding drag-reducing agents (DRAs) to a fluid in pipelines that contains polymers, suspended solid particles, surfactants, and biological additives. The non-additive techniques modify the surface and have compliant surfaces, micro-enhanced surfaces (riblets, dimples, micro-morphology surfaces), oscillating walls, super-hydrophobic surfaces, composite surfaces, microbubbles, and heating walls (Abdulbari et al., 2013; Luo et al., 2015). Many papers that have reviewed drag reduction using additive or non-additive techniques are summarized in Table 3.1.

Table 3.1. Review papers over drag reduction methods

Reference	Additive or non-additive drag reduction techniques
Abdulbari et al. (2013)	• Riblets • Dimples • Oscillating walls • Compliant surfaces • Microbubbles • Polymer additions • Micro-morphology surfaces • Super-hydrophobic surfaces
Luo et al. (2015)	• Microbubbles • Heating walls • Oscillating walls • Composite walls
Al-Sarkhi (2010)	• Polymer additives in two multiphase flows • Polymer additions • Compliant walls • Surface coatings
Bandyopadhyay (1986)	• Spot forcing • Bursting • Riblets • Outer-layer devices • Convex curvatures • Microbubbles
Dean and Bhushan (2010)	• Micro-morphology surfaces • Riblets • Blowing and suction in turbulent boundary layers
Ghanadi et al. (2012)	• Span-wise traveling waves • Span-wise wall oscillations • Riblets
Golovin et al. (2016)	• Super-hydrophobic surfaces
Luo (2015)	• Micro/Nano morphology • Polymer additions • Gas injections (bubbles, air layers, partial air cavity) • Riblets • Compliant surfaces • Outer-layer devices
Perlin et al. (2016)	• Super-hydrophobic surfaces • Surface heating and cooling • Electromotive force • Mass blowing and suction • Active wall motions and deformations
Bixler and Bhushan (2013b)	• Riblets • Riblets
West et al. (2018)	• Micro-morphology surfaces • Super-hydrophobic surfaces
Saravi and Cheng (2013)	• Riblets

This review focuses on engineered-based enhanced surfaces called riblets, a class of passive control techniques used to reduce skin friction drag and, as expected, change the heat transfer in turbulent flow channels. Riblets are micro-structures originally inspired by dermal denticles on fast-swimming shark skins (Bixler and Bhushan, 2013b). The riblets are micro-grooves distended from a wall and generally oriented with the flow direction. Drag reduction can be achievable using riblets in a closed channel using either water (Jung and Bhushan, 2009; Bixler and Bhushan, 2013a; Dean and Bhushan, 2012) or air (Bixler and Bhushan, 2013.a; Enyutin et al., 1995); moreover, in an open channel using either water (Hou et al., 2017; Wang et al., 2000), oil (Grüneberger and Hage, 2011; Bechert et al., 2000, 1997), or air (Sareen et al., 2014; Walsh, 1982; Walsh and Lindemann, 1984) .The essential capability of riblets to enhance contact surface area and reduce friction drag makes riblet surfaces a prospective candidate for HTE equipment (Benhalilou and Kasagi, 1999; Walsh and Weinstein, 1979). Thus, researchers are constantly dealing with technical challenges to get more drag reduction and heat transfer benefits from riblets.

Riblets can also be combined with super-hydrophobic surfaces to make more significant drag reduction (Barbier et al., 2008 and Bixler and Bhushan, 2013a). On super-hydrophobic surfaces, the contact angle of a water droplet typically surpasses 150° , and the roll off angle is less than 10° . Therefore, they intensely repel water droplets and are thus extremely hydrophobic. The super-hydrophobic coating is made of hydrophobic materials which produce roughness with dimension and spacing ranging from 100 nm to 10 μm . The coating causes trapping and retaining air pocket in pores and voids of the roughed surface forming an air plastron. This phenomenon generates a slippery interface with the liquid flowing over the plastron, resulting in drag reduction. Super-hydrophobic surfaces can be created with either random roughness (Ling et al., 2016 and Vajdi Hokmabad and Ghaemi, 2016) or designed roughness (Byun et al., 2008 and Lee and Choi,

2008). Maynes et al., (2008) were the first to numerically document the combined momentum and heat transfer in the thermally developed regime over super-hydrophobic surfaces at a constant temperature. Results showed that a slippery gas-liquid interface acted adiabatic, resulting in a lower heat transfer coefficient than a no-slip boundary condition. More recently, Ciri and Leonardi (2019) numerically studied heat transfer over super-hydrophobic surfaces with designed roughness. They observed a simultaneous increase in surface heat flux and a decrease in drag under the assumption of finite surface tension. Their research highlights the need to investigate heat transfer and drag behavior on riblet surfaces with hydrophobic coatings. Important questions that remain to be answered are: does the combination of riblet and super-hydrophobic help increase thermo-hydraulic performance? does a heterogeneously hydrophobic (semi-slippy) surface help enhance or reduce the heat transfer coefficient?

Generally, riblets are characterized by their design, configuration, material, and manufacturing techniques. Kinematic viscosity and Reynolds number have a key role in friction drag in riblet studies. Greater viscosity intensifies the interaction of flow molecules in near-wall layers, leading to higher friction drag (White, 2006). Moreover, thermo-physical properties such as thermal diffusivity and thermal conductivity of riblet surface material significantly affect heat transfer characteristics. Friction drag reduction and heat transfer advantages generally improve as the Reynolds number increases in riblet studies (Bixler and Bhushan, 2013b; Benhalilou and Kasagi, 1999). Despite this importance, no literature review paper has addressed riblet performance in drag reduction and heat transfer, or even heat transfer.

3.1 The Role of Riblet in Fluid Drag and Heat Transfer

Before discussing how riblets control a turbulent coherent structure in a turbulent boundary layer, it is important to discuss how drag reduction is measured. Drag reduction between riblets

and smooth surfaces is reported as $\Delta\bar{\tau}/\bar{\tau}_s$, in which $\Delta\bar{\tau}$ is the difference of the wall shear stress between riblets and smooth surfaces. For Newtonian fluids, wall shear stress near a smooth surface parallel to the flow with no-slip condition follows the relation of $\bar{\tau}_s = \mu(\partial\bar{u}/\partial y)_w$ in which μ is fluid viscosity.

Riblets technically are a kind of structured roughness. The riblet alters the structure of the logarithmic layer where most turbulent energy is generated (Jiménez, 2004 and Nikuradse, 1933). Nikuradse (1933) experimentally studied the effect of roughness on the law of the wall. He also developed the similarity laws for wall-bounded flows with roughness. The study of near-wall velocity profiles revealed that the structure of riblets stretches the stream-wise velocity gradient in the valleys of riblets resulting in generally lower wall shear stress than a smooth surface (Bannier et al., 2016). Coles (1956) described the law of the wall for roughness as $\bar{u}^+ = (1/\kappa)\ln y^+ + C^+ - F$, where the smooth wall constants are $\kappa = 0.41$ and $C^+ = 5$, and F is a constant that shows the shift of the mean velocity profile in the logarithmic layer because of wall roughness and is defined as a roughness factor. Tani (1988) and Choi (1989) assessed the consequence of riblets as an upward shift in the dimensionless mean velocity profile compared to a smooth surface due to negative values of the roughness factor that lead to a reduction in wall shear stress.

The manifestation of effective protrusion height and its effect on drag reduction was first explained by Bechert et al. (1986) and Bechert and Bartenwerfer (1989). The effective protrusion height is the distance between vertical shifts in the origin of the stream-wise and span-wise velocity profiles measured in a riblet valley (Bechert et al., 1997). In other words, the effective protrusion height parameter can show how much reduction of span-wise flow fluctuations contributes to drag reduction. Moreover, Goldstein et al. (1995) and Choi (1989) declared that riblets provide drag reduction through damping the cross-flow in the near-wall region.

Another theory describing the riblet drag reduction mechanism revolves around the behavior of turbulent coherent structures caused by fluid mixing. With appropriate riblet design, the structure of riblets can eject large scale stream-wise vortices outside the viscous sublayer and thus weakening the vortices as well (Ng et al., 2016; Okabayashi et al., 2018; Lee and Lee, 2001; Dean and Bhushan, 2010; Martin and Bhushan 2016a, b). Lifting the large-scale vortices results in the vortices' contact with just riblet tips compared to the whole wall surface (Bixler and Bhushan, 2013a). The small-scale secondary vortices in the riblet valleys are responsible for moving up and weakening the large-scale stream-wise vortices above the riblets (Hou et al., 2017; El-Samni et al., 2007; Martin and Bhushan, 2016b). The velocity of these secondary vortices and their interaction is not detrimental to drag reduction effectively.

According to the Reynolds analogy, as riblets damp turbulent flow motion near the wall, it is expected that riblet surfaces would have reduced heat transfer. Walsh and Weinstein (1979) and Lindemann (1985) experimentally showed a heat transfer coefficient is remarkably augmented over heated riblet surfaces, even within the drag reduction regime of riblets. The results of heat transfer measurement by Choi (1993) demonstrated riblets cause a downward shift in the logarithmic layer of the dimensionless mean temperature profile compared with that over a smooth surface. In other words, riblet surfaces increase the mean temperature gradient at the wall, inferring the thickness of the thermal-viscous sublayer over riblet surfaces is diminished, which supports the augmentation of the heat transfer coefficient over riblet surfaces (Choi, 1993; Choi and Orchard, 1997). Choi and Orchard (1997) performed statistical analyses of temperature and velocity fluctuations over riblet surfaces. Results demonstrated that the riblet seemed to manipulate the momentum of turbulent structures in the near-wall region but modified the thermal boundary layer much farther away from the wall (about 10 wall units). In other words, the active

manipulation region caused by riblets differs between turbulent and thermal boundary layers. This result shows the apparent discrepancy of the turbulent transport mechanisms between momentum and heat over riblet surfaces can break the Reynolds analogy.

Benhalilou and Kasagi (1999) numerically demonstrated the extremely high local heat transfer coefficient around the riblet peak counterbalanced by the intense reduction among riblet valleys for fluid with low molecular Prandtl number ($Pr = 0.71$). In other words, only the riblet peaks contribute significantly to the heat transfer process, and it thus gets hard to gain a dramatic enhancement compared to the smooth wall. The authors state that for high-Prandtl-number fluid ($Pr = 100$), the thickness of the thermal layer remarkably decreases to the order of riblet height which is in the order of channel half-height for low-Prandtl-number fluid. A high local heat transfer coefficient exists around the riblet peak in high-Prandtl-number fluid, but the local heat transfer coefficient is greater among the riblet valleys than the low-Prandtl-number fluid, leads entire riblet surface contribution in heat transfer improvement compared to a smooth surface. They declared the mean secondary flow has a significant role in the heat transfer mechanism for fluid with a high molecular Prandtl number, when dimensions of the secondary flow vorticity are large enough compared to the thermal boundary layer thickness. Jin and Herwing (2014), and Stalio and Nobile (2003) also supported the view of a high heat transfer coefficient around the riblet tip.

The study of the heat transfer mechanism over riblet surfaces in turbulent flow is limited to only few papers that consider air as the working fluid. Despite the importance of an enhanced surface for heat transfer augmentation, no one clearly explains the physics behind riblets' heat transfer mechanism. It is still controversial whether riblets with appropriate design can enhance heat transfer in different fluid flow or not. The mechanism of turbulent thermal flow over riblet surfaces could be better understood by more fundamental research analysis using different riblet

designs and fluid containing a greater range of molecular and turbulent Prandtl numbers and Reynolds numbers. For example, the research could incorporate turbulent coherent structures with thermal boundary layers to gain the relationship between the scales of thermal and fluid mechanisms.

3.2 Riblet Design and Configuration

The performance of riblets to reduce drag and HTE greatly depends on riblet design variables, including riblet cross-section profiles, dimensions, and configurations (Bechert et al., 1997; Martin and Bhushan, 2016b; Lindemann, 1985; Choi and Hamid, 1991). The most common riblet cross-section profiles, including rectangular (Blade), parabola (scalloped), triangular (saw-tooth), and trapezoidal, are shown in Figure 3.1. Riblet dimensional design variables are defined as riblet height (h), thickness (t), lateral spacing (s), longitudinal spacing (j_r), and longitudinal length (l_r), as shown in Figure 3.2. In most studies, dimensionless design variables are presented to compare with other literature studies. The dimensionless form of length-scale variables is marked by + obtained by multiplying the dimensional length by $\bar{u}_\tau/\nu$.

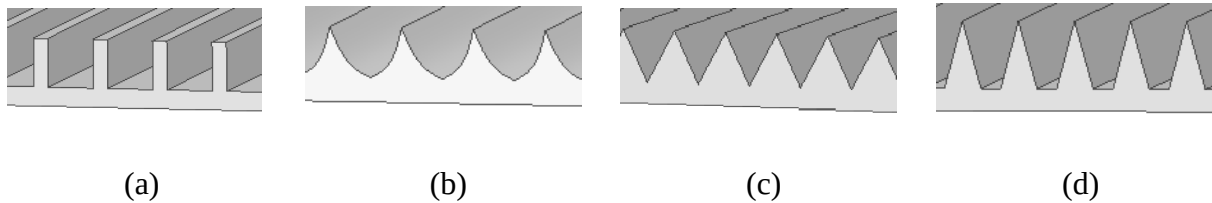


Figure 3.1. Typical designs of riblet cross-section profiles: (a) blade, (b) scalloped, (c) saw-tooth, and (d) trapezoidal

The arrangement of riblets on a surface is called configuration, divided into two categories: 2D and 3D configurations. The 2D riblets are longitudinal micro-grooves with constant cross-sectional profiles over their length called continuous riblets. Continuous riblets can be aligned

parallel to flow direction (continuous longitudinal) or have an offset defined as yaw angle (α) from flow direction (continuous in yaw). The 3D riblets include inline segmented, staggered segmented, herringbone (converging-diverging), and zigzag configurations. A catalog of riblet configuration is shown in Figure 3.2. Many studies have applied different riblet configurations to optimize drag reduction. Examples include continuous longitudinal (Benhamza and Belaid, 2009; Aupoix et al., 2012; Hou et al., 2017), continuous in yaw (Benschop and Breugem, 2017), inline segmented (Jung and Bhushan, 2009; Bixler and Bhushan, 2013a), staggered segmented (Martin and Bhushan, 2016a; Bechert et al., 2000), herringbone (Benschop and Breugem, 2017), and zigzag riblets (Okabayashi et al., 2018).

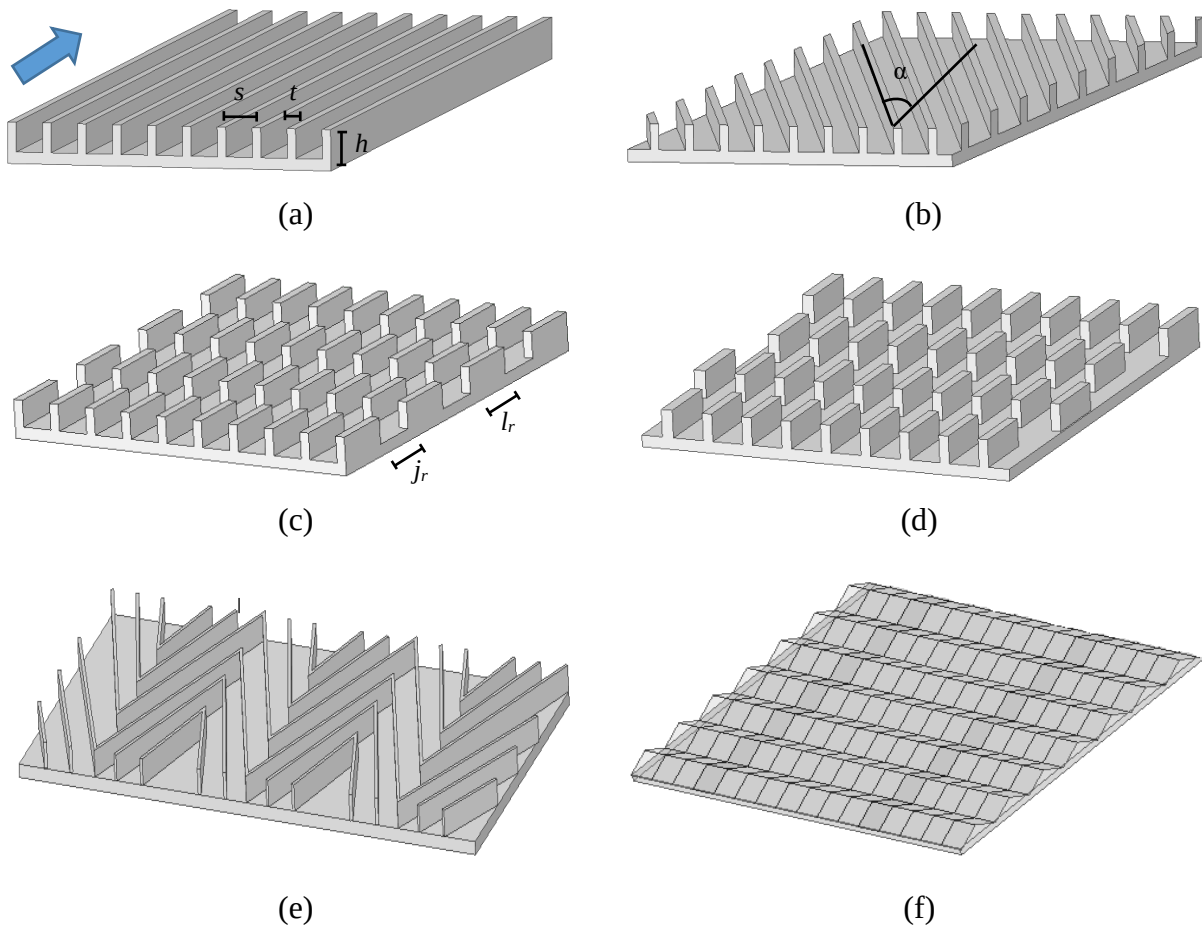


Figure 3.2. Catalog of riblet configurations: (a) continuous longitudinal, (b) continuous in yaw, (c) inline segmented, (d) staggered segmented, (e) herringbone, and (f) zigzag

3.3 Riblet Fabrication Techniques

Micro-scale manufacturing techniques and material properties are two critical factors that dictate what surfaces can be produced. Riblets are produced by various metals and polymers, including different fabrication techniques and assembly procedures. Minor variations in the riblet characteristic dimension significantly affect drag reduction and heat transfer benefits as riblet material can also affect substantially riblet heat conductions. Therefore, the precision of the fabrication techniques and riblet material used to manufacture riblet structures are very important. In addition to riblet bulk material and substrate material, the micro-fabrication technique selection should consider costs, strength, longevity, abrasion, forming, easiness of procedure, and chemical compatibility. This section discusses riblet material and micro-scale fabrication techniques for producing large-scale surfaces with riblets.

The top ten materials referenced in literature with high heat conductivity appropriate for riblet surface material include silver, copper, gold, aluminum nitride, silicon carbide, aluminum, tungsten, graphite, and zinc. Silver has the highest thermal heat conductivity and zinc the lowest. Aluminum riblet surfaces with the aim of HTE and drag reduction can be referred to in the works of Choi and Hamid (1991), Choi and Orchard (1997), and Choi (1993).

Machining consists of subtractive or additive manufacturing and is a common technique for micro-scale manufacturing. An acrylic milling technique (Dean and Bhushan, 2012) and an aluminum milling technique (Hou et al., 2017; Walsh and Lindemann, 1984) are two examples of subtractive machining. Another machining manufacturing type is a rolling technique addressed by Romans and Hirt (2010). Generally, milling and rolling techniques are unable to provide the very sharp riblet tips required for optimal performance and are not suitable for large-scale applications. Well-designed and sharp riblet tips can be fabricated using the profile grinding technique (Denkena

et al., 2010). As pointed out by Kordy (2015) and Dean and Bhushan (2010), the main drawback of this technique is the deficiency of hardening on the ultimate riblet surface, leading to diminished drag reductive surface efficiency as the material gradually wears out.

The 3M Company uses the molding technique and film application to produce saw-tooth riblets on vinyl adhesive sheets, bonding onto large surface areas with various substrates (Marentic and Morris, 1992). The height and spacing of the saw-tooth riblets are the same, and the size varies from 44 to 150 μm . Although, the bonded riblet films have some benefits such as ease of application, they have some disadvantages, including the additional weight of the films, lack of coverage of the entire surface area, the impossibility of bonding some curved surfaces, detaching the films from surfaces, and becoming embossed under pressure or heating (Romans and Hirt, 2010; Stenzel et al., 2011; Marentic and Morris, 1992). It should be noted that vinyl has very low heat conductivity and is not appropriate for riblet surfaces used for heat transfer.

Bechert et al. (1997, 2000) applied the technique of metal shims to create an adjustable plate with slits and ribs. The disadvantage of this technique is the damaging effects of gaps and slits.

Casting (Kim, 2014; Zhao et al., 2012; Chen et al. 2014) and printing (Wen et al., 2014) techniques are other manufacturing processes suitable for more complex 3D riblet structures such as replicated shark skin. Compared to a printing technique limited to small-scale applications, casting is applicable for large surface areas. Bechert et al. (2000) used a combination of printing and casting techniques.

Madou (2011) highlighted lithographic techniques, including electron beam lithography, soft lithography, photolithography, nanoimprint lithography, X-ray lithography, and colloidal lithography. The techniques can fabricate regular or irregular micro/nano-surface structures. The

soft lithography technique to produce riblet surfaces was addressed in the work of Jung and Bhushan (2009). More recently, Bilinsky (2017) provided a novel, direct contactless microfabrication method to implement riblets into an external aircraft surface. The process utilizes the photolithographic technique (Brennan et al., 2010) and computer-chip manufacturing to directly print 3D micro-structures onto the aircraft surface. The method uses a single-exposure 3D printing that manufactures 3D riblets in a single step. They give a method resolution for microstructures down to the order of 1 μm . Bilinsky and Wilson (2018) indicated that the method is applicable for large-scale surfaces, bringing cost and time down.

A laser-etching technique can produce a high-quality riblet surface with an accurate riblet tip and profile (Bixler and Bhushan 2013a, b). The technique has no necessity for material post-processing. But West et al. (2018) point out that this technique is costly and time-consuming. Bixler and Bhushan (2013a) applied a hydrophobic nanostructured coating technique to alter the wettability properties of laser-etched riblet samples.

Kordy (2015) used the lacquering technique originated from a coating process. First, wet lacquer was pressed and embossed with a transparent film, then UV-curing was radiated through this transparent film to cure the wet lacquer and make the layer of riblet unchangeable. Finally, the riblet layer was hardened by heating or other quicker radiation-based techniques. As pointed out by Romans and Hirt (2010), the main drawbacks of this technique are riblet-form deterioration, abrasion, and dirt-repelling composition of the structured lacquer.

Other riblet fabrication techniques include polymer/copolymer extrusion (Marentic and Morris, 1992), hot embossing (Han et al., 2008), embossed polymers (Marentic and Morris, 1992), wet and dry etching (Brennan et al., 2010), and painting (Grüneberger and Hage, 2011).

3.4 Experimental Studies

This section summarily discusses experimental measurement techniques dealing with friction drag and heat transfer characteristics over riblet surfaces in closed and open channels. Afterward, experimental studies on drag reduction and heat transfer over riblet surfaces are reviewed.

Experimental measurement techniques can be divided into drag, flow and temperature visualization, and heat transfer measurement techniques. For closed channels, a pressure distribution measurement is one common way for measuring drag using oil, water, and air (Jung and Bhushan, 2009; Bixler and Bhushan, 2013a; Rohr et al., 1992; Liu et al., 1990; Reidy and Anderson, 1988; Dean and Bhushan, 2012; Bixler and Bhushan, 2013b). In this method, pressure is measured with a pressure manometer between two points through the flow and drag reduction, and is defined with a negative pressure drop.

Wall shear stress can be directly measured employing a force-balance method in open channels using water (Bixler and Bhushan, 2013b), oil (Grüneberger and Hage, 2011), as well as wind tunnels using air (Walsh and Weinstein, 1979; Lindemann, 1985).

Flow and temperature visualization techniques are applied to depict patterns in flow and temperature fields to gain qualitative and quantitative understanding. The 3D measurement of turbulent and thermal boundary layers is critically important to understand the complex dynamic interaction of multiple scales. Robinson (1991) stated that “what is observed in a turbulent boundary layer is strongly dependent on the tools used to make the observation”. PIV, particle-tracking velocimetry (PTV), and Tomographic PIV are techniques to provide 3D velocity and temperature fields. Other techniques are laser-droplet velocimetry (LDV) and hot-wire anemometry that is limited to producing 2D velocity and temperature fields. The 3D velocity field

enables the calculation of velocity gradient tensor, which is essential for most intuitive vortex detection schemes such as Q -criterion, Δ -criterion, λ_2 -criterion (Kolar, 2007).

In the riblet study, capturing 3D experimental data help explain why keeping large scale stream-wise vortices above riblets leads to lower cross-stream velocity fluctuation above a riblet surface than a smooth surface in the turbulent boundary layer. Generally, span-wise shear stress causes drag increase and when it comes to heat transfer mechanism based on the general concept that says the generation of greater either stream-wise or span-wise vortices lead to more mixing in the flow and subsequently more HTE. It can be concluded that span-wise shear stress leads to higher heat transfer. But to answer the question of how the variation of span-wise shear stress affects drag and heat transfer, more 3D experimental data are required.

Hou et al. (2017) applied 2D and 3D PIV and PTV as flow-visualization techniques in water for measuring turbulence over a riblet surface. Wang et al. (2000) applied LDV and hydrogen bubble-flow visualization in water flow to measure turbulence over a riblet surface. Reidy and Anderson (1988) used the LDV to measure the velocity field in water flow over riblet surfaces. The LDV measured velocity as close as 0.05 mm from the wall. Hot-wire and cold-wire anemometers have been applied to measure velocity and temperature fluctuations through a wind tunnel (Choi and Hamid, 1991; Choi and Orchard 1997; Choi, 1993). Tummers and Steunebrink (2019) applied PIV with suspended thermochromic liquid crystals as flow tracer particles to visualize flow and temperature fields which can be a potential technique for riblet studies.

The convective heat transfer coefficient can be obtained by measuring heat flux and the difference in temperature of the heated test-plate surface and free-stream temperature. Heat flux can be measured using different direct and indirect methods. An indirect method based on Fourier's law is carried out by measuring a temperature difference over a piece of test solid with known

thermal conductivity. For more details, see the work of Li et al. (2019). Heat flux is also equal to the power input to the heater test solid and can be indirectly calculated by measuring a voltage drop over a known electrical resistance of the solid (Choi and Hamid, 1991; Choi and Orchard, 1997; Walsh and Weinstein, 1979). The direct method of heat flux measurement relies on heat flux sensors that directly measure the amount of heat passing through the solid where the heat flux sensor is mounted (Peabody, 2001).

3.4.1 Experimental Drag Reduction Studies

Drag reduction using water and oil has been experimentally measured in closed channels with rectangular and circular cross-sections. Since the role of channel dimension and the subsequent neighboring wall in a close channel are significant in the drag mechanism (Bixler and Bhushan, 2013a), closed channels are divided into micro-sized and macro-sized channels. The application of riblets in closed channels can be classified into four placements that include riblets placed on the bottom wall and top wall, aligned riblets on the bottom and top walls, and the spanwise shift of the top-wall riblets compared to the bottom-wall riblets. Experimental studies are presented in Table 3.2, proving the advantage of using riblets in drag reduction in both closed and open channels. The following subsections give more detail on the results presented in the table.

3.4.1.1 Micro-Sized Channel Studies

The application of riblets in a micro-sized rectangular channel has been investigated in several studies. Bixler and Bhushan (2013a) measured a maximum pressure-drop reduction of 34% using water and various riblet geometry and configurations, including a nanostructured coating resulting in super-hydrophobic riblet surface samples. They showed that drag reduction is enhanced by super-hydrophobicity and adhesion corresponding to coated samples. They also studied the effect of riblets on the bottom, top, or both sides of the channel. Results indicated that

placement of aligned riblets on the bottom wall performs more preferably than riblets on the top wall. In addition, aligned riblets on both sides with a suitable channel height minimize drag better than riblets on the top wall. Following to results of their conceptual model, with aligned riblets on both channel sides, the bottom and top wall vortices are mixed in a way that increases drag, but the negative vorticity interaction can be minimized by changing the channel height into twice the vorticity diameter, and also changing riblet spacing equal to the vorticity diameter. They remarked on a 13% improvement in pressure drop when increasing the Reynolds number from 550 to 4,700. Jung and Bhushan (2009) also achieved a 23% pressure drop using water and blade riblets with inline segmented configuration on the bottom and top walls.

3.4.1.2 Macro-Sized Channel Studies

The application of a continuous saw-tooth riblet in a circular closed channel using water was studied by Rohr et al. (1992) and Liu et al. (1990), who achieved drag reduction of 9% and 7%, respectively. Reidy and Anderson (1988) applied a drag-reducing polymer solution in conjunction with riblets. They gained a 28% drag reduction, which almost was equal to the sum of drag reduction of the riblet and polymer techniques applied individually. They theorized that greater drag reduction is possible in a macro-sized, circular closed channel rather than in an open channel using water.

Dean and Bhushan (2012) studied the effect of continuous blade riblets on the bottom wall of a rectangular water channel. The results demonstrated no overall benefit of drag reduction using riblets in this type of water channel. The reason may be due to the roughness left throughout the milling process, nature, or the dimensional characteristic of the rectangular channel. The result of this study appears to be an anomaly. Therefore, the experiment should be redone with higher accuracy for the same rectangular channel dimension and some different channel dimensions.

Experimental studies using water and oil in an open channel are only rectangular cross-sections. A water-channel experiment was studied by Reidy and Anderson (1988) and Neumann and Dinkelacker (1991), who applied continuous saw-tooth riblets and achieved 8.1% and 9% drag reduction, respectively. Hou et al. (2017) also applied the same type of riblets as Wang et al. (2000) and achieved a 7.4% drag reduction.

An open oil-channel experiment was conducted by Grüneberger and Hage (2011), and Bechert et al. (2000) using continuous riblets, who achieved 7.6% and 7.3%, drag reduction, respectively. Bechert et al. (1997) used continuous and staggered segmented riblets and achieved 9.9% drag reduction.

Table 3.2. Experimental studies in closed and open channels by riblets

Fluid	Channel type	Riblet				Maximum pressure drop/drag reduction	Reference
		Profile	Configuration	Material	Fabrication technique		
Water	Close/rectangular	Blade	Inline segmented	Acrylic	Lithography	23% ^(a)	Jung and Bhushan (2009)
		Blade and saw-tooth	Continuous and inline segmented	Vinyl and acrylic	Laser etching/3 M company	34*% ^(a,c)	Bixler and Bhushan (2013a)
		Blade	Continuous	Acrylic	Milling	0	Dean and Bhushan (2012)
	Close/circular	Saw-tooth	Continuous	Vinyl	3 M company	9%	Rohr et al. (1992)
		Saw-tooth	Continuous	Vinyl	3 M company	28% ^(b)	Reidy and Anderson (1988)
		Saw-tooth	Continuous	Vinyl	3 M company	7%	Liu et al. (1990)
	Open	Saw-tooth	Continuous	Vinyl	3 M company	8.1%	Reidy and Anderson (1988)
		Saw-tooth	Continuous	Vinyl	3 M company	9%	Neumann and Dinkelacker (1991)
		Trapezoidal	Continuous	Aluminum	Milling	7.4%	Hou et al. (2017)
		Trapezoidal	Continuous	Polyurethane	Milling/painting	7.6%	Grüneberger and Hage (2011)
Oil	Open	Blade and trapezoidal	Continuous and staggered segmented	Steel	Printing and casting/metal shims	7.3%	Bechert et al. (2000)
		Blade, saw-tooth, and scalloped	Continuous	Plexiglas	Metal shims and milling	9.9%	Bechert et al. (1997)

^a Micro-sized channel^b Combined with polymer injection^c Combined with super-hydrophobic coating.

3.4.2 Experimental Heat Transfer Studies

Despite the potential of riblets to improve the efficiency of HTE equipment, limited experimental data is available dealing with the combined analysis of friction drag and heat transfer characteristics in a thermal turbulent boundary layer over heated riblet surfaces. All available experiments in HTE studies using riblets were conducted in a wind tunnel applying fully developed

turbulent air flow. The primary experiment was carried out by Walsh and Weinstein (1979) at the NASA Langley Research Center. They applied different continuous riblets that varied in cross-section profiles (blade, saw-tooth, and trapezoidal), height, and spacing. They showed the thermo-hydraulic efficiency index as a function of free-stream velocity (15 to 40 m s⁻¹) for different riblet surfaces. Results showed that saw-tooth riblet surfaces with a larger wetted surface area had greater heat transfer augmentation than the corresponding drag increased ($\eta > 1$). They achieved up to 10% increase in thermo-hydraulic efficiency. Results of two saw-tooth riblets with $h^+ = 20$, and $s^+ = 15$ and 20 represented HTE together with a friction drag decrease; as a reminder, h^+ and s^+ are defined in Figure 3.3. Lindemann (1985) experimentally measured the Reynolds analogy factor for a smooth surface, different enhanced surfaces (including riblets, an oscillating transverse curvature, and micro air bearings), and wavy walls in a low-speed wind tunnel. Tests were conducted with free-stream velocity within the range of 13.7 to 27.4 m s⁻¹. He showed the Reynolds analogy factor against a Reynolds number for different types of enhanced surfaces. Results demonstrated the remarkable potential of riblets to enhance the Reynolds analogy factor compared to other enhanced surfaces and wavy walls. Results showed the ability of saw-tooth riblets with $s^+ = 16$ and $h^+ = 24$ to increase the thermo-hydraulic efficiency index up to 36% within the drag-reducing regime of riblets. In addition, all of the heat transfer efficiency data increase to some degree with increasing free-stream velocity for different enhanced surfaces. Another NASA study was performed by Choi and Hamid (1991) to investigate the heat transfer coefficients of saw-tooth riblet surfaces with great attention to minimize compensations of heat loss due to thermal radiation and lateral heat conduction, which apparently studies of Walsh and Weinstein (1979) and Lindemann (1985) have suffered from these problems. The tests were conducted at four different free-stream velocities ranging from 2.25 to 9 m s⁻¹. In their studies, the percentage increase in heat transfer was

investigated over the range of $6 < s^+ < 60$ for two saw-tooth riblet surfaces with a unit aspect ratio ($s = h = 0.73$ and 1.83 mm). The results demonstrated that a heat transfer coefficient could be enhanced by riblets up to 30% within a drag reduction regime of riblets at $s^+ = 30$. They declared the amount of heat loss due to thermal radiation greatly affects the magnitude of the convective heat transfer coefficient for low Reynolds numbers tests. This is a common problem when air is used as the working fluid and is not often accounted for in experimental studies. Choi (1993) did experimental measurements over heated riblet surfaces for a surface with saw-tooth riblets having the same e and s values of 1.83 mm. Based on the results of hot-wire and cold-wire anemometry studies, he confirmed the basic concept of previous studies that riblets can enhance heat transfer even up to 35% without incurring a penalty of drag increase. Choi and Orchard (1997) enhanced the accuracy of previous studies, minimizing experimental errors due to free convection, lateral conduction, and thermal radiation, and found a significantly smaller 10% HTE within the drag-reducing regime ($s^+ \leq 30$). Dagenet-Frick et al. (2012) carried out an experiment in a large boundary-layer wind tunnel to study the application of enhanced surfaces consisting of riblets and actuators in asymmetric heat exchangers. Results showed the combination of actuators and riblets of large dimensions has the best performance in terms of a high heat transfer coefficient and weak pressure losses. But, the dimension of this study's riblet is beyond the drag-reducing regime of riblets ($s^+ > 30$).

3.5 Numerical Studies

Study of turbulent flow and turbulent heat transfer flow over riblets with detailed experimental measurements of flow and temperature fields are expensive, complicated, or even impractical to obtain. CFD provides access to all detailed information. A brief description of the methods and their performance in modeling riblets are given in the following.

Mathematical formulations that can be used to solve the momentum and energy equation in CFD are RANS, LES, and DNS. Navier-Stokes equations describe the motion of fluid and typically require an extensive range of time and length scales to simulate a flow accurately. The RANS technique utilizes an averaging method leading to a time-average solution of the Navier-Stokes equations for turbulent fluid flow. The Reynolds stress term is calculated through a turbulence closure model. Generally, the turbulence model uses a turbulent viscosity approach in the Reynolds stresses formulation. The RANS simulation provides limited information on the structure of turbulent vortices. The LES model divides velocity modeling into large and small length scales, and the Navier-Stokes equations are filtered and solved for large length scales. The effects of small length scales, which are greatly responsible for enhancing computational cost, are modeled. The most accurate simulation method is DNS, which does not use any turbulence model as a closure to Navier-Stokes equations and numerically solves the entire range of time and length scales. The disadvantage of this method is the high computational cost that does not solve turbulent flow in some complex geometries, high Reynolds numbers, and high Prandtl numbers.

Launder and Li (1993) applied the RANS approach with a $k-\varepsilon$ eddy viscosity model to simulate the turbulent flow field over riblet surfaces. They declared the weakness of their computational model suppressed any turbulence-driven secondary-flow motions and hurt the comparison with experimental data. Tullis and Pollard (1994) applied a time-dependent viscous wall-region model in which eddies are only calculated in the region between the wall and outer edge of the viscous wall region. Their result was in agreement with the results of other studies that applied DNS. Benhamza and Belaid (2009) used a Reynold stress model (RSM) to simulate flow over riblet surfaces. They declared that although RSM appropriately predicts drag reduction and flow field, apparent disagreements of Reynolds stress values were found, especially within the

channel core. Aupoix et al. (2012) showed the ability of Spalart-Allmaras (SA), $k-\varepsilon$ baseline (BSL), and shear-stress-transport (SST) turbulence models to capture the shifting velocity profile inside the riblet valleys. Results showed that SA and SST models acceptably predict drag reduction, but the $k-\varepsilon$ BSL model overestimates drag reduction. Martin and Bhushan (2016a) applied LES to predict drag reduction and achieved further insight into the drag reduction mechanism. They validated their result for the smooth surface with Spalding's law of the wall, whereas they did not validate the results of the riblet surface. LES was also used to explore the flow mechanism generated by riblets in turbulent flow in studies of Bannier et al. (2016) and Martin and Bhushan (2014). One of the earliest studies of drag reduction using DNS was the work done by Choi et al. (1993). Results showed a good agreement with experimental data reported by Walsh (1983), Benhalilou et al. (1991), and Vukoslavcevic et al. (1992). The application of DNS for evaluating riblet performance in turbulent flow was addressed by Benschop and Breugem (2017), El-Samni et al. (2007), and Okabayashi et al. (2018).

One of the numerical studies of the combined moment and heat transfer over riblet surfaces was done by Benhalilou and Kasagi (1999). They adopted a nonlinear low-Reynolds number $k-\varepsilon$ model, predicting mean secondary flow. The model predicts fluids with a great range of molecular Prandtl numbers. Their numerical result for drag had accepted accuracy compared to experimental results; however, their heat transfer result seemed to contradict the former experimental results. Aus der Wiesche (2003) adopted LES to study turbulent flow and heat transfer over riblet surfaces. They declared the computational domain of this study limited the results to small Prandtl number fluids ($Pr < 0.01$). Results of the drag study were in good agreement with the DNS results published by Chu and Karniadakis (1993), and Choi et al. (1993), and experimental results published by Walsh (1982), but their heat transfer results only compared to theoretical data. Stalio

and Nobile (2003) applied DNS to investigate heat transfer over riblet surfaces. Their thermal results disagreed with experimental results published by Walsh and Weinstein (1979), and Choi and Orchard (1997). Stalio and Nobile (2003) postulated the discrepancy may have been due to some secondary effects they had not considered in their simulations or because of experimental errors. Jin and Herwig (2014) also did a DNS study on overheated riblet surfaces.

3.5.1 Numerical Studies of Drag Reduction

Riblet studies of drag reduction have been conducted in both closed and open channel flow simulations. Closed channel studies indicate drag reduction using riblets in an infinitive plane channel bounded by two horizontal walls. Open channel simulations indicate drag reduction by applying riblets on a flat plate. Table 3.3 summarizes drag reduction studies using different simulation methods conducted in closed and open channels and their subsequent riblet design, configuration, and placement.

Table 3.3. Numerical studies in closed and open channels with riblets

Channel type	Riblet			CFD approach	Drag reduction	Reference
	Placement	Profile	Configuration			
Close	On bottom wall	Blade	Continuous	DNS	11%	El-Samni et al. (2007)
		Blade	Continuous	RANS/RSM	12.43%	Benhamza and Belaid (2009)
		Blade	Continuous, inline, and staggered segmented	LES	11%	Martin and Bhushan (2016a)
		Blade, saw-tooth, and scalloped	Continuous and inline segmented	LES	11.6%	Martin and Bhushan (2016b)
		Blade	Continuous	LES	11.7%	Martin and Bhushan (2014)
		Saw-tooth	Continuous	DNS	5-6%	Choi et al. (1993)
	On bottom and top walls	Blade, saw-tooth, and scalloped	Continuous	RANS/ $k\text{-}\epsilon$ eddy-viscosity model	11%	Launder and Li (1993)
		Saw-tooth	Zigzag	DNS	3.5%	Okabayashi et al. (2018)
	On bottom and top walls, with and without a shift	Blade	Continuous longitudinal and in yaw, and herringbone	DNS	9.3%	Benschop and Breugem (2017)
		Saw-tooth and scalloped	Continuous longitudinal and in yaw	RANS / SA, $k\text{-}\epsilon$ BSL, and SST	12%	Aupoix et al. (2012)
Saw-tooth and scalloped		Continuous	Viscous wall-region model	?	Tullis and Pollard (1994)	
Trapezoidal		Continuous	LES	9.8%	Bannier et al. (2016)	

The application of riblets on the bottom wall of a closed channel was numerically considered in different studies. Benhamza and Belaid (2009), Choi et al. (1993), Martin and Bhushan (2014), and El-Samni et al. (2007) applied continuous riblet configurations and achieved

12.43%, 5-6%, 11.7%, and 11% of drag reduction, respectively. Martin and Bhushan (2016a,b) modeled the segmented riblet configuration in addition to continuous riblet configuration and achieved 11% and 11.6% of drag reduction, respectively. The application of riblets on the top wall was studied by Martin and Bhushan (2016a). They showed that d^+ has a diameter of 30-40 non-dimensional units on a smooth surface; whether the riblet can lift large scale vortices or not reasonably depends on riblet s^+ and h^+ in a continuous configuration. The turbulent vorticity interaction with a smooth surface and a riblet surface at two different s^+ is shown in Figure 3.3.

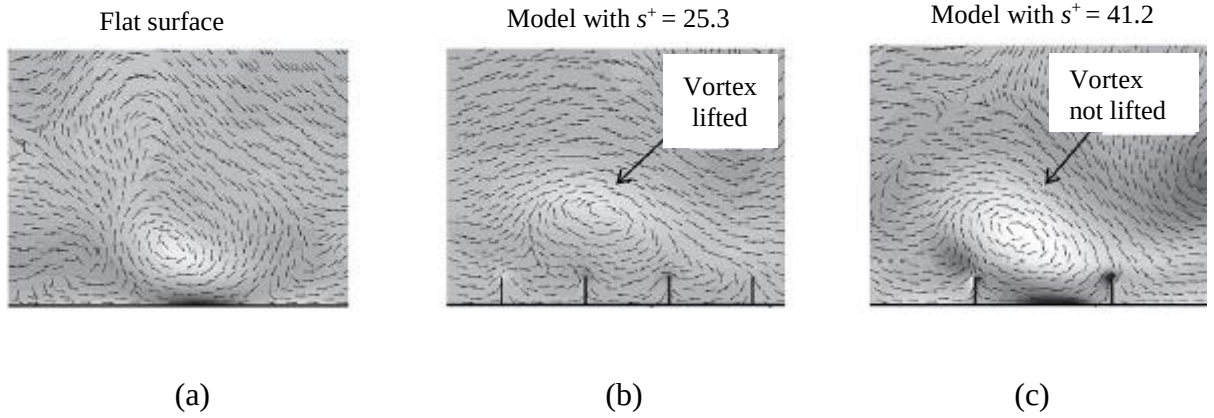


Figure 3.3. Integration of overlying turbulent vorticity with (a) a smooth surface and blade riblets at two different (b) $s^+ = 25.3$ and $s^+ = 41.2$ at $Re = 4,180$ (Martin and Bushhan, 2016a)

Accordingly, a riblet surface with too large s^+ does not lift the large scale vorticities up from the surface, and they settle within the riblet valleys leading to drag increase. In addition, riblets with too high h^+ caused excessive protrusion into the flow, while riblets with too short a height could not lift up the large scale vortices. Again, referencing the same authors, j_r^+ in inline segmented configurations should be shorter than Le^+ so as not to allow residing vorticity in the longitudinal riblet gaps, as illustrated in Figure 3.4.

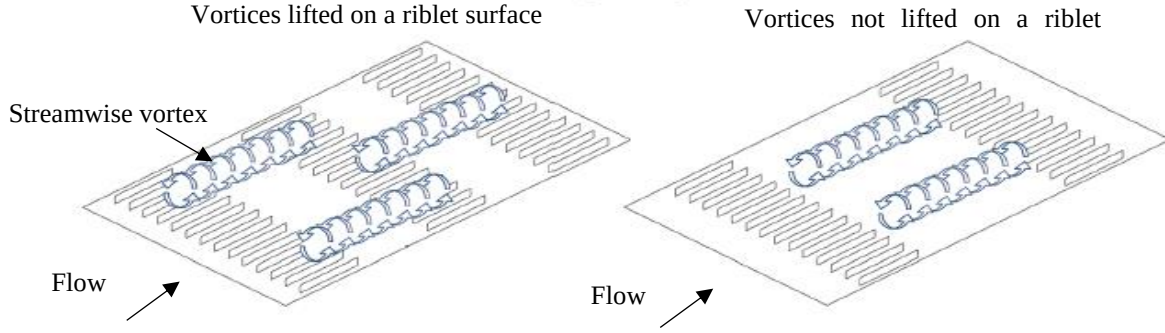


Figure 3.4. Conceptual model of vorticity behavior in inline staggered configuration considering two different j_r^+ (Martin and Bushhan, 2016a).

The application of riblets on the bottom and top wall of a close channel was numerically studied by Launder and Li (1993), who achieved maximum drag reduction of 11% for continuous riblets, and Okabayashi et al. (2018), who applied zigzag riblets and achieved 3.5% of drag reduction. Conceptual model of vorticity behavior in inline staggered configuration considering two different j_r^+ (Martin and Bushhan, 2016a).

Benschop and Breugem (2017) considered placement of different riblet configurations such as continuous longitudinal, continuous in yaw, and herringbone on the bottom and top walls of a closed channel. In addition, they considered two configuration variations for the herringbone called standard and shifted herringbone. They gained a maximum drag reduction of 9.3% and results showed that a stronger mean secondary flow exists in cases with the span-wise shift of the top wall riblets compared to the bottom wall riblets to the aligned riblets on the bottom and top walls. The secondary flow near the channel centerline does not affect drag reduction. Hence, the span-wise shift of the top wall riblets compared to the bottom wall riblets does not contribute to drag reduction.

Continuous riblet configuration in an open channel was numerically modeled by Tullis and Pollard (1994), Aupoix et al. (2012), and Bannier et al. (2016). Aupoix et al. (2012) achieved the

highest drag reduction of 12% among other related studies. They also considered continuous in yaw riblet configuration in their study.

3.5.2 Numerical Heat Transfer Studies

Only a few studies have attempted to simulate a thermal field in a fully developed turbulent flow over riblet surfaces up to now. Benhalilou and Kasagi (1999) studied the effects of both molecular Prandtl number ($0.01 \leq Pr \leq 100$) and Reynolds number ($3,000 \leq Re \leq 20,000$) on the thermo-hydraulic efficiency index for two types of saw-tooth riblets using a nonlinear $k-\varepsilon$ model. Figure 3.5 shows efficiency index clearly increases as the Prandtl number and Reynolds number increase. They achieved 20% enhancement in the thermo-hydraulic efficiency index for a fluid at $Re = 5600$ and $Pr = 100$. They demonstrated that the thermo-hydraulic efficiency index could be augmented beyond a smooth surface only for flow with high molecular Prandtl numbers.

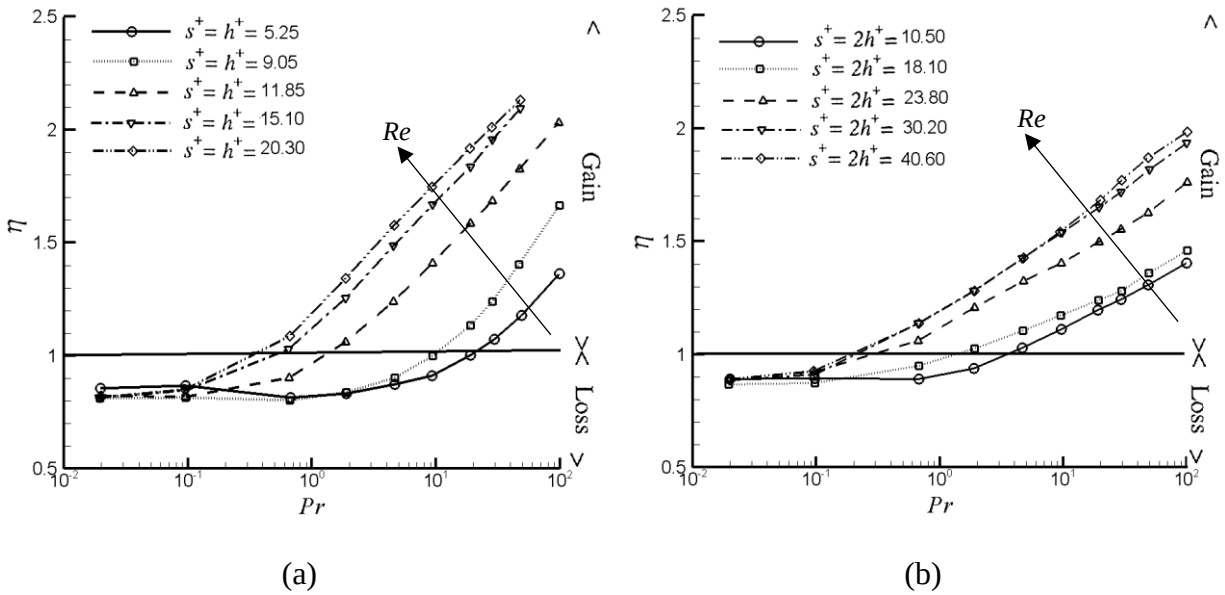


Figure 3.5. Thermo-hydraulic efficiency index over riblet surfaces at geometrical parameters of (a) $h/s = 1$ and (b) $h/s = 0.5$ considering different molecular Prandtl and Reynolds numbers (data from Benhalilou and Kasagi (1999))

Stalio and Nobile (2003) simulated fully developed turbulent flow and heat transfer over riblet surfaces using DNS in an airflow channel at a friction Reynolds number of $Re_\tau = 180$, based on skin friction velocity. The channel contained parallel plates with riblets on the bottom, and constant heat flux was applied to both plates. Two continuous saw-tooth riblets with $s^+ = 20$ and $s^+ = 40$ were considered. The result showed the thermo-hydraulic efficiency index is less than one ($\eta < 1$) for all considered riblet geometries, which means no HTE was gained. Aus der Wiesche (2003) investigated the effects of small Prandtl numbers ($0.001 < Pr < 0.01$) on HTE in a turbulent flow channel with parallel plates and mounted continuous saw-tooth riblets on the bottom wall using LES. The channel and riblet dimensions were the same as those studied by Stalio and Nobile (2003). Results showed the thermo-hydraulic efficiency index could be enhanced up to 22% within a drag-reducing regime of riblets for a flow with $Pr = 0.01$ and at $Re = 4,040$. Jin and Herwig (2014) employed different riblets with blade and trapezoidal profiles and applied them to an airflow channel with parallel plates with riblets on the bottom and top walls. They investigated the entropy generation rate, drag reduction, and HTE of the riblet surfaces at three Reynolds numbers — $Re = 2,800$ (low), $Re = 5,600$ (medium), and $Re = 8,000$ (high), applying DNS. Results showed heat transfer reduces as drag reduces for most riblet surfaces except one case. The one that was able to enhance the *Nusselt number* over a smooth surface to 0.7% and reduce drag to 1%, had a blade profile and was computed at $Re = 8,000$.

3.6 Riblet Optimization

A complex optimization problem is a determination of riblet design variables for a given closed or open channel flow under constraints with the goals of achieving lower drag and maximum heat transfer. Optimization based on cross-section profiles with the objective of drag

reduction is summarized in the studies of Launder and Li (1993), Bechert et al. (1997), Martin and Bhushan (2016b), and Bechert et al. (2000). For instance, Bechert et al. (1997) experimentally showed that continuous blade riblets maximize drag reduction in an open oil channel compared to saw-tooth and trapezoidal riblets as shown in Figure 3.6. Martin and Bhushan (2016b) also numerically proved the advantage of blade riblets compared to saw-tooth and scalloped riblets in a closed channel.

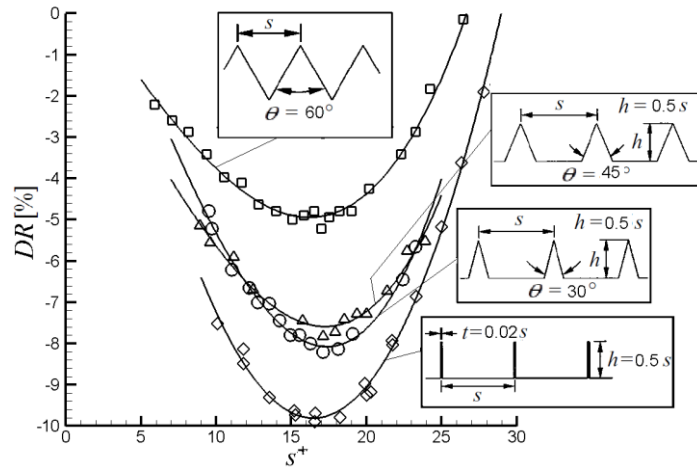


Figure 3.6. Optimization of riblet cross-section profile (data from Bechert et al. (1997))

The most important design variables identified in the literature for optimization are s^+ , h^+ , and t^+ . There are two ways to determine the mentioned optimal variables: 1) varying the dimensions of s , h , and t for a constant-flow velocity, and 2) varying velocity and keeping dimensions constant. Martin and Bhushan (2016) numerically showed the optimum s^+ and h^+ are within the range of 15-20 and 8-10, respectively.

The geometric parameters of h/s and t/s ratios play a key role in obtaining optimum drag reduction. Figure 3.7 shows the effect of s^+ on drag reduction at various h/s and t/s ratios for blade riblets using oil flow. According to the figure, the breakdown point for optimum drag reduction occurred around $s^+=16$ when h/s is 0.5. The optimum t/s shows riblet type as 0.02 with improving

alignment and leakage, although it was better for the comparison to keep the fabrication procedure equal for riblet surfaces in Figure 3.7.

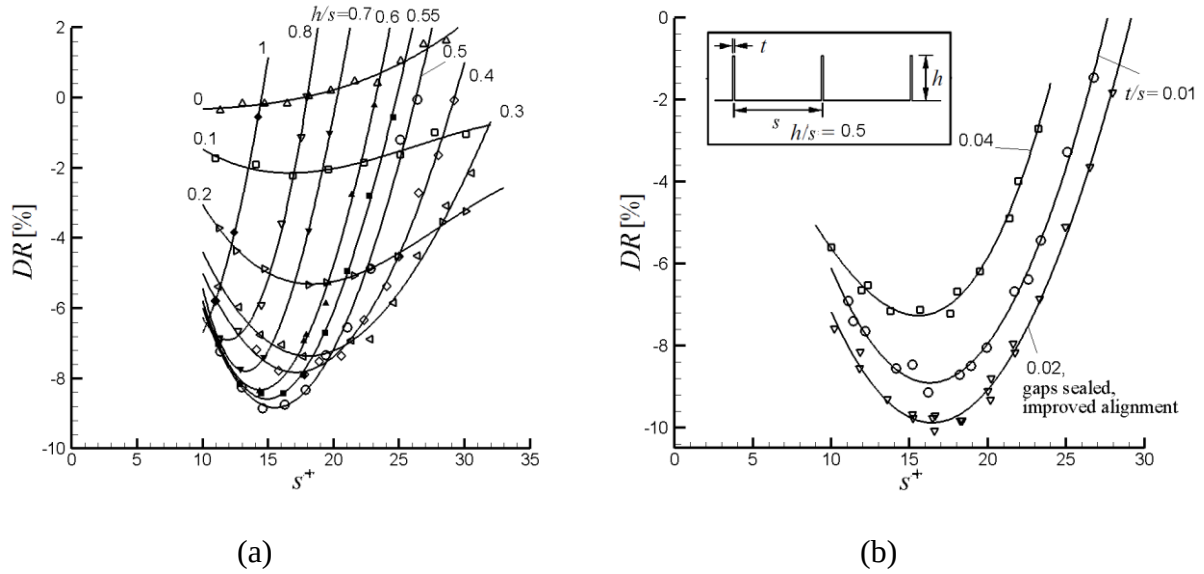


Figure 3.7. Effect of s^+ on drag reduction at different (a) h/s and (b) t/s ratios for blade riblets in an open oil-flow channel (data from Bechert et al. (1997))

Garcia-Mayoral and Jiménez (2011) reported a new dimensionless scale of $l_g^+ = (A_g^+)^{0.5}$ in which A_g^+ is a dimensionless valley cross-section. They numerically showed the optimum value of l_g^+ has less scatter around the optimum l_g^+ compared to the optimum values of s^+ and h^+ . Correspondingly, the optimum l_g^+ was found within the range of 9.7-11.7 for different riblet cross-section profiles.

The effect of riblet configuration on drag reduction has been numerically and experimentally studied by Martin and Bhushan (2016b), Martin and Bhushan (2016a), Benschop and Breugem (2017), Bechert et al. (2000), and Bixler and Bhushan (2013a). Comparing the experimental results of Bechert et al. (1997, 2000), which conducted oil flow in an open channel, revealed that continuous riblet configuration had greater drag reduction than a staggered

segmented configuration when s^+ is close to its optimum value. Martin and Bhushan (2016a) also reported 10% of maximum drag reduction with water flow for continuous blade riblets, which was much greater than the optimum drag reduction of 5% and 3% corresponded to inline segmented and staggered segmented configurations, respectively. In contrast, the effect of continuous and segmented configurations on drag reduction in a micro-sized channel using oil and water showed that segmented riblets provided better drag reduction than continuous ones within a great range of flow velocity (Bixler and Bhushan, 2013a). The optimization of j_r^+ and l_r^+ applied in staggered configuration was done by Martin and Bhushan (2016a). They showed that as l_r^+ increases, the segmented configuration approaches continuous configuration and drag decreases. In addition, drag reduction remains approximately constant by increasing j_r^+ until it becomes greater than L_e^+ , which causes a great drag increase.

Limited optimization research is available in the literature with the aim of HTE within a drag-reducing regime of riblets. Only a few studies considered the effect of riblet cross-section profiles on HTE (Walsh and Weinstein, 1979; Jin and Herwig, 2014). Walsh and Weinstein (1979) declared that saw-tooth riblets have better performance to enhance heat transfer compared to blade and trapezoidal riblets. The effect of s^+ on HTE by riblets has been studied by Choi et al. (1993) and Choi and Hamid (1991).

Unlike riblet studies, many researchers have dealt with finding the best design for HTE surfaces in fin studies, such as fins that optimize heat transfer and pressure drop leading to maximizing the thermo-hydraulic efficiency index. Fin studies are not within the drag-reducing regime because of fin dimensions and mechanisms, but the fin optimization procedure and study objectives, which maximize heat transfer with a minimum pressure drop penalty, are similar to riblet studies. Thus, it would be beneficial to take advantage of fin studies to extend riblet

optimization studies. Webb et al. (1971) experimentally investigated the effect of fin design variables such as fin height and lateral spacing on the thermo-hydraulic efficiency index in a circular airflow. Webb et al. (1999) included the impact of the helix angle on fin design study to enhance the thermo-hydraulic efficiency index.

All design optimization studies that have been published with the objective function of either drag reduction or HTE are mainly based on meta-heuristic and evolutionary algorithms. These algorithms are based on computational and search methods influenced by artificial intelligence (AI) and natural systems. They provide excellent approximating solutions to many types of problems, such as multi-objective problems (Mann and Eckels, 2019; Copiello and Fabbri, 2009; Soleimani et al., 2016; Bozorg-Haddad et al. 2017; Juan and Qin, 2014). The algorithms are problem-independent methods suitable for optimization problems unworkable with other optimization techniques (linear programming, non-linear programming, and dynamic programming). In meta-heuristic and evolutionary algorithms, the objective function is calculated through a sequence of feasible alternative variables that determine the optimal (maximum or minimum) value of the objective function. Multi-objective optimization using evolutionary algorithms is employed to attain a set of Pareto optimal solutions. In a Pareto front, no objective can be improved without sacrificing at least one other objective.

Although CFD-based simulation models have great capabilities to simulate complex flow physics and quantitatively evaluate the system's response, they cannot find the best design solution. In other words, simulation models are mainly based on the question: what is likely to happen if a particular design is implemented? However, control of the thermo-hydraulic efficiency index could only be improved by more fundamental research methods, combining the implementation

of simulation and optimization methods that initiate a practical approach to determine riblet design variables to achieve the best performance in HTE applications.

Multi-objective evolutionary optimization algorithms such as a common MOGA are able to couple with CFD-based simulation models to obtain optimal solutions employing systematic search. In this way, every evaluation of objective function requires the solution of a simulation model whose design variables vary in the optimization procedure. Copiello and Fabbri (2009) studied the simulation-optimization application on fin design, and more recently by Mann and Eckels (2019). Time-consuming and complex calculation of objective function by CFD-based simulation models in every evaluation step may become too cumbersome in some cases, which is a key limitation of the CFD-based simulation-optimization models. To speed up the simulation-optimization technique, Juan and Qin (2014) applied some sort of correlation to evaluate objective function to optimize fin-and-tube, heat-exchanger design variables.

3-8- Summary and Outlook

Riblets are engineered-based extensive surfaces known as a passive control technique for thermal turbulent flow systems. Riblet surfaces are great prospective candidates for HTE applications, which deal with low drag and highly enhanced heat transfer surfaces. Riblets can reduce drag by lifting up large-scale turbulent vortices from the near-wall region and controlling turbulent coherent structures that contribute to transverse shear stress. An acceptable number of experimental and numerical studies with a great range of design variables focused on drag-riblet-reducing abilities in micro-sized or macro-sized closed channels (internal) and open-channel (external) fluid flow. However, attempts to investigate riblet surfaces as heat enhancement surfaces has ended with little experimental research results that were mostly conducted in a wind tunnel using air flow and little numerical research.

The turbulent and thermal boundary layers keep developing over the length of the surface in an external flow, whereas these layers are restricted by channel size and turbulent vorticity interaction from surrounding walls, different from the internal flow. These dissimilarities necessitate investigation into drag-reducing and HTE properties of riblet surfaces for open and closed channel flow, as well as investigating exchange interactions between the scales of thermal and fluid mechanisms.

Numerical studies clearly showed that RANS-based simulations had the least agreement with experimental data compared to other CFD-based models. LES can greatly predict near-wall motions but the results obtained must be validated with experimental data in cases including riblets and not only smooth surfaces. More studies with validated results are required to discover the efficiency of the performance of LES. Although DNS nicely predicts drag reduction and captures near-wall motions, it greatly increases computational cost and is not suitable for complicated geometries.

Riblet design variables, including cross-section profiles, dimensions, and configurations greatly affect riblet surface performance. In addition, the micro-sized, closed-channel placement of riblets on the bottom, top, or both channel side walls changes the performance of riblet surfaces and channel height dimensions. The result of the applicability of riblets in a circular closed channel demonstrated the benefit of riblets to drag reduction, but to further understand the possible benefit of riblets in rectangular channels, more studies should be experimentally and numerically conducted, including various channel dimensions.

Despite decades of studies of roughness design, researchers have not succeeded in providing optimum drag reduction or an enhanced heat-transfer surface. No current research clearly explains the science behind the high thermo-hydraulic efficiency index in riblet surfaces.

In addition, no one has improved the thermo-hydraulic efficiency index using optimization algorithms, and specifically employing the multi-objective optimization of reducing drag and enhancing heat transfer in riblet studies. If illuminated, it would enable an entirely new level of engineering designs affecting all heat-transfer applications.

4 Numerical Model Setup

This study develops a three-dimensional conductive and convective heat-transfer simulation through coupled solid and fluid domains applied in a commercial package, ANSYS Fluent 19.1, under the following assumptions: (1) the single-phase flow is 3D, incompressible, steady-state, fully developed, and turbulent; (2) natural convection and radiation heat transfer are disregarded; and (3) the thermodynamics and transport properties are constant.

4.1 Governing Equations and Parameters

The simulation model requires the solution of the continuity equation, Navier-Stokes equations, and energy equation. The conservation of the governing equations can be expressed as follows:

$$\nabla \cdot \mathbf{u} = 0 \quad (4.1)$$

$$\rho \mathbf{u} \cdot \nabla \mathbf{u} = \rho \mathbf{g} - \nabla p + \mu \nabla^2 \mathbf{u} \quad (4.2)$$

$$\rho c_p (\mathbf{u} \cdot \nabla T) = K \nabla^2 T + \mu \left[\nabla \mathbf{u} + (\nabla \mathbf{u})^T \right] : \nabla \mathbf{u} \quad (4.3)$$

in which, $\mathbf{u}$ is the velocity vector, T is temperature, and g is the gravitational acceleration.

The RANS technique utilizes an averaging method leading to a time-average solution of the Navier-Stokes. The technique applies a turbulence closure model, which uses a turbulent viscosity approach in the Reynolds stress formulation. This study employs RANS with a k - ε turbulence model for modeling the fluid flow motion. The k - ε model first proposed by Launder and Spalding (1974) contains two transport equations that derive the turbulence using the turbulent kinetic energy generation rate (k) and turbulent kinetic energy dissipation rate (ε). The assumption

of the $k-\varepsilon$ model is that flow is fully turbulent and turbulence viscosity is isotropic, which means the ratio between Reynolds stress and mean rate of deformations is the same in all directions.

The CFD Flow solver applies a pressure-based method. The pressures-based method is more coordinated with the incompressible flow. Continuity and momentum equations are modified to extract a pressure correction equation that leads to a pressure field solution. A pressure-velocity coupling equation is derived by discretizing Navier-Stokes equations which shows a relationship between velocity and pressure correction. The procedure models and solves the governing equations based on the finite-volume method in turbulence flow.

Table 4.1 shows the values of thermodynamics and transport properties used in the computational model.

Table 4.1. Thermodynamic parameters

Parameter	Fluid (water)	Solid (copper)	Unit
ρ	998.2	8,978	kg m ⁻³
μ	0.001003		kg m ⁻¹ s
K	0.6	387.6	W m ⁻¹ K ⁻¹
c_p	4,182	381	J kg ⁻¹ K ⁻¹
Pr	6.99		

4.2 Discretization and Convergence

The pressure-velocity coupling problem is solved using a coupled scheme and the pseudo transient method since the coupled scheme is generally preferred for transient and periodic simulations due to its convergence behavior. The discretization for pressure is second-order, momentum and energy equations are second-order upwind, and turbulence parameters of kinetic energy and dissipation rate were first-order upwind. Convergence for momentum and energy equations are monitored by periodic pressure gradient changes from the periodic domain and total heat flux from the solid-fluid interface using the default ANSYS Fluent stop criterion scheme with

stop criteria of 10^{-5} over the previous ten iterations. The fluid inlet's maximum stream-wise velocity magnitude is monitored similarly to ensure the simulation is in the fully developed region. Besides, convergence criterion of 10^{-5} is defined for variables of the continuity; velocity in x , y , and z directions; kinetic energy; and dissipation rate, while convergence criterion of 10^{-6} is designated for energy.

4.3 Boundary Conditions

This study aims to simulate fully developed flow in the computational flow domain, but the length required to simulate this leads to high computational costs. Hence, in the current study, translationally periodic conditions are applied to obtain a solution in the fully developed region of the flow. According to the periodic condition, it is required that the flow pattern be restricted to a single module and repeats along the direction of the flow, while that flow needs to be incompressible. Due to the heat transfer periodic condition, the wall thermal boundary condition must be of constant heat flux or constant wall temperature. A periodic thermally developed regime leads to having a stable heat transfer coefficient along with the flow. The domain is solved repeated times as flow over the small region evolves to the fully developed solution. Periodicity signifies a repetition of velocity field over length L and can be expressed as:

$$\mathbf{u}(x, y, z) = \mathbf{u}(x + L, y, z) \quad (4.4)$$

in which, x , y , and z are stream-wise, wall-normal direction, and transverse coordinates. Periodicity in the pressure field is different than the velocity field. In this case, a pressure drop is defined as periodic instead of the pressure, and the pressure drop across each module is constant. At first, the pressure field is divided into two terms, as shown in Equation (4.5).

$$p(x, y, z) = -\beta \cdot x + P(x, y, z) \quad (4.5)$$

$$P(x, y, z) = P(x + L, y, z) \quad (4.6)$$

in which, β is a linearly subtracting term for the pressure, and it is solved as a volume force to satisfy the required mass flow in the momentum equations. The $P(x,y,z)$ term is the periodic spatially pressure term related to local motions. Hence, the momentum equation in the x-direction can be rewritten as:

$$\rho \mathbf{u} \cdot \nabla \mathbf{u} = \rho g - \frac{\partial P}{\partial x} + \mu \nabla^2 \mathbf{u} + \beta. \quad (4.7)$$

A similar method to the pressure field can be employed for the temperature field. Therefore, the periodic thermally developed field is achieved by defining a non-dimensional temperature variable as below:

$$\theta_t = \frac{T(x, y, z) - T_w}{T_{in} - T_w} \quad (4.8)$$

in which, θ_t is the non-dimensional temperature variable. Since there is a difference between the temperatures at the inlet and outlet in the periodic domain, Patankar et al. (1977) calculated the ratio between inlet and outlet bulk temperature as below:

$$\lambda_t = \frac{T_{in} - T_w}{T_o - T_w} \quad (4.9)$$

in which, T_o is the fluid temperature at $x = L$ or the module outlet. Therefore, the thermal periodicity condition can be written as follows:

$$\theta_t(x = 0, y, z) = \theta_t(x = L, y, z) \cdot \lambda_t. \quad (4.10)$$

The simulation utilized a constant and uniform wall temperature at the outer wall of the solid surface. A specified mass flow rate and inlet bulk temperature are defined at the flow inlet. Periodic boundary conditions make all physical values periodic at each time step of solving. As a

result, inlet and outlet boundary values such as velocity, temperature, and pressure evolve with each time step, converging after many iterations based on the defined mass flow rate.

4.4 Data Reduction

The key parameters of interest from the simulation result are the Fanning friction factor and average Nusselt number. These parameters were derived from the simulation results in the following way.

The average heat transfer coefficient is calculated as follows:

$$h_{co} = \frac{q}{LMTD} \frac{A_{en}}{A_s} \quad (4.11)$$

in which A_{en} is the true or enhanced surface area; A_s is the root or smooth surface area, and $LMTD$ is an abbreviation of logarithmic mean temperature difference, which is calculated as below:

$$LMTD = \frac{(T_w - T_{in}) - (T_w - T_o)}{\ln \left(\frac{T_w - T_{in}}{T_w - T_o} \right)}. \quad (4.12)$$

The outlet fluid temperature is calculated as follows:

$$T_o = \frac{qA_{en}}{\dot{m}c_p} + T_{in}. \quad (4.13)$$

With the value of heat transfer coefficient, the Colburn j factor and Nusselt number are calculated as:

$$j = \frac{h_{co}}{\rho V c_p} Pr^{2/3} \quad (4.14)$$

$$Nu = \frac{h_{co} D_i}{K} \quad (4.15)$$

where, V is inlet velocity. The effect of surface geometry on wall shear stress can be determined from the pressure different of flow inlet and outlet when the flow is fully developed, and the mean velocity is constant. The shear stress in an enhanced tube is calculated as follows:

$$\tau = \frac{dp}{dx} \frac{D_i}{4} \quad (4.16)$$

In a channel that includes bottom enhanced wall, top smooth wall, and periodic or symmetry side-walls, the following procedure will be followed to separate the effect of the enhanced side of the wall. First, the simulation result reports the pressure gradient and shear stress of the top wall. For the same bulk fluid velocity, the wall shear stress on the bottom of the channel can then be calculated by subtracting the effect top wall of the channel with the assumption that the bottom wall enhancement changes the shear stress on the top wall. The expression for the bottom shear stress is:

$$\tau_{en,w} = \frac{dp}{dx} D_i - \tau_{s,w} \quad (4.17)$$

where, $\tau_{en,w}$ and $\tau_{s,w}$ is the shear stress of the enhanced and smooth walls of the channel, respectively. In a channel that includes enhanced bottom and top walls with periodic or symmetry side-walls, the shear stress is calculated as:

$$\tau = \frac{dp}{dx} \frac{D_i}{2} \quad (4.18)$$

At last, the Fanning friction factor is calculated using Equation (4.19).

$$f = \frac{\tau}{0.5\rho V^2} \tag{4.19}$$

5 Parametric Study of Transverse and Axial 2D Micro-fins on Fluid Flow and Heat Transfer

Turbulent flow and heat transfer characteristics have been numerically studied in a turbulent channel flow with micro-fins on the lower wall, as shown in Figure 5.1. Studies of flow fields around fins have appeared in the literature but often do not include conduction analysis in the fin, leaving a key optimization parameter unexplored. This study aims to fill this gap by finding out the effects of micro-fin design variables on fluid drag increase and HTE by holding the micro-fin volume constant over the numerical space. Design variables of interest are micro-fin cross-sectional profile, height, yaw angle, and pitch. A coupled numerical simulation of the turbulent flow at Reynolds number 14,825, with periodic boundaries and solid domains, was set up and validated in ANSYS Fluent, and the performance was mapped. In addition, the shift in the logarithmic portion of law of the wall for surfaces with micro-fins is discussed in terms of roughness function and roughness Reynolds number.

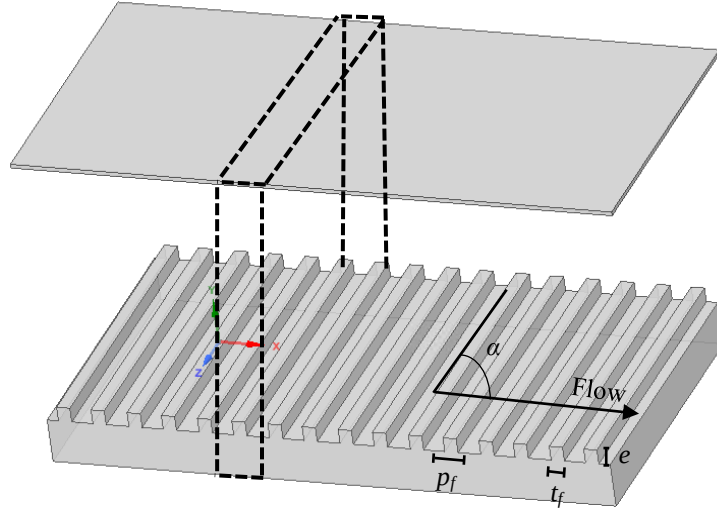


Figure 5.1. A schematic of a single channel with transverse micro-fins on the lower wall

5.1 Scenario Description

This study defines four different micro-fin cross-sectional profiles (rectangular, trapezoidal, triangular, and parabola) at six different heights where the remaining geometric parameters such as micro-fin thickness are varied to keep fin volume constant. Two different yaw angles are considered: transverse (90 yaw angle) and axial (0 yaw angle) micro-fins. Figure 5.1 illustrates the micro-fin parameters, and Table 5.1 summarizes the different scenarios. Six different scenarios are defined based on the micro-fin heights so that each scenario contains four different cross-sectional profiles and two different yaw angles. Consequently, 48 micro-finned channels with a constant $p_f = 2$ mm cases are defined first. All cases have a constant micro-fin cross-sectional area of 0.08 mm^2 and a constant solid volume.

The solid cross-sectional area of the smooth case is considered as if the fins are melted, and materials returned to the solid volume. Next, the effect of the micro-fin pitch parameter on the

thermo-hydraulic efficiency index is considered by looking at the micro-fin pitch between 0.3 to 3 mm.

Table 5.1. Scenario description for different micro-fin geometries

Scenario number	e (mm)	t_f (mm)			
		Rectangle	Trapezoid	Triangle	Parabola
1	0.25	0.32	0.52	0.64	0.96
2	0.30	0.27	0.41	0.53	0.80
3	0.35	0.23	0.33	0.46	0.69
4	0.40	0.20	0.28	0.40	0.60
5	0.45	0.18	0.23	0.36	0.53
6	0.50	0.16	0.20	0.32	0.48

5.2 Geometry and Boundary Condition

The 3D geometry of the fluid-flow system contains a channel flow heat exchanger, and the lower wall includes a solid base where the micro-fins are placed. Figure 5.2 shows the geometry of the periodic domain for an infinitive channel flow with a solid base, including a rectangular micro-fin. The dash lines in Figure 5.1 represent the periodic domain as well. In Figure 5.2, solid height is 2 mm, and the width, length, and height of the channel section are 2, 2, and 15 mm, respectively. Line 1 in Figure 5.2 is the location where numerical results are compared. All scenarios of the trapezoidal cross-sectional profile have a constant upper width of 0.125 mm and all triangular trapezoidal and parabola shapes are isosceles.

The problem is defined as fluid enters the channel with constant velocity ($V = 3.16 \text{ m s}^{-1}$) with entering temperature of $T_{in} = 20 \text{ C}^\circ$. A constant and uniform wall temperature of $T_s = 40 \text{ C}^\circ$ is defined at the outer wall of the solid surface, as shown in Figure 5.2.

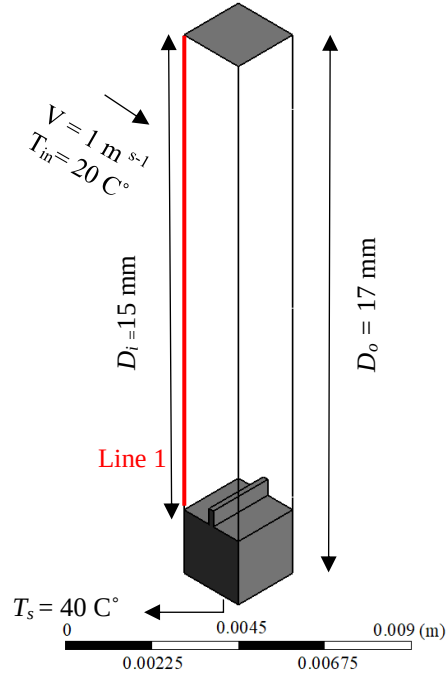


Figure 5.2. Model geometry for a rectangular micro-fin profile

5.3 Meshing and Independence Study

Meshing is a discrete representation of geometry. A hybrid technique was applied to generate the mesh. Small portions of the grid situated around the micro-fin are defined as unstructured, and the remaining parts of the grid are defined as structured. Figure 5.3 shows the hybrid mesh for a case with an axial rectangular micro-fin. Mesh size is one of the most critical issues to consider when validating the simulation against experimental results. Specifically, a mesh independence analysis is required to prove convergence in the results, adding confidence that the mesh is fine enough to capture the most complex details of the flow. The mesh independence analysis is shown in the results section.

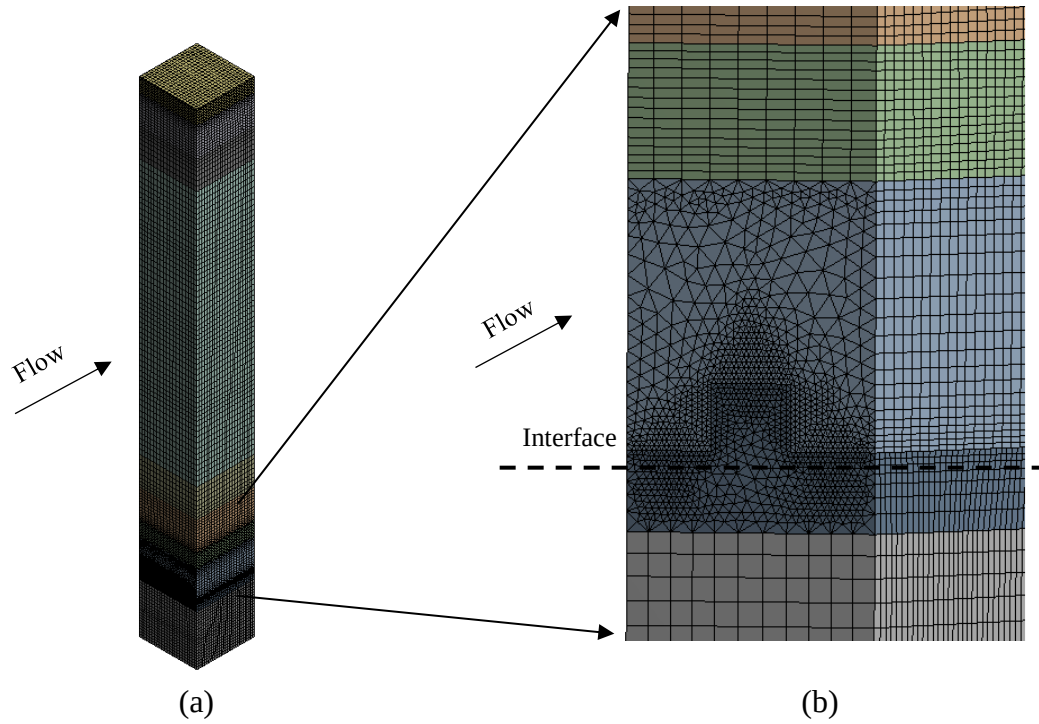
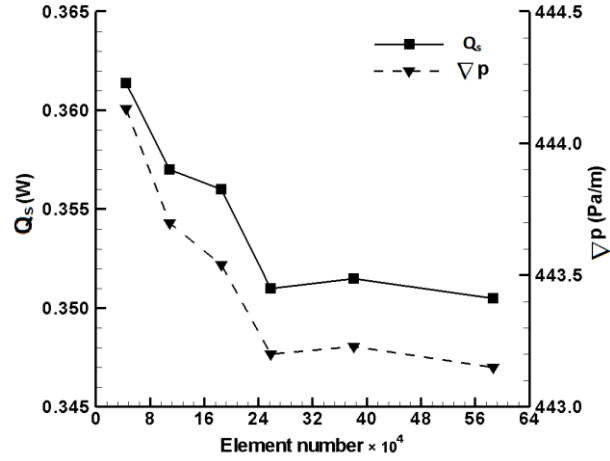
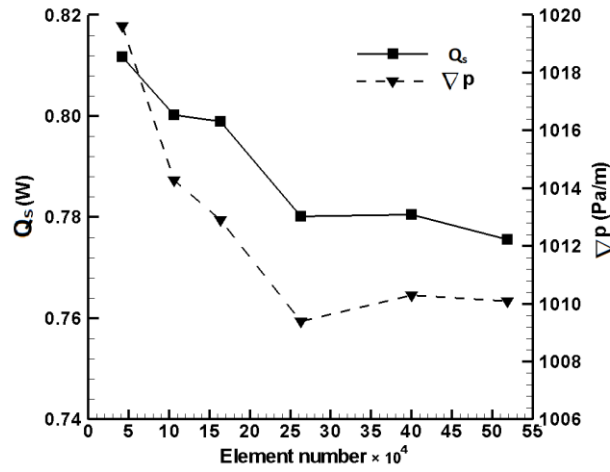


Figure 5.3. The grid system of (a) whole domain and (b) zoom-in view at the solid-fluid interface for a case with a rectangular micro-fin.

To investigate a proper grid size for the numerical simulation, mesh independency analysis was carried out for both the smooth case and a transverse micro-finned case. Figure 5.4 shows the results of the total heat-transfer rate and pressure gradient for different mesh resolutions of the smooth case and a case with a transverse rectangular micro-fin. It is observed that the total heat-transfer rate and pressure gradient approach constant values with refining the mesh in both cases. According to the figure, it is concluded that the grids having 257,440 elements for the smooth case and grids having 263,006 elements for the micro-fin case are fine enough to capture the detailed information of flow and temperature fields. The selected grids are applied for all subsequent numerical calculations.



(a)



(b)

Figure 5.4. Total heat-transfer rate and pressure gradient for different mesh resolutions of the (a) smooth case and (b) a case with a transverse rectangular micro-fin

5.4 Model Validation

The numerical results for the smooth case were compared to experimental data published by Li and Eckels (2017) and Li et al. (2018). Table 5.2 compares results for numerical and experimental data for the smooth case. The relative error for the heat-transfer coefficient and Darcy friction factor between numerical and experimental data is about 6.7% and 1.5%, respectively,

which is acceptable for the goal of this study. It should be noted that this study calculated the experimental Darcy friction factor based on data published by Li and Eckels (2017). The micro-fin simulation was validated against the experimental data published by Drain and Martin (1985). The validation used fin-design variables defined by Drain and Martin (1985) while keeping the current studies' computational domain, meshing techniques, and numerical model. Figure 5.5 shows the stream-wise mean-velocity profile for numerical data at line 1 in Figure 5.5, as well as experimental data for the smooth case and a case with transverse fins. Results are in good agreement in both numerical and experimental analysis for smooth and micro-finned cases.

Table 5.2. Comparison of experimental and numerical results for the smooth case

Variable	Experimental	Numerical
V (m/s)	0.9	0.90
Darcy friction factor (f_d)	0.0271	0.0275
h_{co} (W/m ² .K)	3521	3283

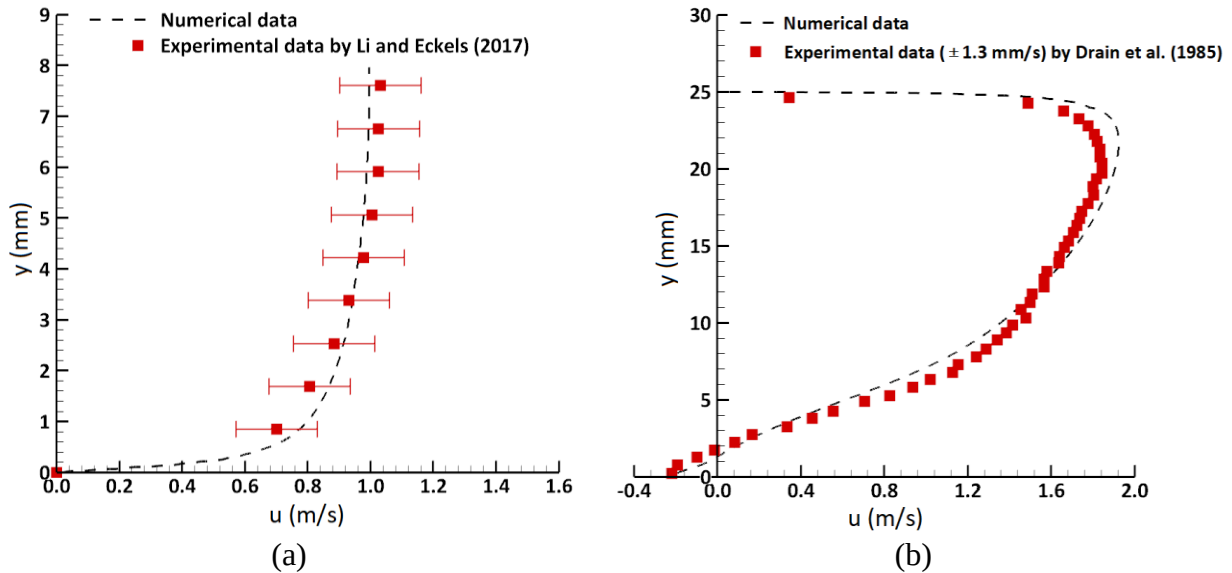


Figure 5.5. Comparison of numerical and experimental results of stream-wise mean-velocity profiles for (a) the smooth case and (b) a case with transverse fins.

5.5 Results and Discussion

5.5.1 Velocity and Thermal Fields Around the Micro-fins

The flow field around the fins is important in understanding the final pressure drop and heat transfer results. Figure 5.6 displays velocity streamlines around Scenario 6 of transverse micro-fins for different profiles at $p_f = 2$ mm. The flow separates at the upstream tip of the micro-fins, which causes the two-dimensional stable vortices with the same size of the enclosure between two micro-fins for all profiles. These vortices prevent the outer flow from reattaching to the surface of the enclosure. The formation and center of the vortices vary with the micro-fin cross-sectional profile. For instance, the center of vorticity is placed in the middle of the enclosure in the rectangular profile, while it is inclined to the downstream side of the micro-fins in the parabola profile. Smaller vorticity with opposite-flow circulation is observed in the lower corner downstream of the rectangular, trapezoidal, and triangular micro-fin profiles. Velocity streamlines cover and lie on the parabola profile better than other profiles. Flow is parallel to the micro-fin axis in axial micro-fins, so no flow separation or circulation occurs in axial micro-fins.

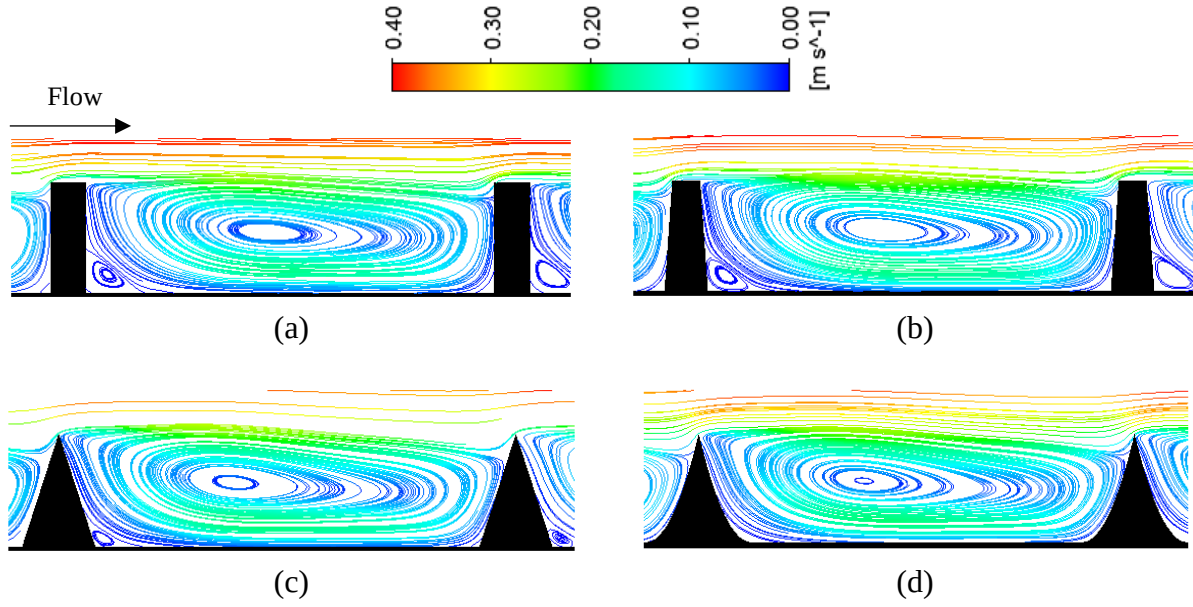


Figure 5.6. Velocity streamlines around transverse (a) rectangular, (b) trapezoidal, (c) triangular, and (d) parabola micro-fins of Scenario 6 at $p_f = 2$ mm

The y^+ was plotted versus dimensionless mean-velocity u^+ for the smooth case, and transverse and axial micro-fins of Scenarios 1 and 6, through line 1, as shown in Figure 5.2. According to the figure, changes in micro-fin height alter the velocity profile more than the micro-fin profile for both transverse and axial cases. Figure 5.7 shows that the difference between the smooth case's mean-velocity profile and the axial micro-finned cases is much more obvious in the logarithmic layer than the viscous sublayer, buffer layer, and outer region. Figure 5.7 clarifies that mean-velocity profiles of transverse cases are shifted more than the axial cases. The flow circulation near the surface is the main reason for the change of mean-velocity profiles in the transverse cases. There is no observed laminar sublayer in mean-velocity profiles in transverse cases; instead, the micro-fins produce a turbulent flow next to the surface. The study of near-wall velocity profiles revealed that the structure of micro-fins shortened the mean-velocity gradient in

the enclosures between micro-fins, resulting in general more significant wall shear stress for micro-fins cases than the smooth case.

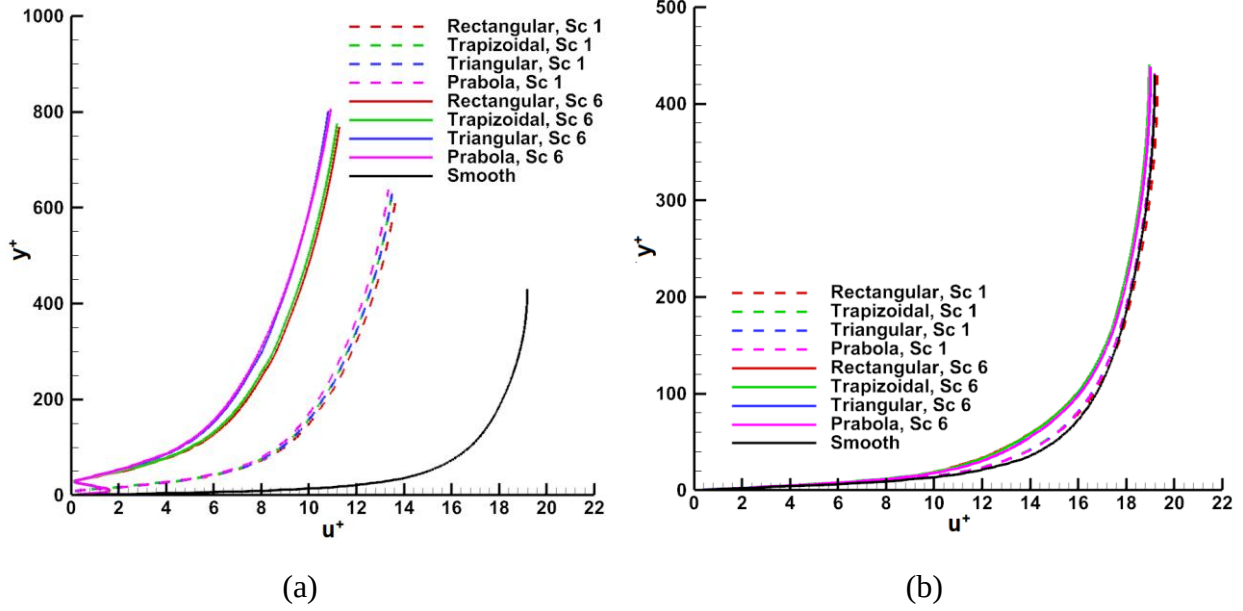


Figure 5.7. Mean-velocity profiles of turbulent boundary layers, through line 1 for (a) transverse and (b) axial micro-fins of Scenario 1, 6, and the smooth case.

To obtain the form of the universal velocity profile for rough surfaces, a dimension of roughness size must account for the logarithmic relationship, as discussed in Chapter 2. The dimensional size of the roughness is considered as the micro-fin height, which modifies the logarithmic relationship to $u^+ = 1/\kappa(\ln(y/e) + C_r^+)$, where C_r^+ is the integration constant and is referred to as the roughness function. To pursue the non-dimensional analysis, e was made dimensionless through $e^+ = eu_\tau/\nu$, in which e^+ is referred to as roughness Reynolds number (Webb and Kim, 2005).

Figure 5.8 shows how C_r^+ and κ in the logarithmic layer change with e^+ for different micro-fin profiles of transverse and axial cases. Accordingly, C_r^+ decreases and increases with an

increasing e^+ for transverse and axial cases, respectively. A significant finding of this study is that κ also increases with e^+ . Following classifications proposed by Schlichting and Gersten (2016), it should be noted that this study's turbulence regime is considered a transitionally rough regime based e^+ on variation for different cases.

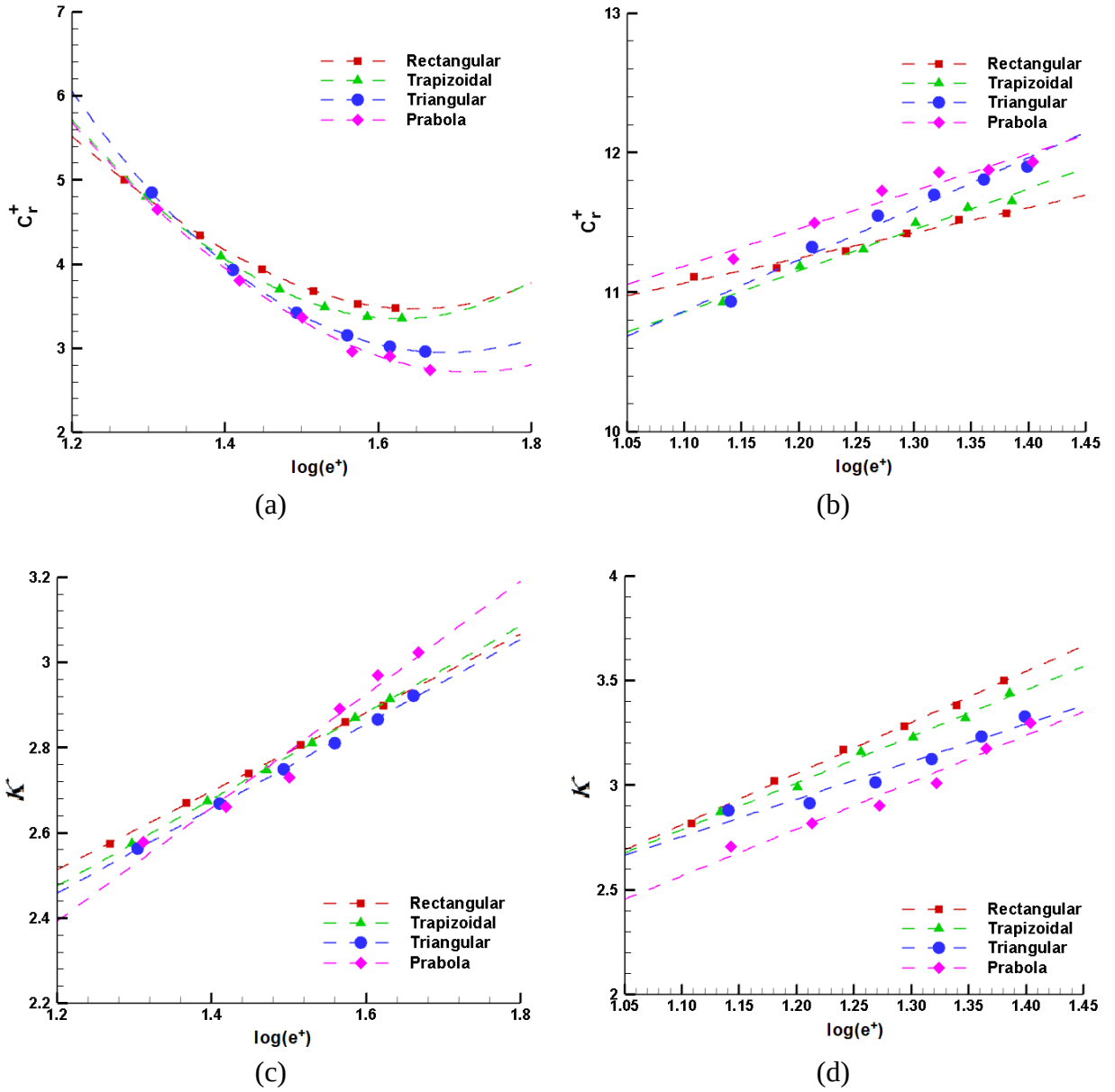


Figure 5.8. Function curves of C_r^+ in (a) transverse and (b) axial cases, and κ in (c) transverse and (d) axial cases for different e^+

Temperature contour comparisons are presented in Figure 5.9 for three different transverse micro-fin heights at $p_f = 2$ mm. Temperature contour lines become denser, indicating higher heat flux when the micro-fin height increases concerning unfinned surfaces (exposed portions of the base surface). The variation of temperature contours among different micro-fin profiles at a particular height was not remarkable.

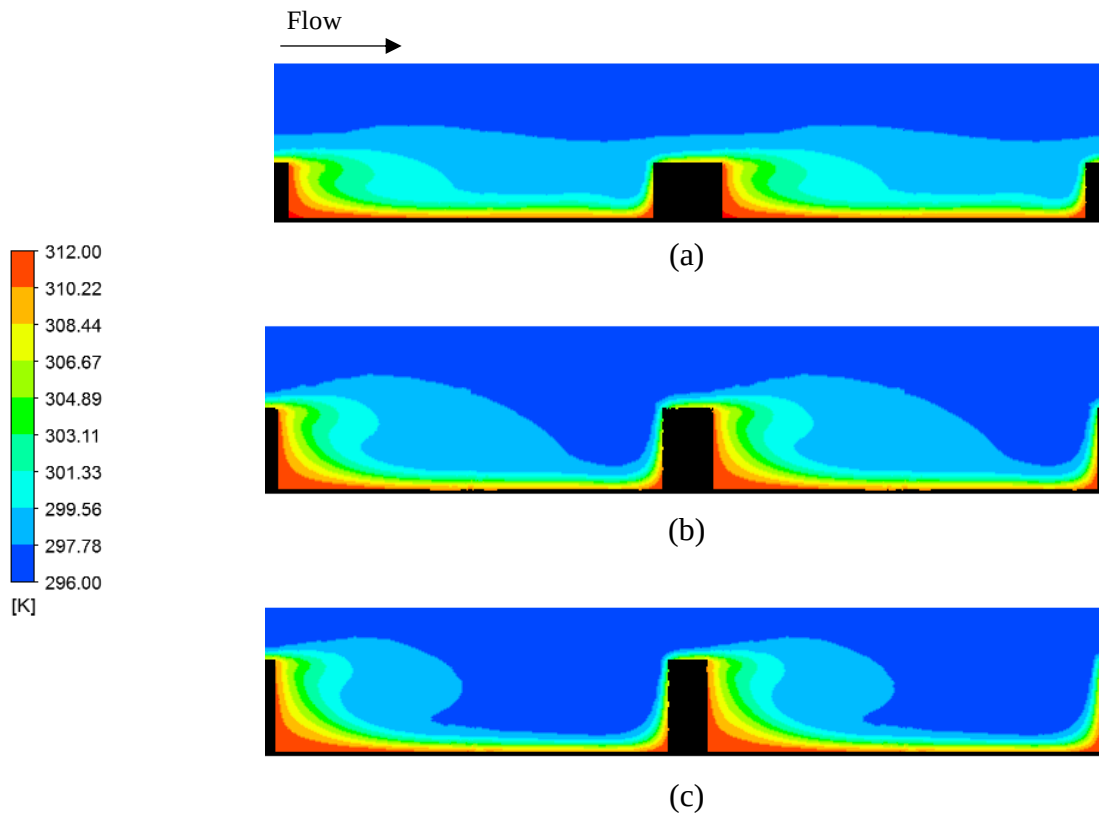


Figure 5.9. Contours of temperature for transverse rectangular micro-fins of Scenarios (a) 1, (b) 3, and (c) 5 at $p_f = 2$ mm

Figure 5.10 shows the temperature distribution in the solid domain of Scenario 6 for different transverse and axial micro-fin profiles at $p_f = 2$ mm. Accordingly, the left symmetry side of the solid domain of transverse cases has a sharper temperature gradient, indicating higher heat removal than on the right side. As flow is parallel to the micro-fin axis, the temperature gradient

is symmetrical throughout the solid domain of axial cases. In summary, rectangular and trapezoidal profiles of both transverse and axial cases have greater gradients than triangular and parabola profiles.

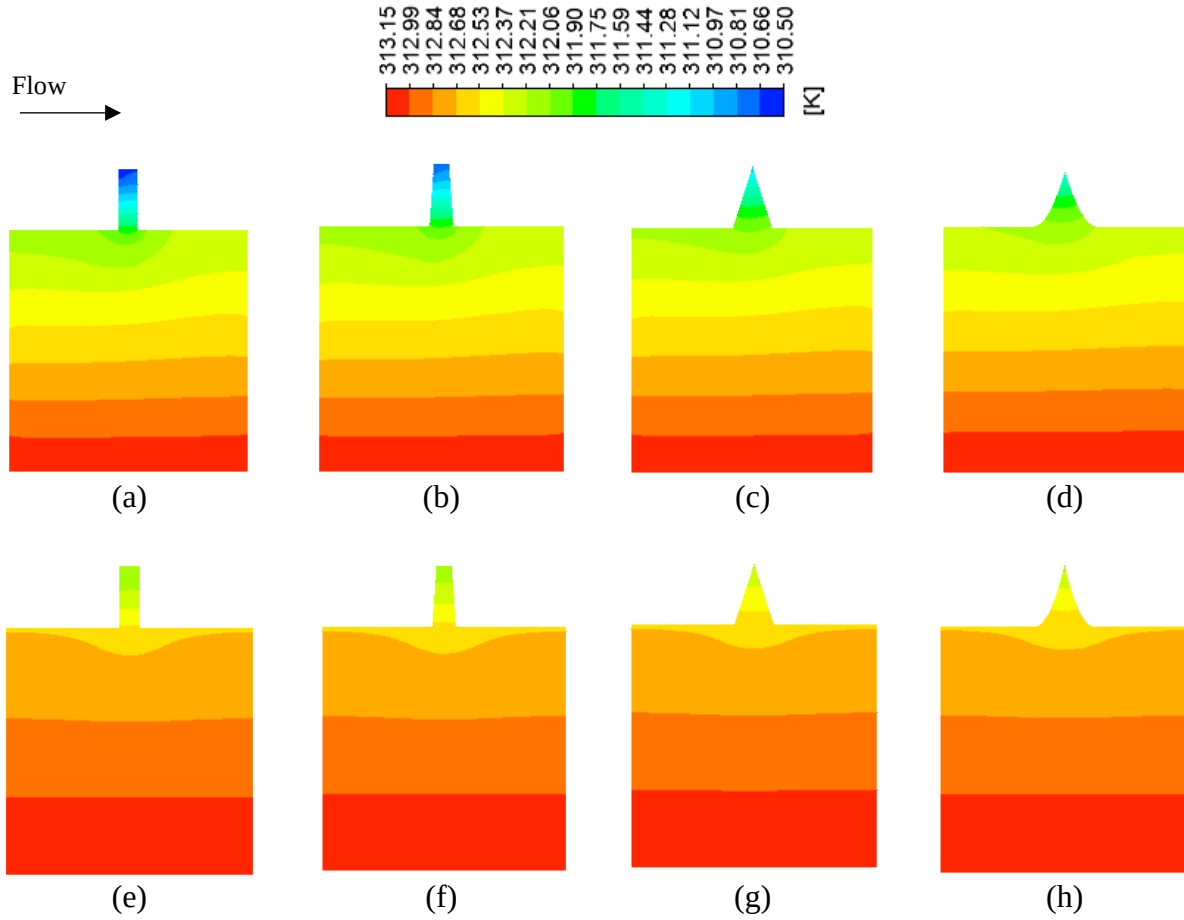


Figure 5.10. Contours of solid body temperature in transverse (a) rectangular, (b) trapezoidal, (c) triangular, and (d) parabola micro-fins, and axial (e) rectangular, (f) trapezoidal, (g) triangular, and (h) parabola micro-fins of Scenario 6 at $p_f = 2$ mm

5.5.2 Fin Performance

The ratio of total heat-transfer rate from micro-fin surfaces to the smooth surface (Q/Q_s) and the ratio of Fanning friction factor of micro-fin surfaces to the smooth surface (f/f_s) for

cases with $p_f = 2$ mm, are shown in Figure 5.11. As seen, the Q/Q_s and f/f_s of transverse cases are dramatically higher than axial cases. The micro-fin profile and height in the axial cases have minor effects on Q/Q_s and f/f_s . The f/f_s sharply increases as micro-fin height increases in transverse cases.

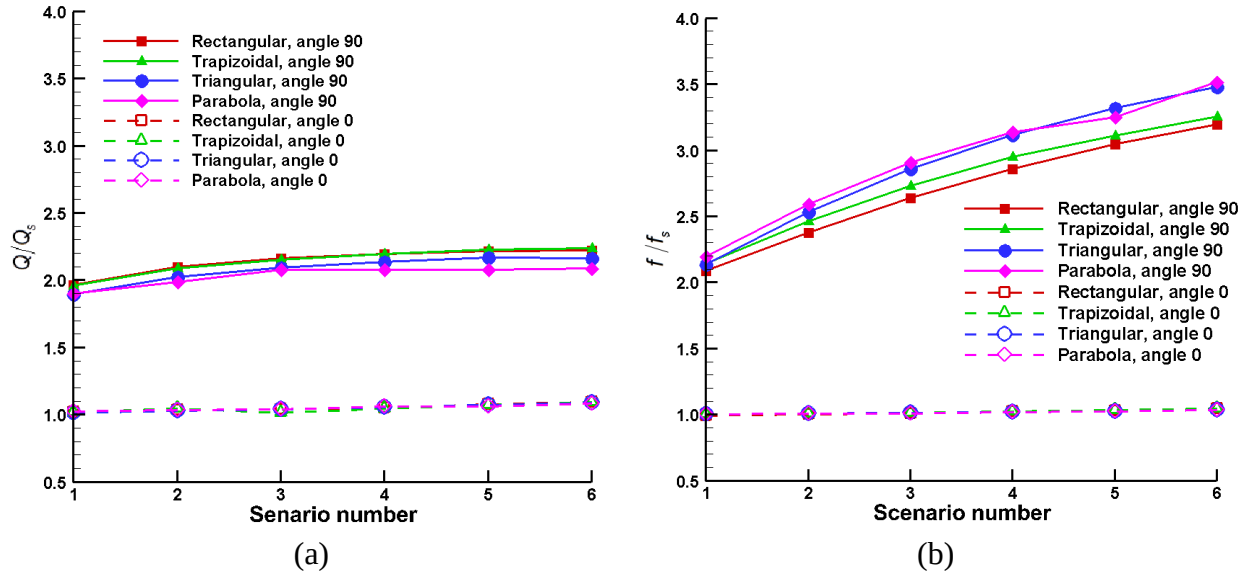


Figure 5.11. Performance characteristics: (a) Q/Q_s and (b) f/f_s for transverse and axial cases at $p_f = 2$ mm

Figure 5.12 shows the thermo-hydraulic efficiency index for all cases at $p_f = 2$ mm. The behavior of the thermo-hydraulic efficiency index is opposite for transverse and axial cases. The thermo-hydraulic efficiency index decreases as the micro-fin height increase in transverse cases, and it increases as the micro-fin height increases in axial cases. The micro-fin profile and height have a greater effect on the thermo-hydraulic efficiency index in transverse cases compared to axial cases. It is shown in Scenario 6 that the axial cases have the maximum efficiency index of 1.05.

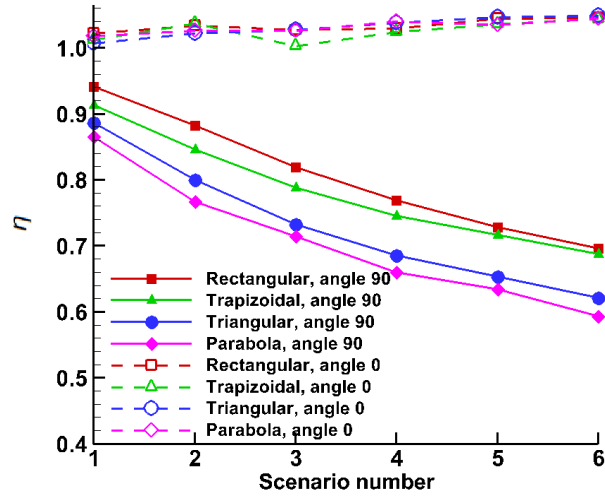


Figure 5.12. Thermo-hydraulic efficiency index for all cases at $p_f = 2$ mm

5.5.3 Pitch Control for Optimum Design

Scenario 1 of transverse and 6 of axial rectangular micro-fins were selected for this portion of the study because they had the best thermo-hydraulic efficiency index based on Figure 5.12. Figure 5.13 represents the comparison of velocity vectors around Scenario 1 of transverse rectangular micro-fins among the different ratios of p_f/e . According to the classification suggested by Jiménez (2004), Figure 5.13. Figure 5.13a corresponds to a d -type roughness in which $p_f/e < 4$ and the temperature gradient is very thick, which results in low heat transfer. Figure 5.13.c and d correspond to a k -type roughness ($p_f/e > 4$). And, Figure 5.13b corresponds to a transition between a d -type to a k -type roughness. There is no flow separation and reattachment for d -type micro-fins. As changing from d -type to k -type, the center of vorticity shifts from upstream of the micro-fin to downstream, and the temperature gradient becomes denser overall. The reattachment point varies with p_f/e in k -type micro-fins, and the redeveloping boundary

layer occurs after the reattachment point. The length of the redeveloping boundary layer increases as p_f/e increases. It can be concluded that there are two maximum heat-transfer points in k -type micro-fins, one at the micro-fin upstream tip and one at the reattachment point.

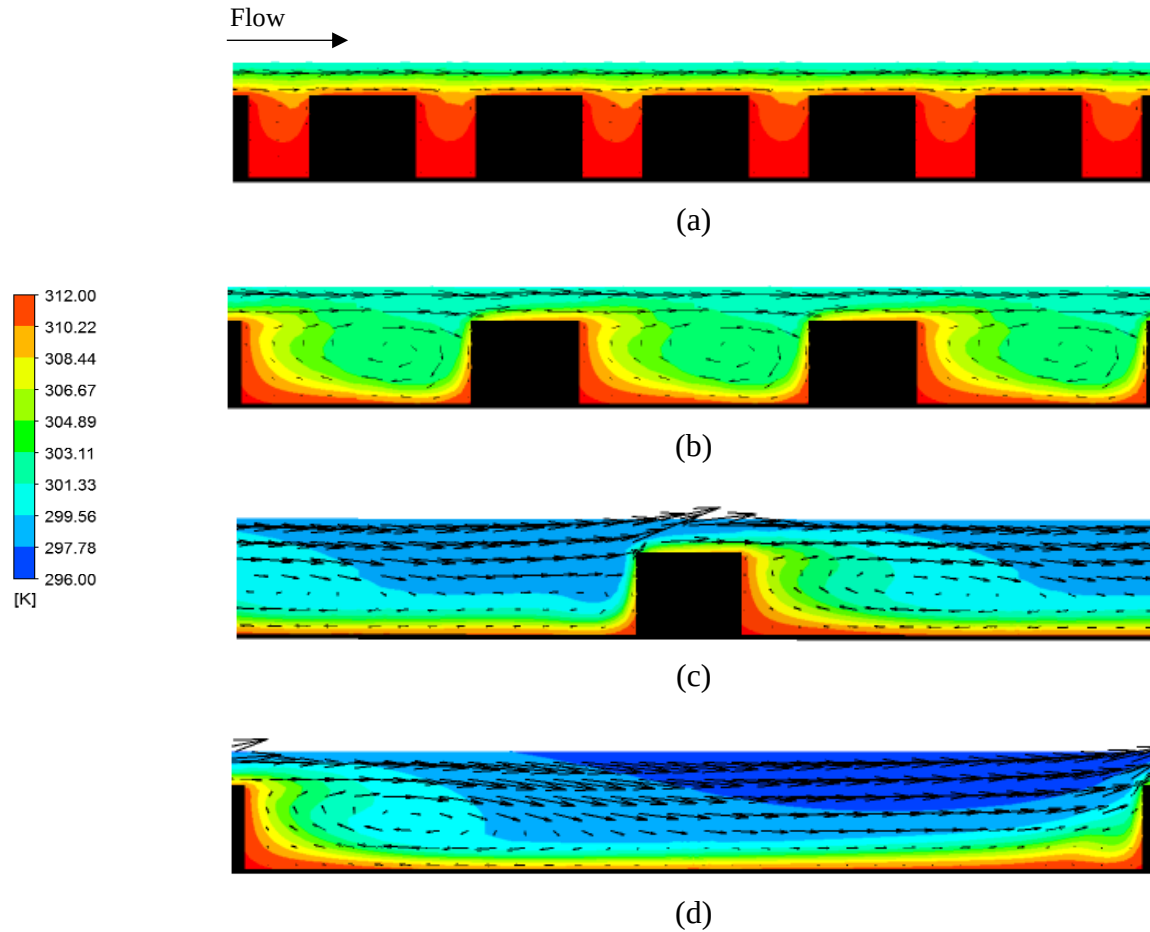


Figure 5.13. Velocity vectors and contours of temperature for transverse rectangular micro-fins of Scenario 1 with p_f/e ratios of (a) 2, (b) 4, (c) 8, and (d) 12.

Figure 5.14 represents the comparison of velocity contours for Scenario 6 of axial rectangular micro-fins among the different ratios of p_f/e . No flow gets inside the micro-fin's enclosure when p_f/e is very narrow based on the velocity magnitude in Figure 5.14a. Generally,

the average velocity magnitude in the middle of the micro-fin enclosure increases as the p_f/e gets bigger.

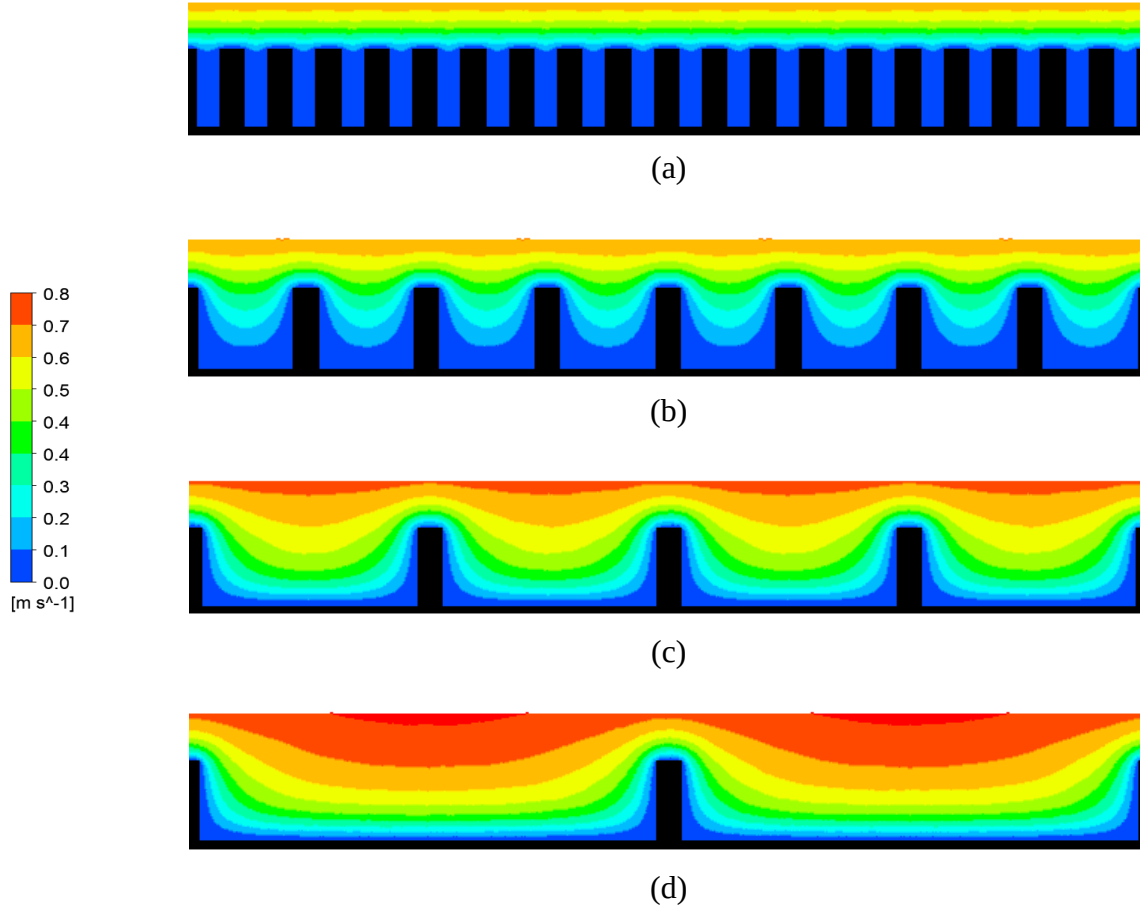


Figure 5.14. Velocity contours for axial rectangular micro-fins of Scenario 6 with p_f/e ratios of (a) 0.6, (b) 1.5, (c) 3, and (d) 6

Figure 5.15 shows the impact of p_f/e ratio on Q/Q_s , f/f_s , and the thermo-hydraulic efficiency index for Scenario 1 of the transverse rectangular micro-fin and Scenario 6 of the axial rectangular micro-fin. According to Figure 15a, the Q/Q_s increases as p_f/e increases from 2 to 5 and decreases after $p_f/e = 5$. Thus, an optimal pitch ratio maximizes the thermal performance characteristic. The f/f_s increases as p_f/e increases, but its increase after $p_f/e = 8$ is negligible. The pitch ratio that maximizes the thermo-hydraulic efficiency index to 1.4 corresponds to a d-

type micro-fin with $p_f/e = 3.5$. According to the so-called Reynolds analogy, changing p_f/e demonstrated that Q/Q_s and f/f_s do not always have a positive correlation. In transverse rectangular micro-fins, all p_f/e ratios lead to HTE. In contrast, in axial rectangular cases, when $p_f/e < 1.5$ then Q/Q_s becomes less than one, just as this phenomenon was explained in Figure 5.14. In addition, since there is no flow inside the micro-fin enclosure at $p_f/e < 1.5$, the channel cross-area decreases and velocity gets higher throughout the channel, which results in an increase of f/f_s at p_f/e less than 1.5. At ratio of $p_f/e = 2$, the thermo-hydraulic efficiency index is maximum for axial cases.

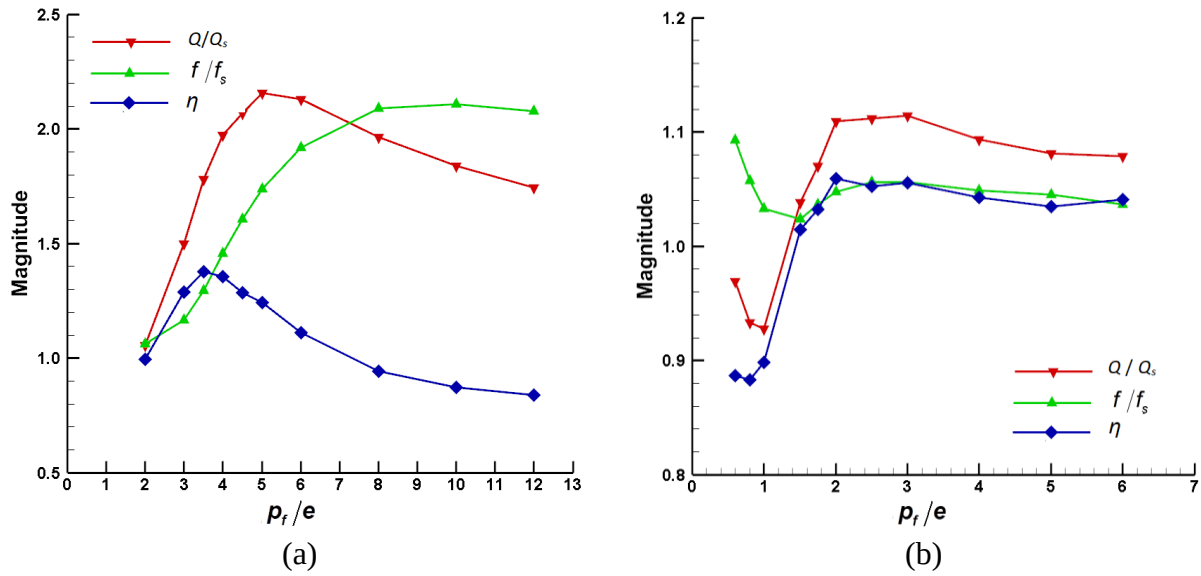


Figure 5.15. Comparison of Q/Q_s , f/f_s , and the thermo-hydraulic efficiency index for different p_f/e ratios of (a) transverse rectangular micro-fin of Scenario 1 and (b) axial rectangular micro-fin of Scenario 6

6 Parametric Study of 2D and 3D Micro-fin Tubes on Fluid Flow and Heat Transfer

This study applies turbulent fluid flow and conductive and convective heat transfer simulation to evaluate the performance of different 2D and 3D micro-fin tubes. Different configurations are generated by varying micro-fin height, helix angle, number of starts, and discontinuity features. Coupled solid and periodic fluid domains were applied in ANSYS Fluent 19.1. An automated model was set up using a python library to read micro-fin design variables, update the simulation model, and send the model to a local supercomputer (Beocat), a high-performance computing (HPC) cluster at Kansas State University to run the model and post-process the results. Performance was mapped for 211 different simulations (including a smooth tube) at Reynolds number (Re) of 49,000.

6.1 Design Variables

This study aims to better understand performance by modifying 2D micro-fin tubes to 3D by adding discontinuity features known as crosscuts. Figure 6.1 illustrates design variables for a 3D micro-fin tube.

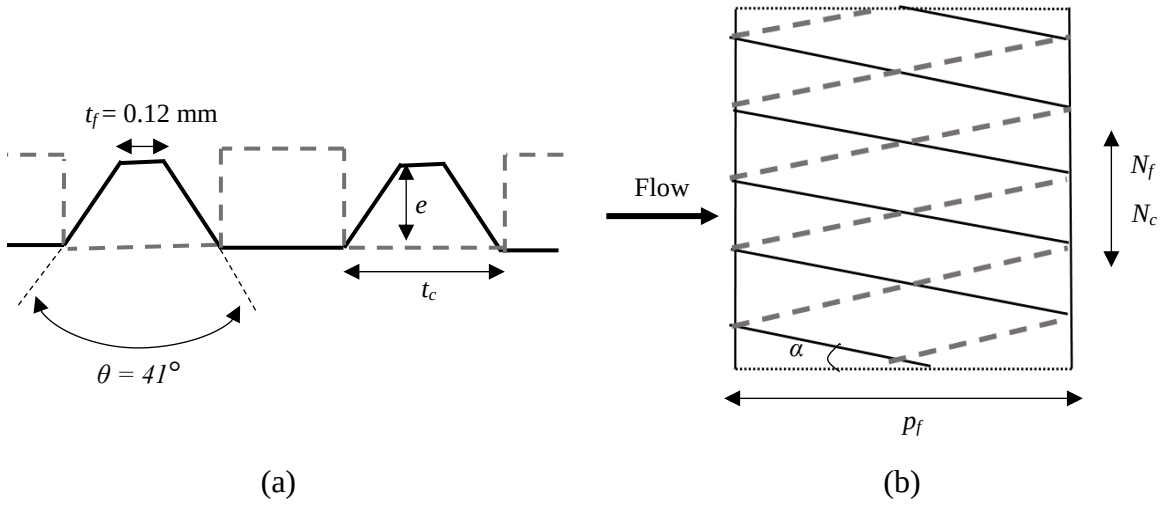


Figure 6.1. A schematic of (a) micro-fin and cutter profiles and (b) an unrolled tube with micro-fins and cutters (solid and dash lines represent micro-fins and cutters, respectively)

A micro-fin cross-section profile has little influence on heat transfer enhancement and pressure drop compared to other designed variables, as concluded in Chapter 5. This study sets the following base variables drawn from Webb et al. (1999): a fixed trapezoidal micro-fin profile with an included angle between sides of micro-fins (θ) of 41° , micro-fin thickness (t_f) of 0.12 mm, $D_i = 15.54 \text{ mm}$, and outer tube diameter (D_o) of 19 mm. The number of discontinuities per pitch is calculated by N_c/N_f , in which N_c is the cutter number of starts. The axial length of discontinuity is characterized by $L_a = t_c / \sin(\alpha)$. The axial cutter pitch (p_c) is equal to $(1/\delta) \times p_f$ in which p_f is defined as the length of one complete helix turn and depends on variables of helix angle and number of starts as follows:

$$p_f = \frac{\pi D_i}{N_f \tan(\alpha)}. \quad (6.1)$$

The selected numerical value of design variables are described in the following:

$$e \text{ (mm)} = [0.1, 0.3, 0.5] \quad (6.2)$$

$$\alpha \text{ (}^\circ\text{)} = [25, 45, 60] \quad (6.3)$$

$$N_f = [15, 30, 45, 60] \quad (6.4)$$

$$N_c = [15, 30, 45] \quad (6.5)$$

$$t_c \text{ (mm)} = [0, 0.5, 1, 1.5]. \quad (6.6)$$

Based on the defined design variables and equal values of helix angle for micro-fins and cutters, 44 and 167 micro-fin configurations are defined for 2D and 3D micro-fin tubes, respectively.

6.2 Geometry and Boundary Condition

Micro-fin and cutter profiles are monotonically distributed in tube circumferential direction based on their number of starts. Then, they are helically rotated along the tube's center axis with an identical helix angle but with a reverse helix direction. Therefore, cutters cut micro-fins on their extension on the tube's inner surface. Figure 6.2 geometrically represents solid and fluid computational domains of 2D and 3D micro-fin tubes.

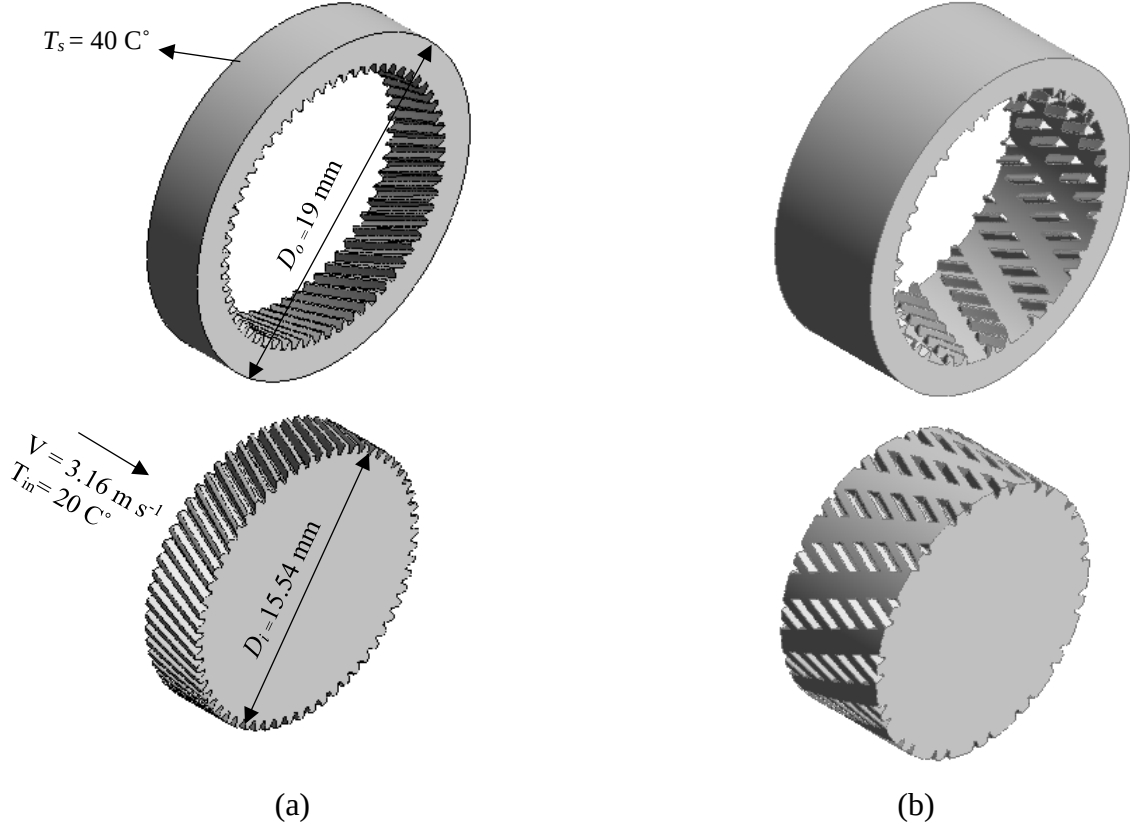


Figure 6.2. Geometrical representation of solid and fluid computational domains of a (a) 2D and (b) 3D micro-fin tube

The problem is defined as fluid enters the channel with constant velocity ($V = 3.16 \text{ m s}^{-1}$) with entering temperature of $T_{in} = 20 \text{ C}^\circ$. A constant and uniform wall temperature of $T_s = 40 \text{ C}^\circ$ is defined at the outer wall of the solid surface.

Since the tube inlet and outlet boundaries are periodic, a domain length study was conducted to find the minimum number of helix turns that gave convergence satisfaction. Table 6.1 shows results of different simulation tests with different number of helix turn for Tube 3 ($e = 0.398 \text{ mm}$, $\alpha = 45$, $N_f = 30$) which is the tube from Webb et al. (1999) at $Re = 48,928$. Accordingly, the solution did not satisfy convergence if the domain was only set to one helix turn. The number of helix turns makes a negligible difference in j and f when an integral number of two or greater is used.

Table 6.1. Results of length study (errors are in respect to six helix turns)

Number of helix turn	length (mm)	j	f	Error (%)	
				j	f
2	0.0033	0.00644	0.011604	0.6	- 0.1
4	0.0065	0.00645	0.011589	0.5	0.0
6	0.0098	0.00648	0.011593	-	-

6.3 Meshing and Independence Study

An unstructured tetrahedral grid system is applied to generate the mesh for smooth and micro-fin tubes. In the fluid part, the minimum height of the first grid point was set at $0.003D_i$ from the wall to satisfy $y^+ < 5$. Figure 6.3 shows the grid system for a 3D micro-fin tube.

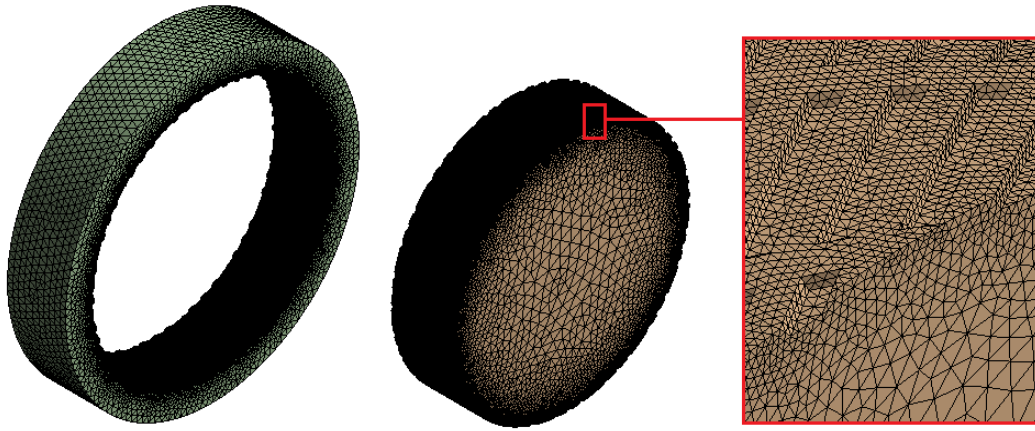


Figure 6.3. Grid system of solid and fluid domains with a zoom-in view at the solid-fluid interface for a 3D micro-fin tube

Mesh independence testing was carried out at $Re = 48,928$ to find proper grid size for smooth and micro-fin tubes. Simulations with different grid sizes were performed for a smooth tube and Tube 3 from Webb et al. (1999). Figure 6.4 shows j and f versus element number of mesh for a smooth tube and Tube 3.

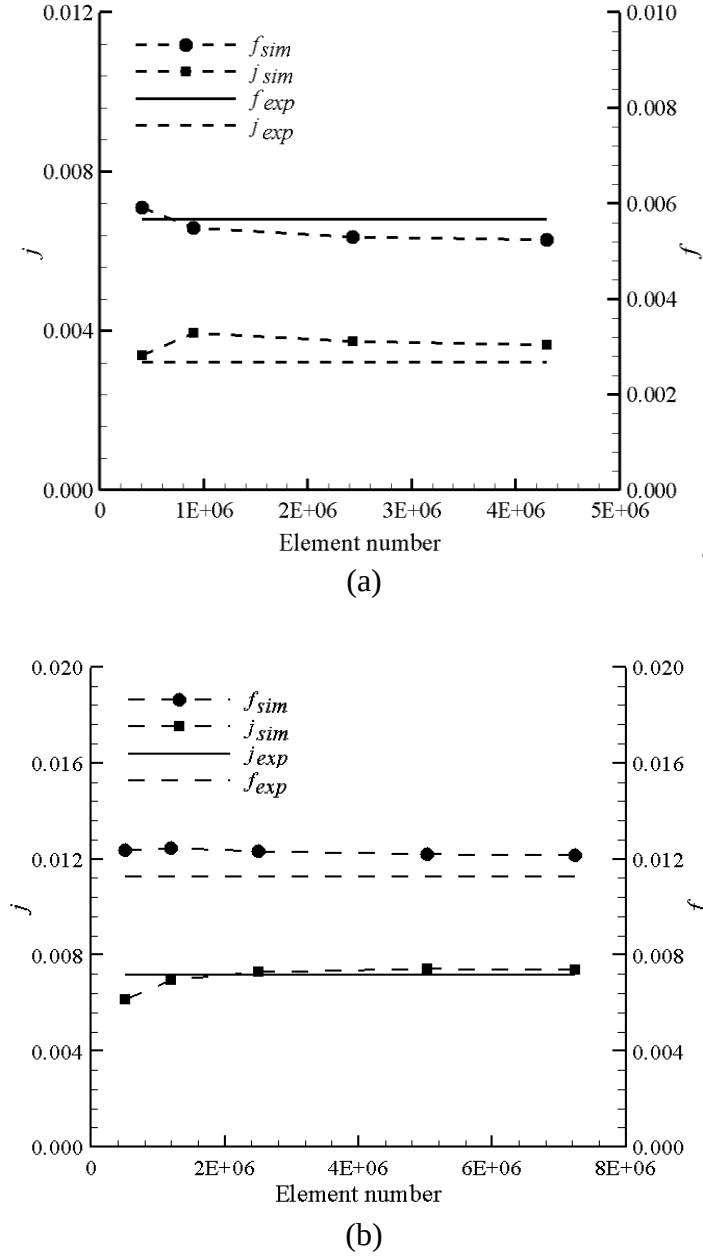


Figure 6.4. Mesh independency analysis for (a) the smooth tube and (b) Tube 3

Figure 6.4 shows no remarkable change for j and f beyond 24×10^5 and 25×10^5 elements for the smooth tube and Tube 3, respectively. Hence, this study chose a grid size of 0.8 mm for both solid and fluid domains of smooth and micro-fin tubes and a grid size of 0.06 mm and 0.07 mm for solid-fluid contact surface of smooth and micro-fin tubes, respectively.

6.4 Model Validation

The numerical results were validated with experimental results for the smooth tube and variety of d -type and k -type 2D micro-fin tubes reported by Webb et al. (1999). Table 6.2 compares the numerical and experimental results for the chosen mesh size. The numerical model overestimates j and underestimates f for three-fourths of the tubes. Generally, it is concluded that the model accuracy is sufficient for this study.

Table 6.2. Numerical and experimental results comparison at $Re = 49,000$

Tube #	p_f/e	j	f	j_{exp}	f_{exp}	Error (%)	
						j	f
1	Smooth	0.00365	0.0052	0.00326	0.0057	-13.9	7.5
2	3.31	0.00786	0.0114	0.00724	0.0117	-8.6	2.7
3	4.08	0.00729	0.0123	0.00718	0.0112	-1.6	-9.7
4	11.25	0.00536	0.0102	0.00533	0.0100	-0.6	-1.4
5	3.74	0.00748	0.0114	0.00695	0.0116	-7.7	1.4
6	5.65	0.00625	0.0101	0.00639	0.0110	2.1	8.4
7	7.87	0.00561	0.0082	0.00590	0.0090	4.9	8.3
8	10.49	0.00522	0.0077	0.00480	0.0087	-8.9	11.3

6.5 Result and Discussion

6.5.1 Parametric Study on 2D Micro-fin Tubes

Figure 6.5 shows changes of the ratio of j/j_s with the micro-fin height and number of starts for different helix angles in 2D micro-fin tubes. The highest and lowest height shows the best and the worst performance for both k -type and d -type micro-fins, respectively. The maximum ratio of j/j_s (2.45) belongs to a d -type micro-fin tube with $e = 0.5$ mm, $\alpha = 60^\circ$, and $N_f = 30$.

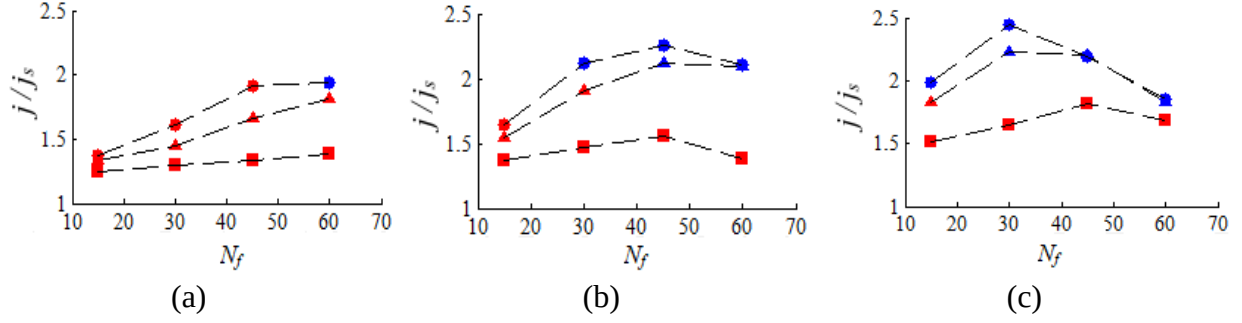


Figure 6.5. Effects of the micro-fin number of starts and height for different helix angles of (a) 25°, (b) 45°, and (c) 60° (square dot represents $e = 0.1$ mm, triangle dot represents $e = 0.3$ mm, and round dot represents $e = 0.5$ mm; and blue and red dots represent d -type and k -type micro-fins, respectively)

Figure 6.6 shows changes of the ratio of f/f_s with the micro-fin height and number of starts for different helix angles for 2D micro-fin tubes. For different values of number of starts and helix angles, micro-fins with greatest height have the largest ratio of f/f_s . The trend of f/f_s ratio with the increase of number of starts at $\alpha = 25^\circ$ is in opposite with the increase of number of starts at $\alpha = 60^\circ$. The minimum ratio of f/f_s (1.11) belongs to a k -type micro-fin with $e = 0.1$ mm, $\alpha = 25^\circ$, and $N_f = 15$.

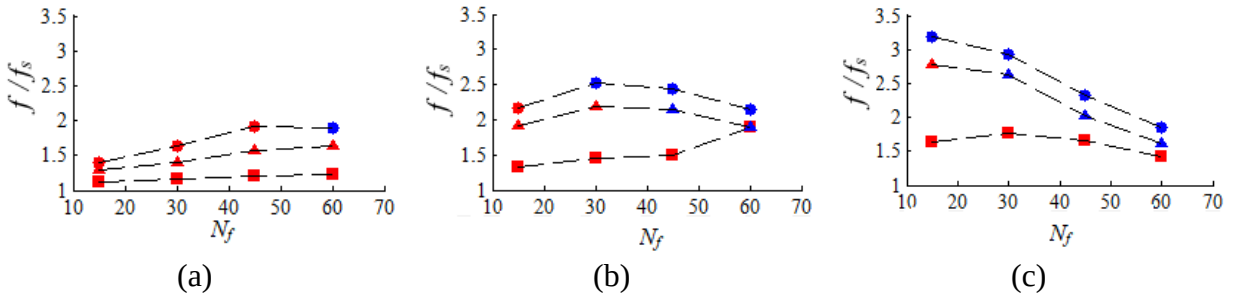


Figure 6.6. Effects of the micro-fin height and number of starts on f/f_s for different helix angles of (a) 25°, (b) 45°, and (c) 60° (square dot represents $e = 0.1$ mm, triangle dot represents $e = 0.3$ mm, and round dot represents $e = 0.5$ mm; and blue and red dots represent d -type and k -type micro-fins, respectively)

Figure 6.7 shows changes of the thermo-hydraulic efficiency index with the micro-fin height and number of starts for different helix angles for 2D micro-fin tubes. The helix angle of 25° brings $\eta \geq 1$ for micro-fins. The thermo-hydraulic efficiency index of d -type micro-fins increases with the increase of the number of starts. The highest values of the thermo-hydraulic efficiency index ($\eta = 1.18$) is for a k -type micro-fin with $e = 0.1$, $\alpha = 60^\circ$, and $N_f = 60$.

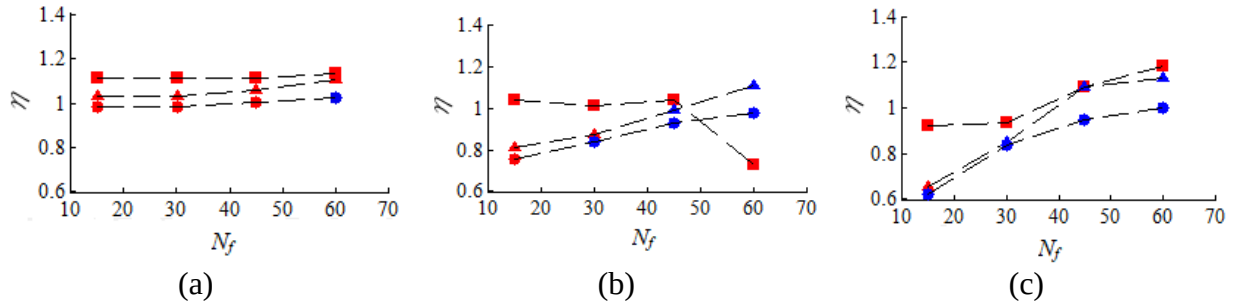


Figure 6.7. Effects of the micro-fin height and number of starts on the efficacy index for different helix angles of (a) 25° , (b) 45° , and (c) 60° (square dot represents $e = 0.1$ mm, triangle dot represents $e = 0.3$ mm, and round dot represents $e = 0.5$ mm; and blue and red dots represent d -type and k -type micro-fins, respectively)

6.5.2 Parametric Study on 3D Micro-fin Tubes

Figure 6.8 shows the effects of the micro-fin height, helix angle, and number of starts on ratios of j/j_s and f/f_s for 3D micro-fin tubes. The dashed line represents $\eta = 1$ so that micro-fins placed below of the line have $\eta < 1$ and above of the line have $\eta > 1$. Generally, increasing micro-fin height increases ratios of j/j_s and f/f_s , and decreases the thermo-hydraulic efficiency index. Micro-fins with helix angles of 25° and 60° have the minimum and maximum deviation from the dashed line, respectively. Generally, for a particular f/f_s ratio, increasing the number of starts has a positive effect on values of heat transfer enhancement and the thermo-hydraulic efficiency index.

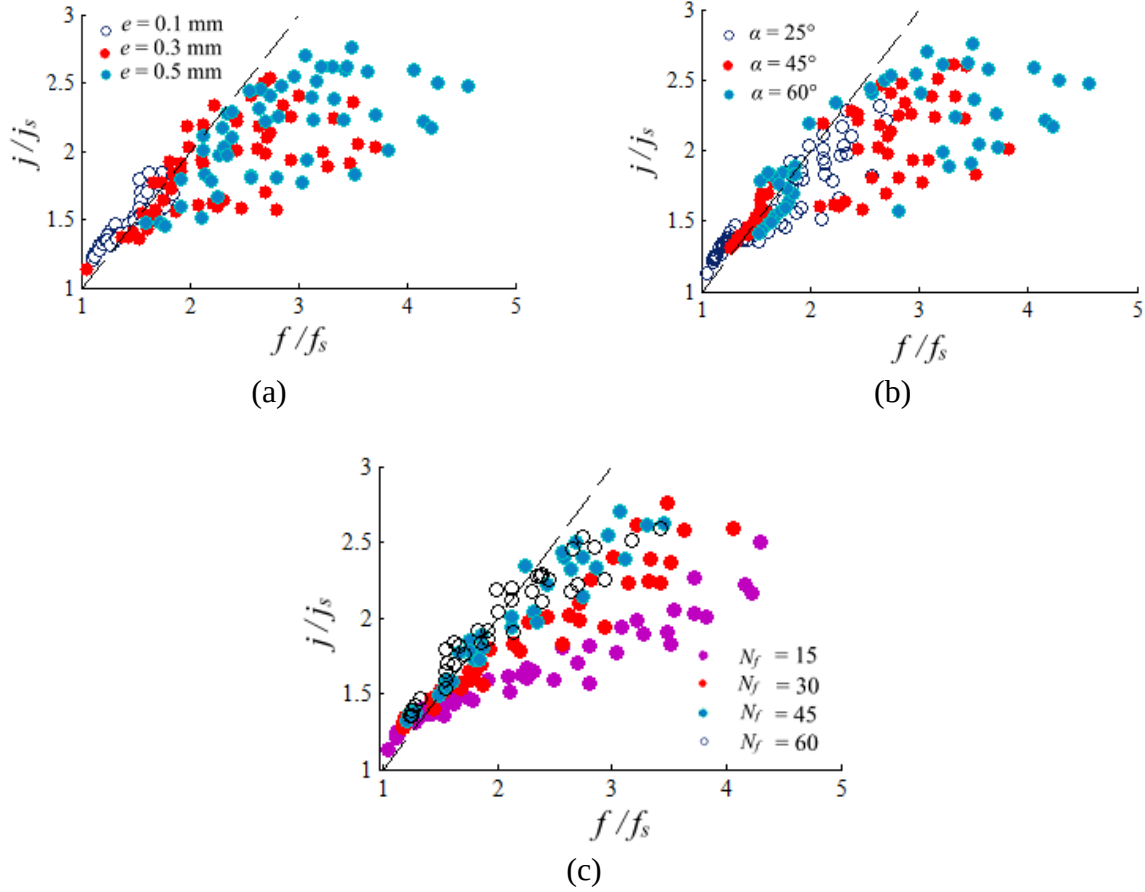


Figure 6.8. Effect of micro-fin variables of micro-fin (a) height, (b) helix angle, and (c) number of start on j/j_s and f/f_s

Figure 6.9 shows the effects of discontinuity features and the p_f/e ratio on the ratio of j/j_s for 3D micro-fin tubes. Accordingly, the j/j_s ratio decreases with the increase of p_f/e ratio. The maximum ratio of j/j_s (2.76) belongs to a d -type micro-fin ($p_f/e = 1.82$) with $e = 0.5$ mm, $\alpha = 60^\circ$, $N_f = 30$, $\delta = 1$, and $L_a = 0.57$. The 2D and 3D micro-fins have similar values for design variables when the ratio of j/j_s is maximum. The 3D micro-fin tube enhanced the j/j_s ratio 13% more than the 2D micro-fin tube.

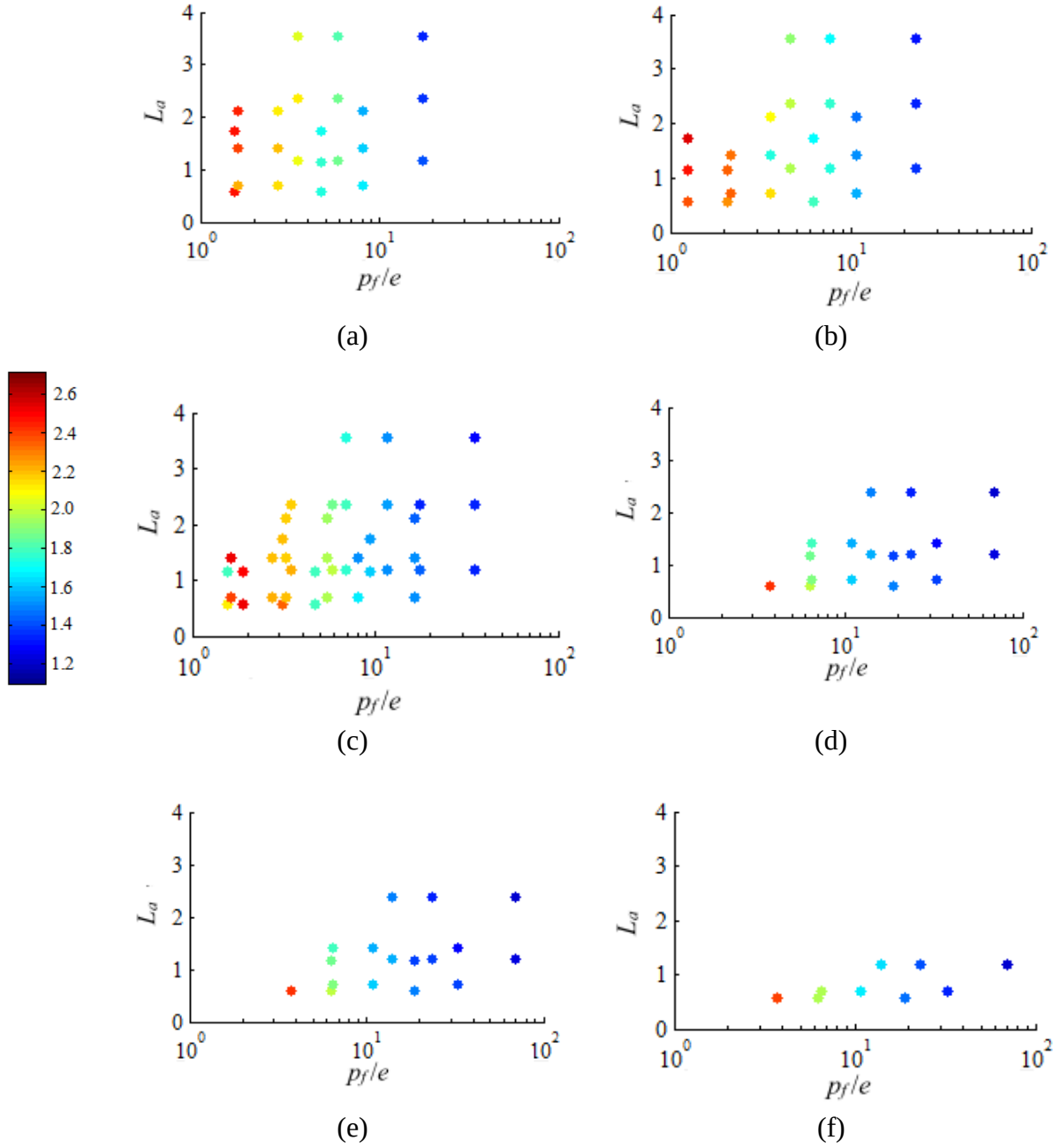


Figure 6.9. Effects of the p_f/e ratio and axial length of discontinuity on the j/j_s ratio for different number of discontinuities of (a) 0.25, (b) 0.33, (c) 0.5, (d) 1, (e) 2, and (f) 3

Figure 6.10 shows the effects of discontinuity features and the p_f/e ratio on the ratio of f/f_s for 3D micro-fin tubes. The ratio of f/f_s decreases with the increase of p_f/e ratio overall.

For a particular ratio of p_f/e , the ratio of f/f_s increases with the increase of the number of discontinuity.

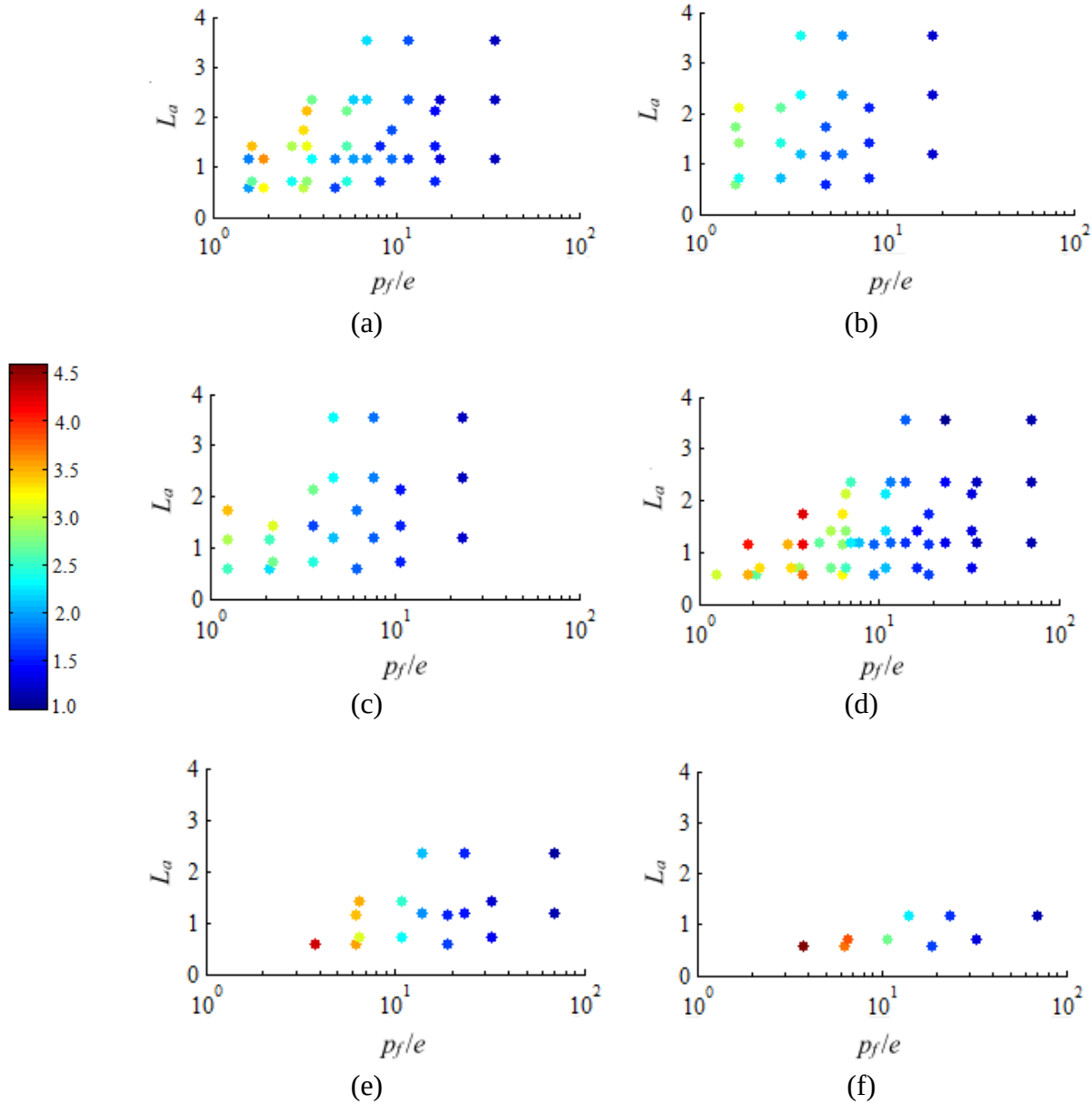


Figure 6.10. Effects of the p_f/e ratio and axial length of discontinuity on the f/f_s ratio for different number of discontinuities of (a) 0.25, (b) 0.33, (c) 0.5, (d) 1, (e) 2, and (f) 3

Figure 6.11 shows the effects of discontinuity features and the p_f/e ratio on the thermo-hydraulic efficiency index for 3D micro-fin tubes.

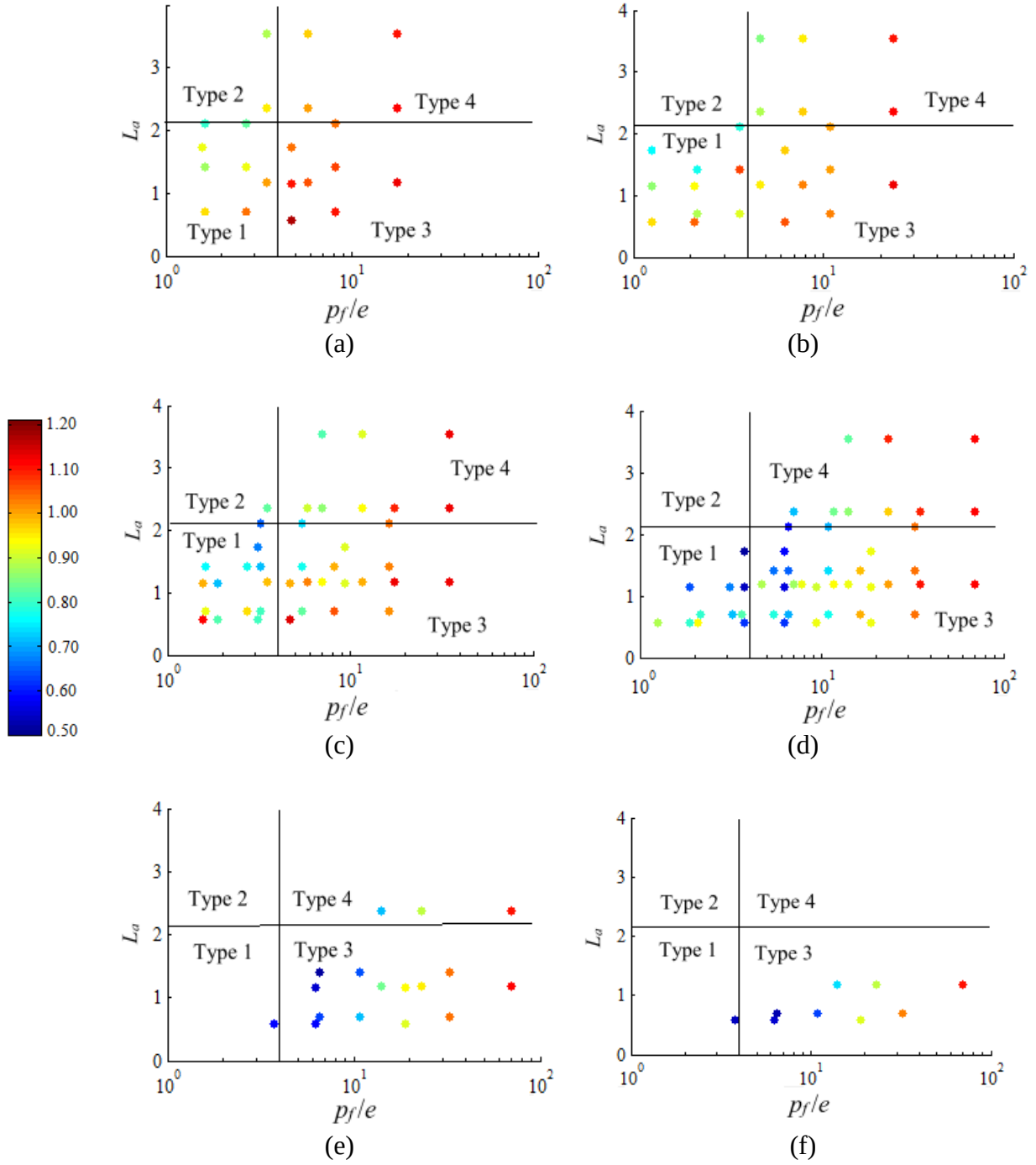


Figure 6.11. Effects of the p_f/e ratio and axial length of discontinuity on the thermo-hydraulic efficiency index for different number of discontinuities of (a) 0.25, (b) 0.33, (c) 0.5, (d) 1, (e) 2, and (f) 3

Based on Figure 6.11, four different micro-fin types can be defined. Type 1 refers to d -type micro-fins with $L_a < 2.1$ mm, Type 2 refers to d -type micro-fins with $L_a > 2.1$ mm, Type 3 refers

to k -type micro-fins with $L_a < 2.1$ mm, and Type 4 refers to k -type micro-fins with $L_a > 2.1$. For all micro-fin types, the thermo-hydraulic efficiency index decreases with the increase of the number of discontinuities. For $\delta < 1$, micro-fins with Type 3 have a greater thermo-hydraulic efficiency index than other micro-fins. According to Figure 6.11.e and f, the thermo-hydraulic efficiency index increases with the increase of the p_f/e ratio. The maximum of the thermo-hydraulic efficiency index is 1.16 which belongs to a k -type micro-fin ($p_f/e = 4.7$) with $e = 0.1$ mm, $\alpha = 60^\circ$, $N_f = 60$, $\delta = 0.25$, and $L_a = 0.43$. The 2D and 3D micro-fins have similar values for design variables for the maximum value of the thermo-hydraulic efficiency index. The best 2D micro-fin tube enhanced the thermo-hydraulic efficiency index up to 1.7% greater than the best 3D one. And, the best 3D micro-fin tube can enhance j/j_s 7% more than the best 2D tube.

6.5.3 Velocity and Thermal Fields for Smooth and Best Micro-fin Tubes

Figure 6.12 illustrates temperature contours of fluid and solid cross-sections of the smooth and the best 2D micro-fin tube and a partial solid-fluid interface of the micro-fin tube. The greater temperature gradient of the fluid and solid cross-sections and colder center flow temperature of the micro-fin tube prove the greater heat transfer enhancement of the micro-fin tube compared to the smooth tube. According to Figure 6.12.f, the maximum heat transfer happens upstream of the micro-fin tip, and minimum heat transfer occurs at the macro-fin' valley center.

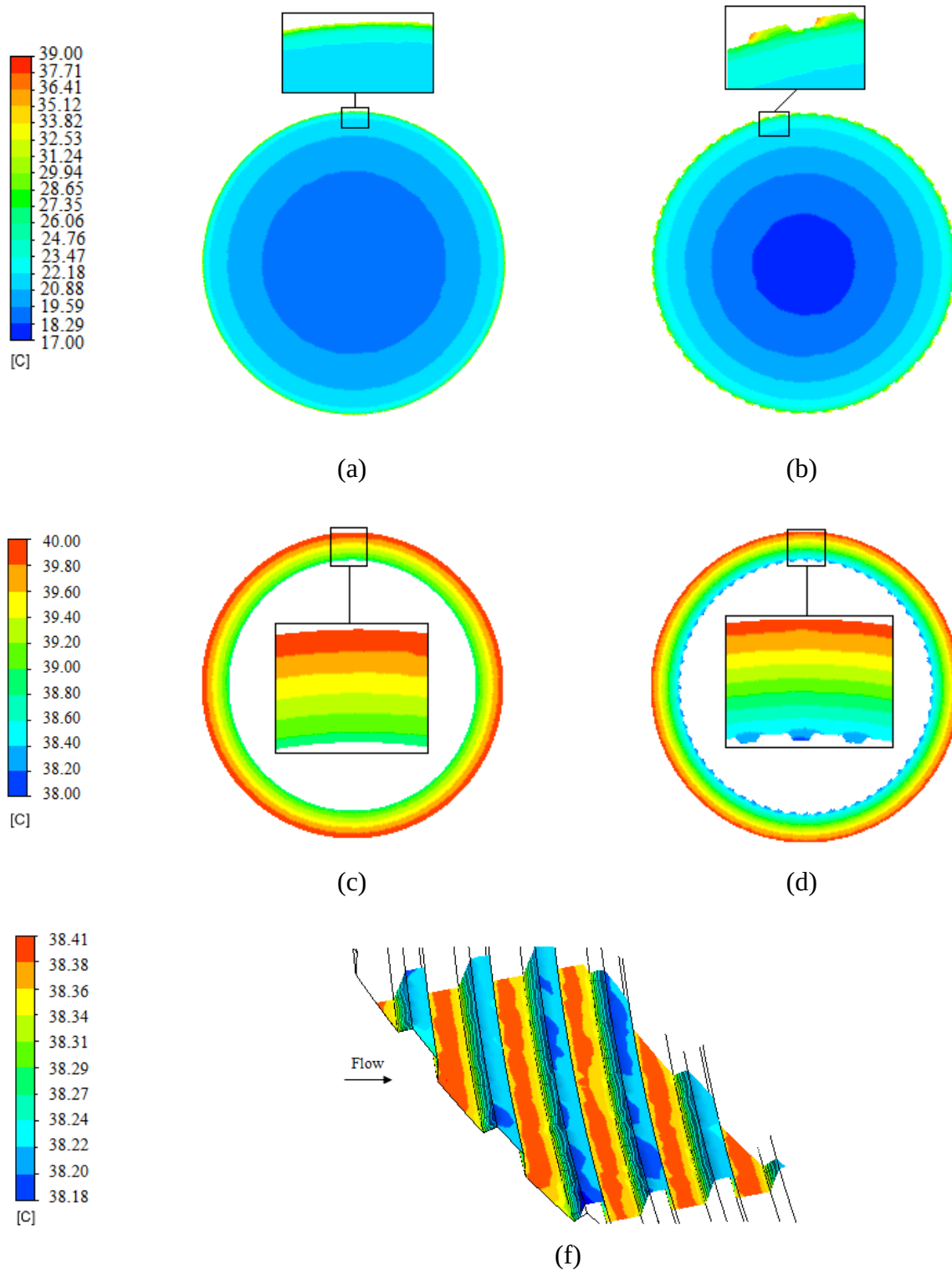


Figure 6.12. Temperature contours of a fluid cross-section of the (a) smooth and (b) micro-fin tube; a solid cross-section of the (c) smooth and (d) micro-fin tube; and (f) a partial solid-fluid interface of the micro-fin

Figure 6.13 shows velocity contours for the smooth and best 2D micro-fin tube, including velocity vectors above a partial surface of the best 2D micro-fin tube. The velocity gradient is sharper in the micro-fin tube compared to the smooth tube. The center flow velocity is 0.3 ms^{-1} greater in the micro-fin tube than the smooth tube. According to Figure 6.13.c, flow circulations in the micro-fin valleys assure a higher friction factor for the micro-fin surface rather than the smooth surface. The center of the flow circulation is close to the upstream of the micro-fin. There is no flow reattachment in the micro-fin valley due to the shortness of this area.

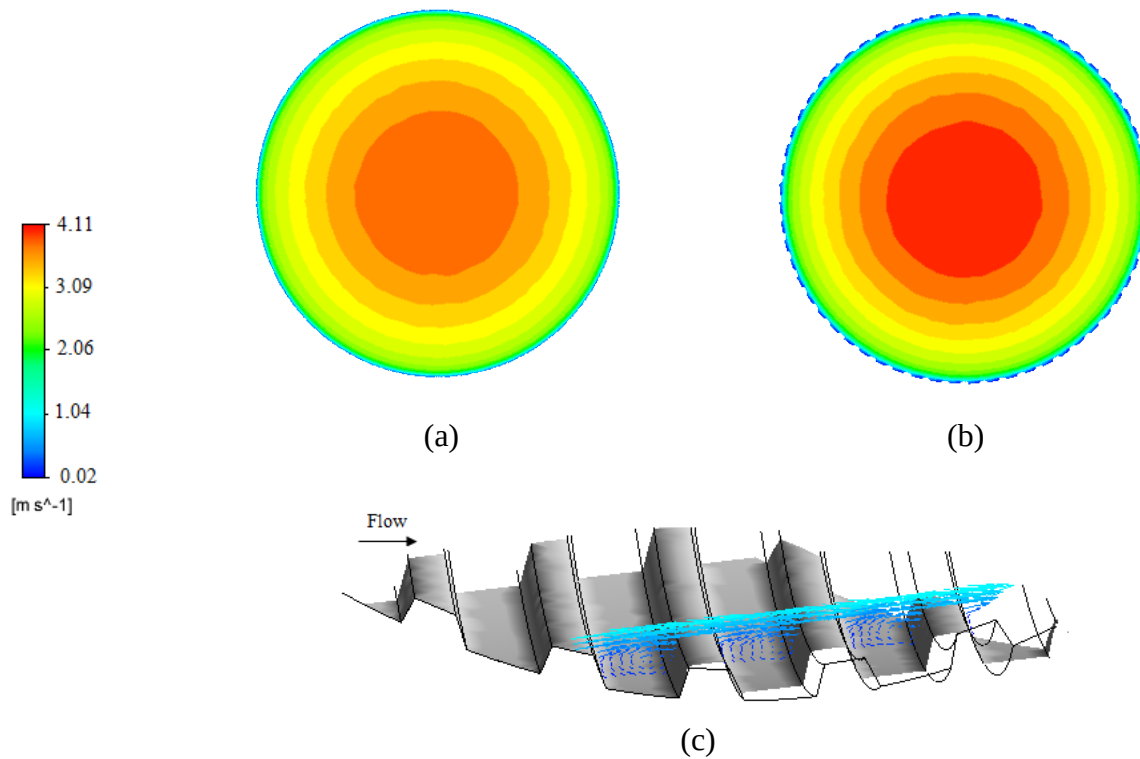


Figure 6.13. Velocity contours for the (a) smooth and (b) micro-fin tube and (c) velocity vectors above a partial surface of the micro-fin

7 Performance Study of LS-SVR Model on 2D and 3D Micro-fin Tubes

This study evaluates the potential of the LS-SVR model as a surrogate of the CFD model, using the results of simulations in Chapter 6 in 2D and 3D micro-fin tubes. Two different LS-SVR models were employed to estimate the Colburn j factor as a function of micro-fin geometric variables and the Fanning friction factor as a function of geometric variable and Colburn j factor. The results are discussed using statistical criteria.

7.1 Application of a Data-Mining Method in a Micro-Fin Study

This section explains the LS-SVR method mathematically, discusses the criteria for LS-SVR performance evaluation, and explains the implementation steps for building an LS-SVR model.

7.1.1 LS-SVR Method

Data mining refers to discovering patterns in large datasets using machine learning, statistics, and database systems. The SVR method is a data-mining method proposed initially by Vapnik et al. (1996) and later modified to LS-SVR by Suykens et al. (2002). The LS-SVR presumes that the relation between input and output data is non-linear. This nonlinearity can be converted to linearity in a feature space developed by a non-linear mapping function. Equation (7.1) explains a linear relationship between input and output data in the feature space.

$$\lambda(\chi) = \varpi^T \varphi(\chi) + b \quad (7.1)$$

in which, χ and $\lambda(\chi)$ are input and output of a dataset, respectively; ϖ^T = transposed form of weighting vector ϖ , φ is a non-linear function that maps data from the original space to the feature space, and b is a bias. In the following, an optimization problem can be written as Equation (7.2) using the structural risk minimization principle to find the values of ϖ and b .

$$\begin{aligned} \text{Min } J_l(\varpi, \xi) &= \frac{1}{2} \varpi^T \varpi + \frac{1}{2} \gamma \sum_{n=1}^N \xi_n^2 \\ \text{Subject to} & \\ \lambda_n &= \varpi^T \varphi(\chi) + b + \xi_n \quad n = 1, 2, \dots, N \end{aligned} \quad (7.2)$$

where, $J_l(\varpi, \xi)$ is a loss function; n is data counter that varies from 1 to N ; ξ is an error for each data; and γ is a regularization parameter which specifies the trade-off between the training error minimization and smoothness of the estimated function. The solution of the optimization problem relies on constructing a Lagrange function based on the objective function and constraints follows:

$$\begin{aligned} LA(\varpi, b, \xi_n, l_n) &= J_l(\varpi, \xi_n) - \sum_{n=1}^N l_n (\varpi^T \varphi(\chi_n) + b + \xi_n - \lambda_n) \\ n &= 1, 2, \dots, N \end{aligned} \quad (7.3)$$

where $LA(\varpi, b, \xi_n, l_n)$ is Lagrange function and l_n is a Lagrange multiplier. The unknown Lagrange multipliers are obtained by taking partial derivation of the Lagrange function with respect to ϖ , b , ξ_n , and l_n making the expressions equal to zero.

$$\begin{aligned}
\frac{\partial LA}{\partial \varpi} &= 0 \rightarrow \varpi = \sum_{n=1}^T l_n \varphi(\chi_n) \\
\frac{\partial LA}{\partial b} &= 0 \rightarrow \sum_{n=1}^T l_n = 0 \\
\frac{\partial LA}{\partial \xi_n} &= 0 \rightarrow l_n = \gamma \xi_n \\
\frac{\partial LA}{\partial l_n} &= 0 \rightarrow \varpi^T \varphi(\chi_n) + b + \xi_n - \lambda_n = 0 \\
\forall n &= 1, 2, \dots, N
\end{aligned} \tag{7.4}$$

Eventually, the extrema of the Lagrange function is calculated by substituting Equation (7.4) expressions in Equation (7.3). Based on Mercer's theorem, a Kernel function is selected for the unknown non-linear function of $\varphi(\chi)$ as follows:

$$K_f(\chi, \chi_n) = \langle \varphi(\chi), \varphi(\chi_n) \rangle \quad n = 1, 2, \dots, N \tag{7.5}$$

where K_f is a Kernel function and $\langle \cdot, \cdot \rangle$ signifies inner product. As a result, the output λ is estimated by the LS-SVR method is given in Equation (7.6).

$$\lambda_{est}(\chi) = \sum_{n=1}^N l_n K_f(\chi_n, \chi) + b \quad n = 1, 2, \dots, N \tag{7.6}$$

in which λ_{est} is an estimated value of λ . The Kernel Function can be chosen from linear, polynomial, Sigmoid, radial basis (RBF), and MLP functions. The RBF Kernel is defined by Equation (7-8).

$$K_f(\chi, \chi_n) = \exp\left(-\frac{\|\chi - \chi_n\|^2}{2\sigma^2}\right) \quad n = 1, 2, \dots, N \tag{7.7}$$

where σ is a regularization parameter and $\|\chi - \chi_n\|^2$ is a squared Euclidean distance between two input vectors.

7.1.2 LS-SVR Performance Criteria

The performance of LS-SVR method dramatically depends on the quality and quantity of the datasets. Evaluating the performance of LS-SVR requires splitting the datasets into training (calibration) and testing data. The LS-SVR performance is optimum only if the error terms in the training dataset follow a normal probability distribution. Training dataset with heavy-tailed errors or outliers degrades or entirely fails the solution of LS-SVR since the method's main attempt is to minimize the loss function defined as the sum of squared errors (*SSE*), which overemphasizes the impact of outliers. Another problem that highly affects the performance of LS-SVR method is overfitting training data. It happens when random noise is captured and learned as if it represents the underlying model structure. There are some ways to avoid overfitting, such as applying *k*-fold cross-validation (a resampling procedure) on data and increasing training data size. The predictive capability of data-mining models is remarkably affected by the ratio of training to testing data as well. Nguyen et al. (2021) investigated the performance of different data-mining models with various training to testing data ratios. They observed that the 70/30 ratio brings the optimal performance for the models. The LS-SVR method precision is greatly dependent on the selection of regularization parameters of γ and σ . One typical way to choose the regularization parameters is to analyze the model performance using statistical criteria. The general statistical criteria applied to evaluate model performance are root mean squared error (*RMSE*), R^2 , Nash-Sutcliffe (*NS*), maximum absolute error (*MAE*), and *AAE*.

7.1.3 Implementation of the LS-SVR Model

The LS-SVMLab software developed by Suykens et al. (2002) is employed to implement the LS-SVR method in micro-fin tube study. The implementation steps to build the LS-SVR model are illuminated in Figure 7.1.

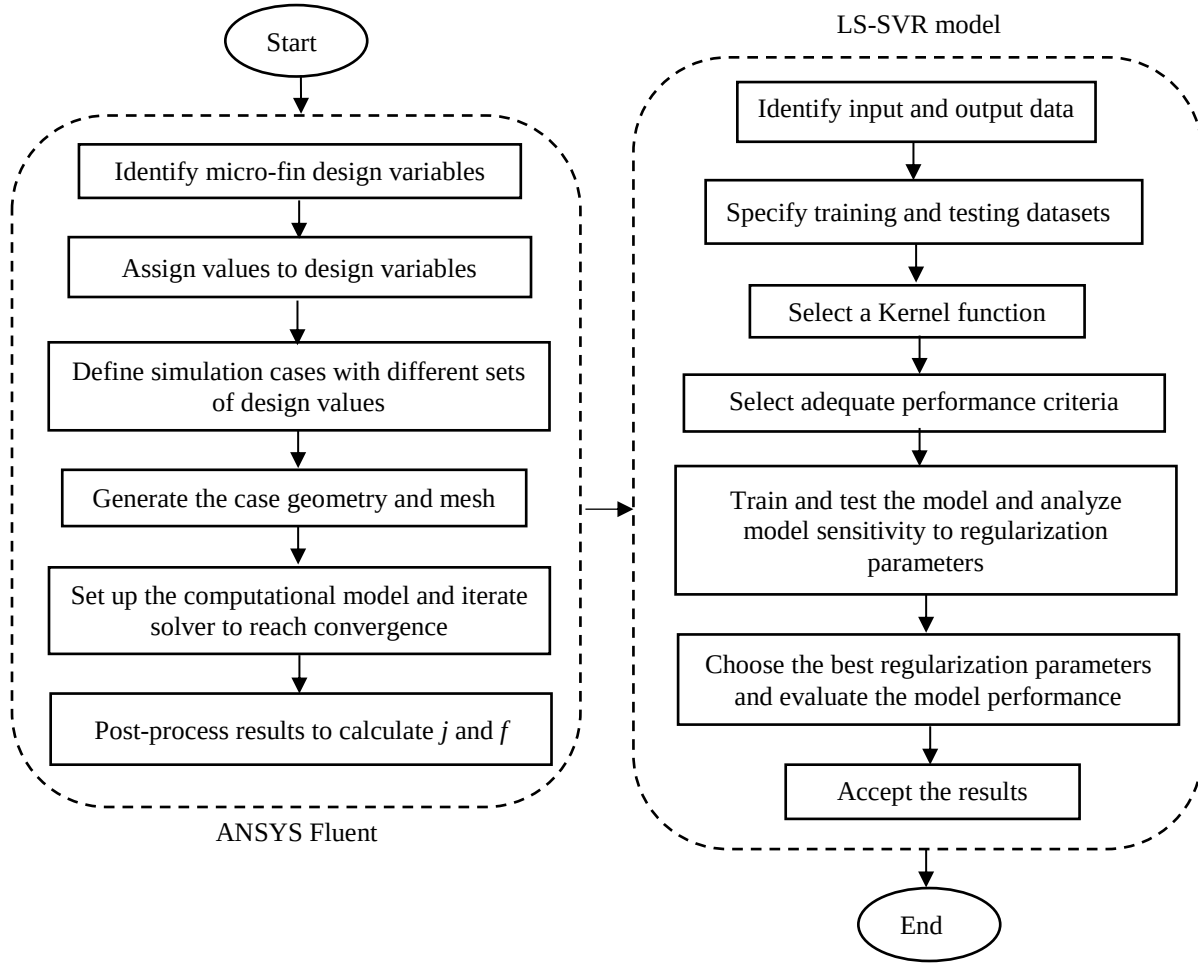


Figure 7.1. Flowchart of the LS-SVR model implementation

The input and output of the LS-SVR model are illuminated in Table 7.1. This study used the advantage of the strong relation between j and f with $R^2 = 0.67$ to improve the estimation of f . Therefore, data of the simulated j and estimated j (j_{est}) was chosen for the training and testing dataset input, respectively, in addition to the geometric variables of 2D and 3D micro-fin tubes and the A_{en}/A_s ratio.

Table 7.1. Input and output of LS-SVR method for training and testing datasets for 2D and 3D micro-fin tubes

Micro-fin type	Dataset	Model input							Model output
2D	Train	e	α	N_f			A_{en}/A_s	j	j_{est}
		e	α	N_f			A_{en}/A_s	j	f_{est}
	Test	e	α	N_f			A_{en}/A_s		j_{est}
		e	α	N_f			A_{en}/A_s	j_{est}	f_{est}
3D	Train	e	α	N_f	N_c	t_c	A_{en}/A_s	j	j_{est}
		e	α	N_f	N_c	t_c	A_{en}/A_s	j	f_{est}
	Test	e	α	N_f	N_c	t_c	A_{en}/A_s		j_{est}
		e	α	N_f	N_c	t_c	A_{en}/A_s	j_{est}	f_{est}

The results of 44 (2D micro-fin) and 166 (3D micro-fin) simulation cases were used to train and test the LS-SVR method. This study used the ratio of 70/30 for the ratio of training to testing data. Testing data for the 2D micro-fin tubes were taken from simulation results of commercial tubes reported by Webb et al. [30] and one of the best micro-fin tubes ($e = 0.291\text{mm}$, $\alpha = 51.2$, and $N_f = 61$) reported by Mann and Eckels (2019). Based on random selection for 3D micro-fin modeling, training and testing data were chosen. The *RMSE*, *MAE*, and *AAE* were applied to evaluate the potential of the *RBF*-based LS-SVR model.

7.2 Result and Discussion

Table 7.2 shows the sensitivity analysis of the LS-SVR model performance on values of regularization parameters of γ and σ for 2D and 3D micro-fin tubes.

Table 7.2. Results of the RMSE for different values of regularization parameters in 2D and 3D micro-fin tubes

Micro-fin tube	Model output	γ	σ	<i>RMSE</i>	
				Train	Test
2D	j	0.5	0.5	0.000638	0.000795
		0.5	5	0.000547	0.000595
		5	5	0.000238	0.000511
		50	5	0.000120	0.000493
		5,000	2	0.000001	0.000401
		50,000	20	0.000086	0.000362
		50,000	15	0.000067	0.000322
		500	5	0.000047	0.000318
		5,000	5	0.000014	0.000298
		5,000	10	0.000046	0.000286
		5,000	7	0.000029	0.000261
		0.5	1	0.001306	0.000651
	f	0.9	5	0.000987	0.000515
		0.9	1	0.000963	0.000490
		500	3	0.000017	0.000375
		2	5	0.000737	0.000347
		500	10	0.000110	0.000338
		2	4	0.000689	0.000322
		2	3	0.000636	0.000322
		500	5	0.000037	0.000319
		1,000	5	0.000021	0.000319
		500	6	0.000049	0.000317
3D	j	5	7	0.000213	0.000263
		5,000	2	0.000001	0.000258
		500	10	0.000081	0.000210
		500	7	0.000061	0.000201
		5,000	15	0.000064	0.000197
		50,000	10	0.000019	0.000194
		5,000	7	0.000028	0.000191
		500	10	0.000138	0.000461
	f	5,000	15	0.000092	0.000451
		500	4	0.000031	0.000440
		500	8	0.000103	0.000437
		5,000	8	0.000047	0.000434
		5,000	5	0.000008	0.000430

According to Table 7.2, for 2D micro-fin tubes, the best values of γ and σ in the testing dataset are 5,000 and 7 to estimate the j , respectively. The best values of γ and σ in the testing dataset are 500 and 6 to estimate the f , respectively. For 3D micro-fin tubes, the *RMSE* of the testing dataset is minimum at γ and σ equal to 5,000 and 7 to estimate the j , respectively. The parameters of γ and σ equal to 5,000 and 8 were selected to estimate the f , respectively. The

chosen values avoid overfitting the training dataset and guarantee the method generalization ability to new datasets.

Figure 7.2 portrays the simulated and estimated j and f for training and testing datasets for 2D and 3D micro-fin tubes using Fluent and LS-SVR models.

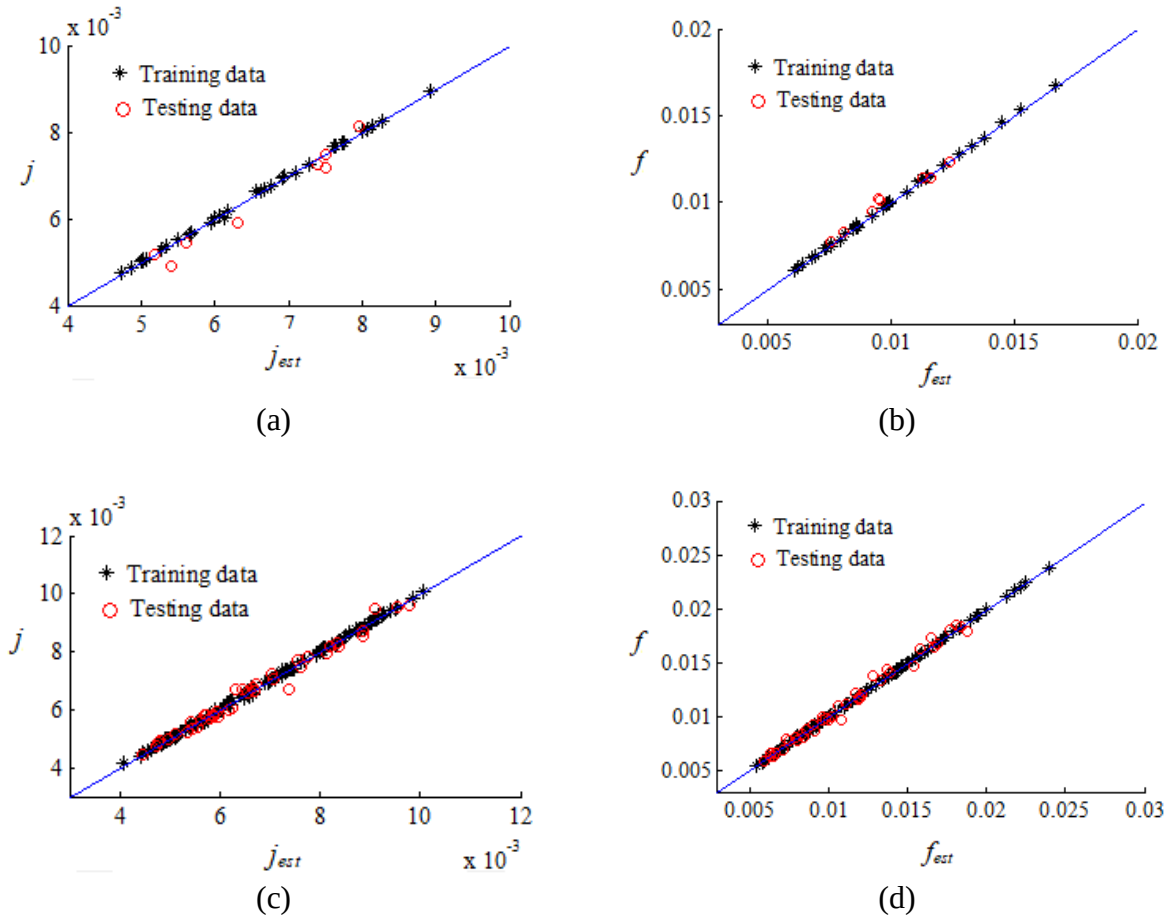


Figure 7.2. Simulated and estimated (a) j and (b) f in 2D micro-fin tubes and simulated and estimated (c) j and (d) f in 3D micro-fin tubes

The LS-SVR estimations disclose some errors, but the method performance is outstanding overall for both 2D and 3D micro-fin tubes. The simulation running time in the Fluent model for either 2D or 3D micro-fin tubes is about 20 minutes which does not include the time for geometry

and mesh generation. The calculation time for the LS-SVR method is less than a second to estimate a new output which is about 900 times faster than the Fluent model.

Table 7.3 shows the LS-SVR model performance in terms of statistical criteria for training and testing datasets in 2D and 3D micro-fin tubes. The model performance to estimate the Fanning friction factor is better than the model performance to estimate j based on results of $RMSE$ for 2D and 3D micro-fin tubes. The MAE of j and f are greater in 3D micro-fin tubes than 2D micro-fin tubes. The largest MAE of 11.1% is for the testing dataset with the output of f in 3D micro-fin tubes. The largest AAE of 3.84% is for the testing dataset with the output of j in 2D micro-fin tubes. The results of statistical criteria indicated that the LS-SVR model is efficient in estimating j and f in 2D and 3D micro-fin tubes with an associated reduction in the computational time reduction.

Table 7.3. Calculated statistical results with LS-SVR model for 2D and 3D micro-fin tubes

Micro-fin tube	Model output	$RMSE$		MAE (%)		AAE (%)	
		train	test	train	test	train	test
2D	j	0.000029	0.000261	1.50	10.80	0.33	3.84
	f	0.000049	0.000317	1.30	6.40	0.39	2.47
3D	j	0.000045	0.000185	2.17	9.97	0.49	2.05
	f	0.000047	0.000436	1.34	11.10	0.35	2.93

8 Optimization Study of 3D Micro-fins on Fluid Flow and Heat Transfer

This study aims at providing a robust automated methodology for multi-objective optimization of 3D micro-fins in a fluid flow channel with a novel computational domain. The micro-fin design variables are micro-fin height, angle, thickness, pitch, length of discontinuity, and number of discontinuity per micro-fin pitch. The channel contains two-sided periodic domains and micro-fins on the top and bottom walls having an opposite angle. NSGA-II is used to achieve the best trade-off (Pareto front) solutions between enhancement of Nusselt number and friction factor of a micro-fin surface to a smooth surface at $Re = 49,000$. The objective function evaluations are calculated using CFD simulations.

8.1 Problem Description

Optimization can be expressed as the task of achieving the optimal configuration for a system having a specified number of degrees of freedom and subjected to certain constraints, while degrees of freedom in a system are equal to decision design variables. The mathematical formulation of the study optimization problem with two objective functions and constraints is defined as:

$$\begin{aligned}
Max F_1(X) &= \frac{Nu}{Nu_s} \Big|_{Re}, \quad X = (e, \alpha, t_f, p_f, t_c, \delta) \\
Min F_2(X) &= \frac{f}{f_s} \Big|_{Re} \\
\text{subject to :} & \\
\left\{ \begin{aligned}
0.05 &\leq e \leq 0.5 \\
20 &\leq \alpha \leq 65 \\
0.1 &\leq t_f \leq 0.5 \\
0.15 &\leq p_f \leq 3 \\
0.1 &\leq t_c \leq 0.8 \\
\delta &= 0.33, 0.5, 1, 2, 3, 4, 5
\end{aligned} \right. \quad (8.1)
\end{aligned}$$

in which X is a set of decision variables, and the subscript of s refers to the smooth channel.

8.1.1 Design Variables

The optimization study requires a representation of the geometry of 3D micro-fins with enough design variables to encompass a wide range of physically realizable surface geometries. The design variables of the study are divided into the decision, state, and constant variables. A design variable function can be continuous, discrete, or a combination of both. The decision variables are the independent micro-fin design variables that are unknown in an optimization problem. State variables are the type of dependent design variables formulated based on decision variables. Table 8.1 lists the state constant design variables of the study.

Table 8.1. Definition of state and constant design variables of the study

Type of variable	Symbol	Description	Function	Value
State	p_c	Cutter pitch (mm)	Continuous	$p_c = p_f / \delta$, $0.03 \leq p_c \leq 9$
	L	Channel length (mm)	Discrete	$L = p_c / \sin(\alpha)$ for $p_f \leq p_c$
				$L = p_f / \sin(\alpha)$ for $p_f > p_c$
	W	Channel width (mm)	Discrete	$W = p_c / \cos(\alpha)$ for $p_f \leq p_c$
				$W = p_f / \cos(\alpha)$ for $p_f > p_c$
Constant	e_c	Cutter height (mm)	Discrete	$e_c = 0.7$
	D_i	Channel inside thickness (mm)	Discrete	$D_i = 15.54$
	D_o	Channel outside thickness (mm)	Discrete	$D_i = 19$

The problem design variables are shown in Figure 8.1. To make a 3D micro-fin configuration, micro-fins are cut through their length by cutters. Specifically, the 3D fin starts with a base 2D structure and then removes fin material by crosscutting in a manner similar to the actual manufacturing process. Since the periodic boundary condition in Fluent requires a pure translation between two periodic walls, the domain length and width are calculated to enforce translation symmetry on inlet and outlet and sidewalls. The results of the micro-fin parametric study reported in Chapter 5 showed that a micro-fin cross profile does not have a significant impact on heat transfer and fluid flow characteristics. Hence, this study uses a rectangular cross-section for micro-fins and cutters.

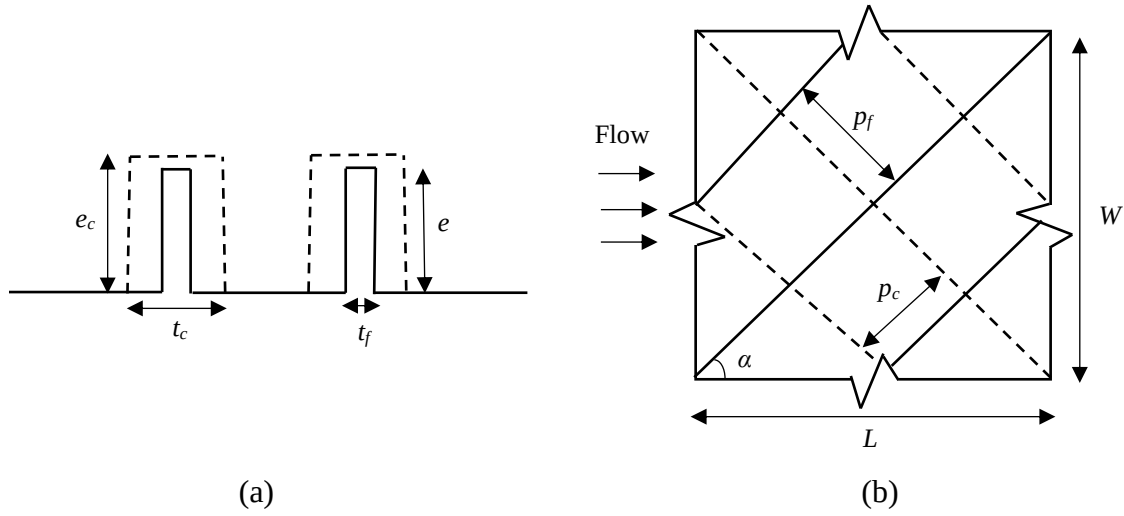


Figure 8.1. A schematic of (a) micro-fin and cutter profiles and (b) a bottom wall of a channel with micro-fins and cutters (solid and dash lines represent micro-fins and cutters, respectively)

8.2 Geometry and Boundary Condition

A geometric representation of a 3D micro-fin channel with defined boundary conditions is shown in Figure 8.2.

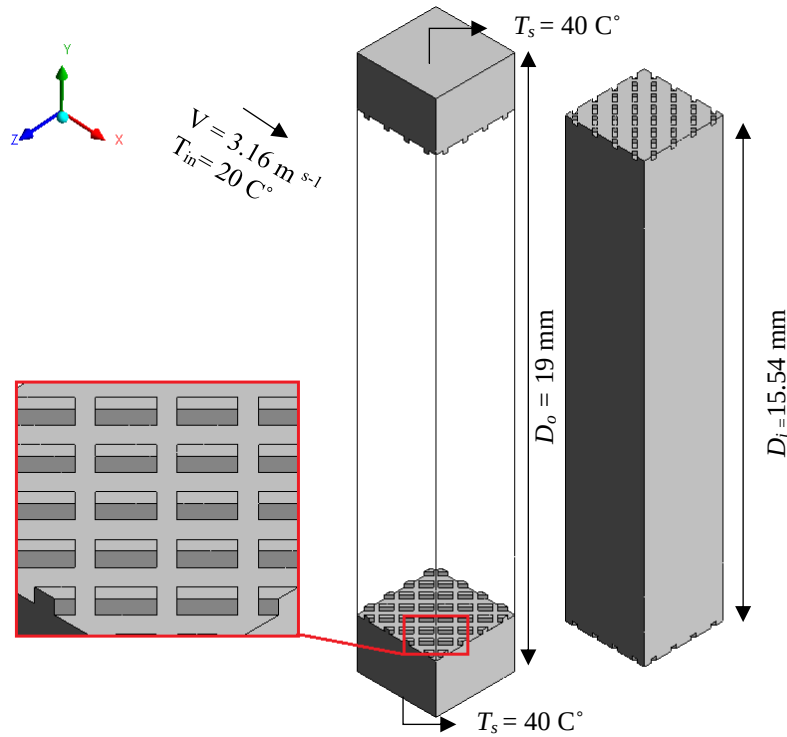


Figure 8.2. Geometric representation of solid and fluid sections of a channel with 3D micro-fins

The problem is defined as fluid enters the channel with constant velocity ($V = 3.16 \text{ m s}^{-1}$) with entering temperature of $T_{in} = 20 \text{ C}^\circ$. A constant and uniform wall temperature of $T_s = 40 \text{ C}^\circ$ is defined at the outer walls of the solid surface.

Since the domains are periodic, a series of simulations were conducted to evaluate the convergence of results to study computational domain length and width independence. Table 8.2 shows the results of simulations by varying the periodic domain length and width for a micro-fin channel at $Re = 49,000$.

Table 8.2. Results of domain independency study (errors are in respect to 20 repetitions for length and width domains)

Number of width domain	Number of length domain	Width (mm)	Length (mm)	Nu	f	Error (%)	
						Nu	f
2	2	1.41	1.41	324.57	0.00815	0.03	0.11
3	3	2.12	2.12	325.39	0.00817	0.28	0.38
4	4	2.83	2.83	323.46	0.00815	-0.31	0.05
3	4	2.12	2.83	324.36	0.00815	-0.03	0.09
4	3	2.83	2.12	324.63	0.00817	0.05	0.31
2	4	1.41	2.83	324.48	0.00813	0.00	-0.11
4	20	2.83	14.14	324.84	0.00814	0.11	-0.06
20	4	14.14	2.83	324.31	0.00815	-0.05	0.09
20	20	14.14	14.14	324.47	0.00814	-	-

According to Table 8.2, all of the results are within 0.5% deviation from the results of the simulation case with 20 repetitions for width and length domains. Hence, the periodic domain length and width are independent of the Nu and f . The results for cases with a short length and width domain (less than 1.4 mm) did not converge. Therefore, to make sure the convergence is guaranteed for the range of domain length and width shown in Table 8-2, a set of conditions were defined as shown in Equation (8.2) to set the minimum length and width of micro-fin channels greater than 2 mm.

$$\begin{aligned}
&\text{if } L \leq 0.5 \text{ mm then } L = 6L \quad \text{or} \quad \text{if } W \leq 0.5 \text{ then } W = 6W \\
&\text{if } L \leq 1 \text{ mm then } L = 4L \quad \text{or} \quad \text{if } W \leq 1 \text{ then } W = 4W \\
&\text{if } L \leq 2.5 \text{ mm then } L = 2L \quad \text{or} \quad \text{if } W \leq 2.5 \text{ then } W = 2W
\end{aligned} \tag{8.2}$$

8.3 Meshing and Independence Study

As shown in Figure 8.3, an unstructured tetrahedral grid system is applied for both smooth and micro-fin channels. To specify an appropriate grid size for the numerical simulations, a mesh independence study was carried out for a smooth and a micro-finned channel at $Re = 49,000$.

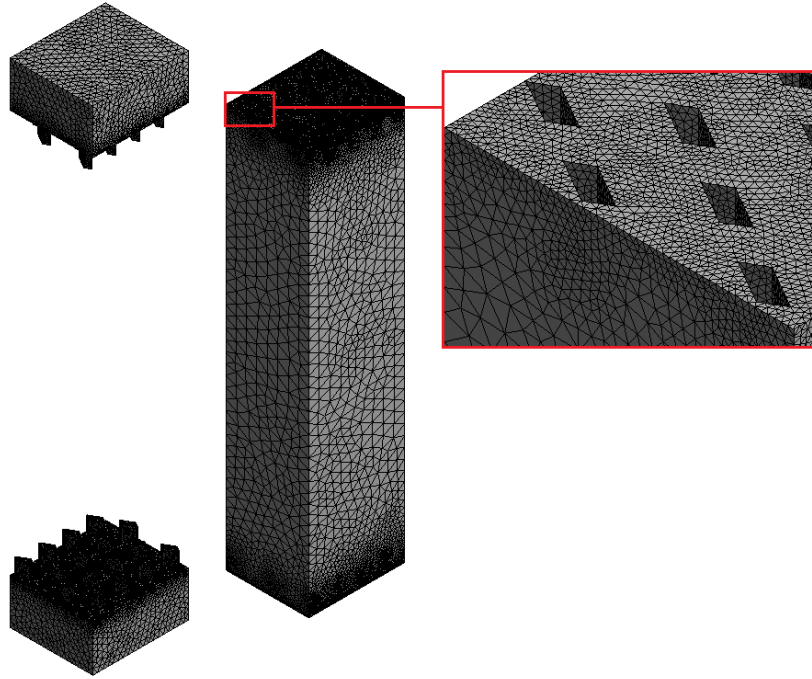


Figure 8.3. Mesh system of solid and fluid domains with a zoom-in view at the solid-fluid interface for a 3D micro-fin channel

Figure 8.4 shows the results of simulation models with different mesh element numbers for a smooth and a micro-fin channel. The independence mesh study was conducted for four different grid systems. The relative deviation of Nu and f between the grid system with 10^5 and 3.5×10^5 elements is less than 1% and 2.3% for the smooth channel, respectively. The relative deviation of

Nu and f between the grid system with 3.4×10^5 and 7.2×10^5 elements is less than 2.6% and 1% for the micro-fin channel, respectively. Therefore, this study chose the grid system with 10^5 and 3.4×10^5 elements for the smooth and micro-fin channels, respectively.

Due to the final grid system, the grid size for the solid and fluid domains are 0.2 and 0.4 mm, respectfully. The grid size at the solid-fluid interface is 0.06 mm, which leads to near-wall first layer grids with a height of less than 0.06 mm. The grid system satisfies the proper near-wall mesh resolution $y^+ = 1$ for the Fluent turbulence model to resolve the viscous sublayer and capture the velocity and temperature gradient at the wall.

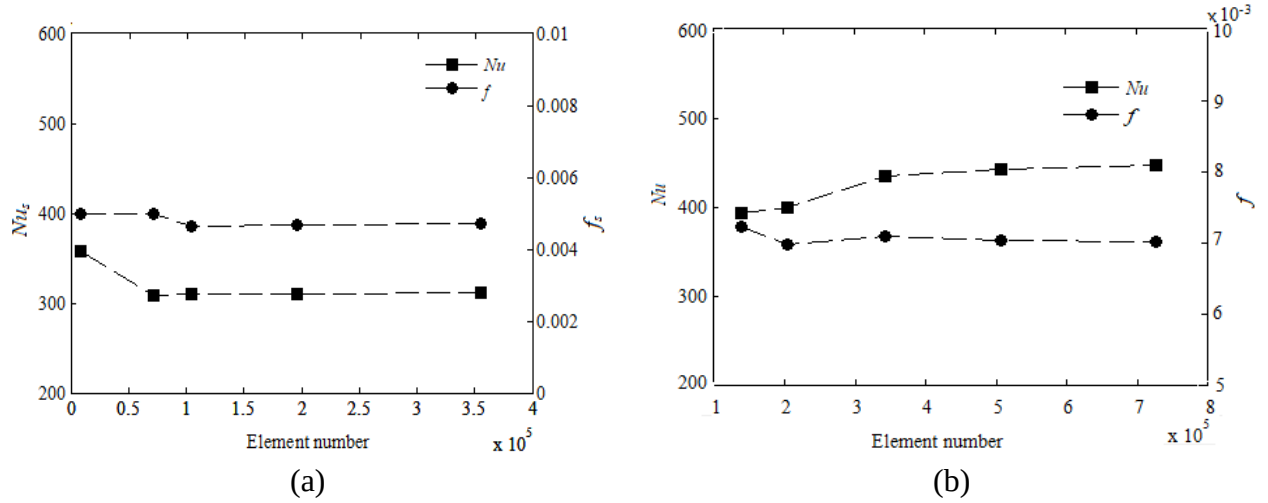


Figure 8.4. Mesh independency study for a (a) smooth and (b) micro-fin channel

8.4 Optimization Study

Evolutionary and meta-heuristic algorithms can solve both continuous and discrete decision variable functions. Since the evolutionary algorithms are based on an iterative procedure, they enable applying parallel computation in CFD-based optimization problems. Besides, evolutionary algorithms let the optimization procedure continue even if some simulations fail due to possible incompatibility of geometry or mesh generation or lack of numerical convergence.

To implement the CFD-based optimization for the study, ANSYS Workbench 19.1 is coupled with NSGAI algorithm through a Python Library developed initially by Mann and Eckels (2019). For the sake of simplicity, the computational study framework is divided into the optimization framework and simulation framework as discussed in the following.

8.4.1 NSGAI Framework

NSGAI is inspired by the theory of evolution and was introduced by Deb et al. (2002). The concept of this algorithm is based on non-dominated sorting and elitist selection theory. A solution is non-dominated if no other feasible solutions can improve one of the objectives without diminishing another. This study performed NSGAI with the implementation of Inspyred Python Library.

Figure 8.5 illustrates the optimization framework employed to implement NSGAI. At first, a set of random individuals called the parent population is generated. The parent population contains many chromosomes, whereby each chromosome is a solution and contains many genes that are the decision variables. Then objective functions are evaluated using the study simulation model for each solution corresponding to the parent population. Parents are ranked as follows: the objective values of the total population are presented by sets called Pareto fronts, such that the objective values with the first rank are called the first front. Then, a crowding distance magnitude is computed for members with a specific front in Pareto fronts for diversity presentation. The crowded distance comparison guides the selection process at the various stages of the algorithm pushing towards a uniformly spread-out Pareto front. As a result, each member on Pareto front has two indexes one for rank and one for crowding distance magnitude. According to the index number, total population is cut to send to the next generation (iteration) as parent population. In

the next step, the offspring population is generated based on the parent population using well-known GA operators named selection, crossover, and mutation.

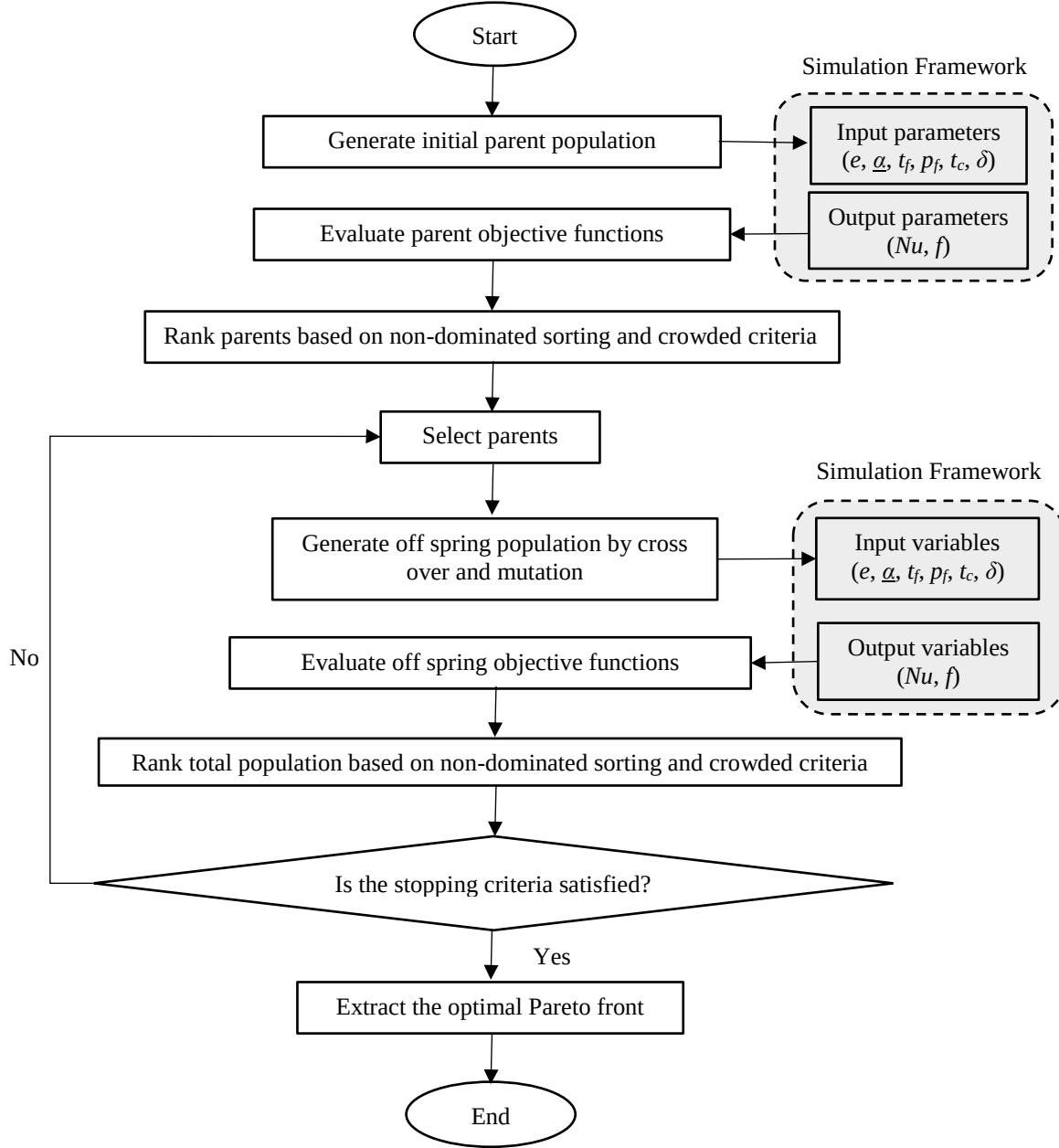


Figure 8.5. An optimization framework for implementing NSGAI

Figure 8.6 illustrates the beginning procedure of NSGII algorithm. Accordingly, R_0 and R_1 are the initial parent population and parent population at the second iteration, respectively; $\mathcal{Q}_1$ is

the offspring population at the second iteration, and Fr represents a non-dominated front. For example, $Fr^{(1)}$ is the first front. According to Figure 8.5, the objective values of the offspring population are evaluated through the simulation framework. Next, the parent and offspring populations are combined and sorted based on the ranking process. The iteration is terminated if the solution converges to an optimum. At this time, the algorithm checks if the stopping criterion is satisfied or not. The algorithm provides the optimal Pareto front if the stopping criterion is satisfied. If this is not satisfied, the algorithm goes back to the parent selection from the current population and continues the further steps until the stopping criterion is satisfied.

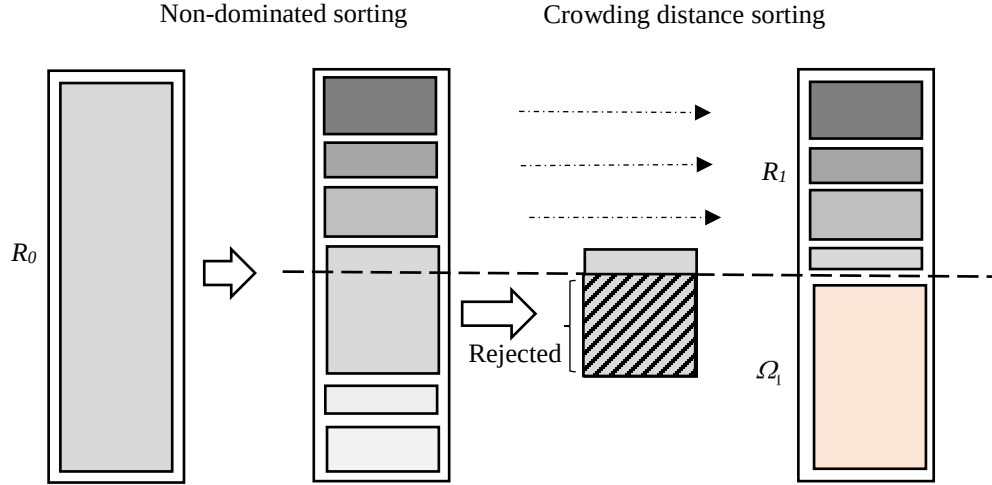


Figure 8.6. NSGAII flowchart at first iteration

8.4.2 Simulation Framework

To quickly and cost-effectively perform Fluent simulations, this study used Beocat to perform Fluent simulations. Figure 8.7 depicts the simulation framework of the study.

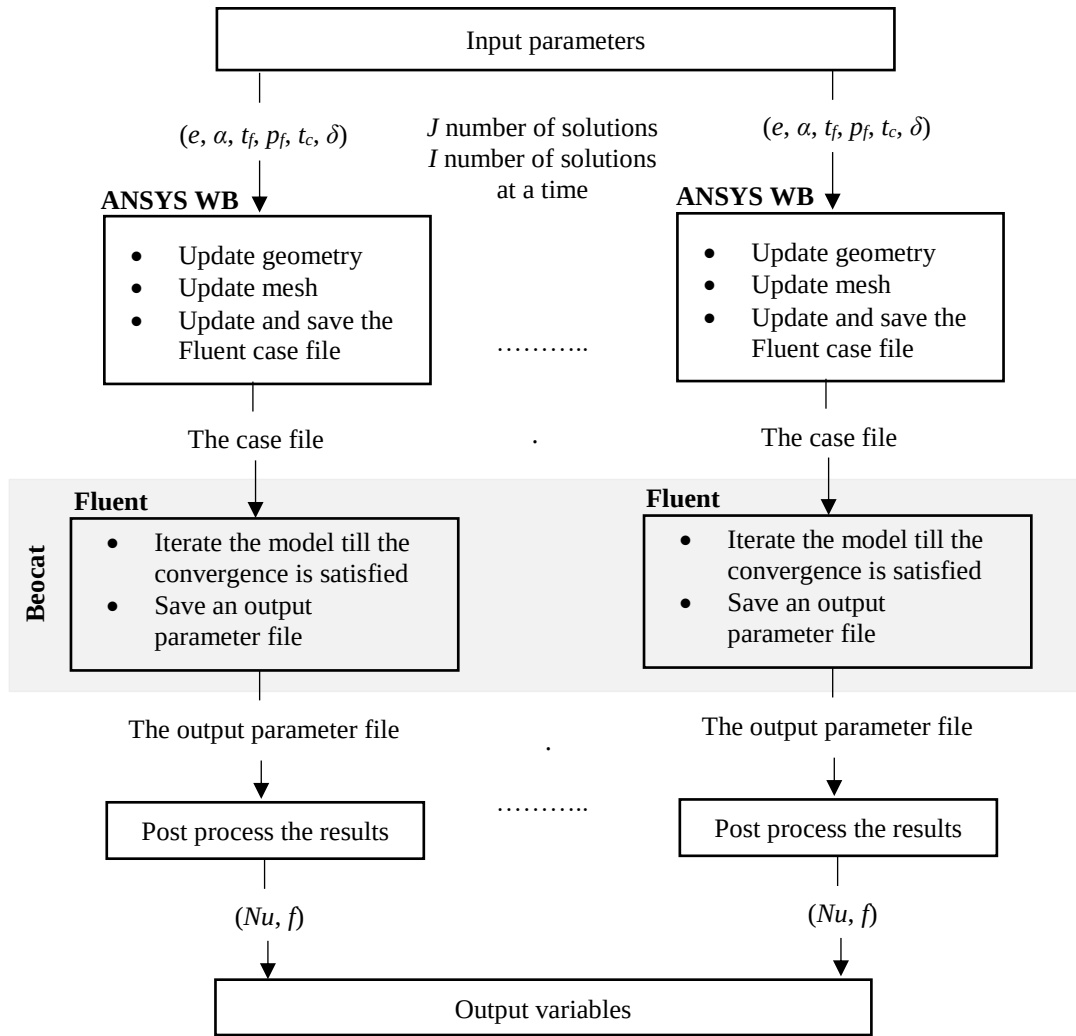


Figure 8.7. Simulation framework

According to Figure 8.5 and Figure 8.7, at each optimization iteration, J number of individual solutions with I number at a time in a parallel platform are sent to be simulated. A Python code is written to automatically read solutions and generate geometries using Space Claim, 19.1. The updated geometries are sent to ANSYS Meshing to be updated, then the Fluent case files are written. The study algorithm sends the case files to Beocat to iterate the solver. Then, output parameter files are written for individual simulations with satisfied model convergence. Finally, the study algorithm reads the output parameters to calculate and accumulate Nu numbers and f

factors. Output parameters are not written for individual simulations that failed because of meshing or convergence. In this condition, a fixed unreal low Nu number (-999) and high f factor (999) are assigned for the failed cases, thus they are automatically removed from the optimization process because of the bad objective values.

8.5 Result and Discussion

The stopping criterion for the NSGAII algorithm was selected as the number of iterations. The algorithm ran for 52 iterations (starting from zero). In the computation, the number of individual solutions was selected to be 140 for each optimization iteration, in which 14 individual solutions were simultaneously simulated in a parallel platform. The computational time of 53 iterations was ten days for a total number of simulations of 7,420, with an average of about two minutes per simulation. Only 21 simulations (less than 2.7%) failed to converge from total simulations. In the computation, The Fluent solver for each solution was parallelized by 16 processes, thus this study used the total of 224 cores (14 simulations at a time times 16 processes).

The initial parent population for NSGAII was selected using uniform sampling from the decision variable range. The GA operators were tournament selection (with tournament size of 2), blend crossover with a 10% blend rate, and Gaussian mutation with 10% rate of mutation from a standard normal. Figure 8.8 depicts sampling from feasible decision variable space during NSGAII algorithm progress. Accordingly, the initial population was equally distributed throughout the decision space. During algorithm progress, only samples of the micro-fin angle within the range of 25 to 44 were carried out to the last iteration. For micro-fin width, population elites within the range of 0.38 to 0.5 mm were sent to the last iteration. The sample range for micro-fin height does not change throughout the iterations. In Figure 8.8.b, it is clear that elite points at the last iteration are within the range of 0.54 to 1.21 mm for micro-fin pitch and 0.1 to 0.19 mm for the discontinuity

length. The samples of number of discontinuity per micro-fin pitch after 52 iterations were mostly converged to five discontinuities.

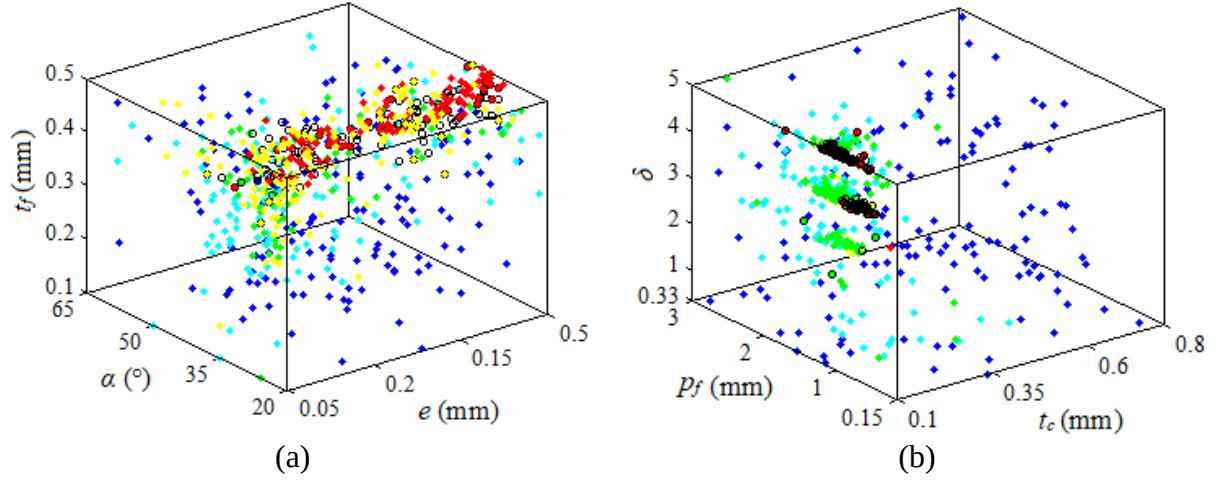


Figure 8.8. Scattering of micro-fin (a) height, angle, pitch and (b) width, length of discontinuity, and number of discontinuities per micro-fin pitch at the initial population (blue dots), 10th (cyan dots), 20th (green dots), 30th (yellow dots), 40th (unfilled black dots), and 52nd (red dots) iteration

Figure 8.9 shows the convergence of objective functions of the NSGAI algorithm. The objective function of Nu/Nu_s attains the maximum value of 5.36 at the 6th iteration and the objective function of f/f_s attains the minimum value of 0.92 at the 33rd iteration. After reaching the maximum value, the objective function values remain constant till the last iteration happens for both objective values, which is one way to show the algorithm convergence.

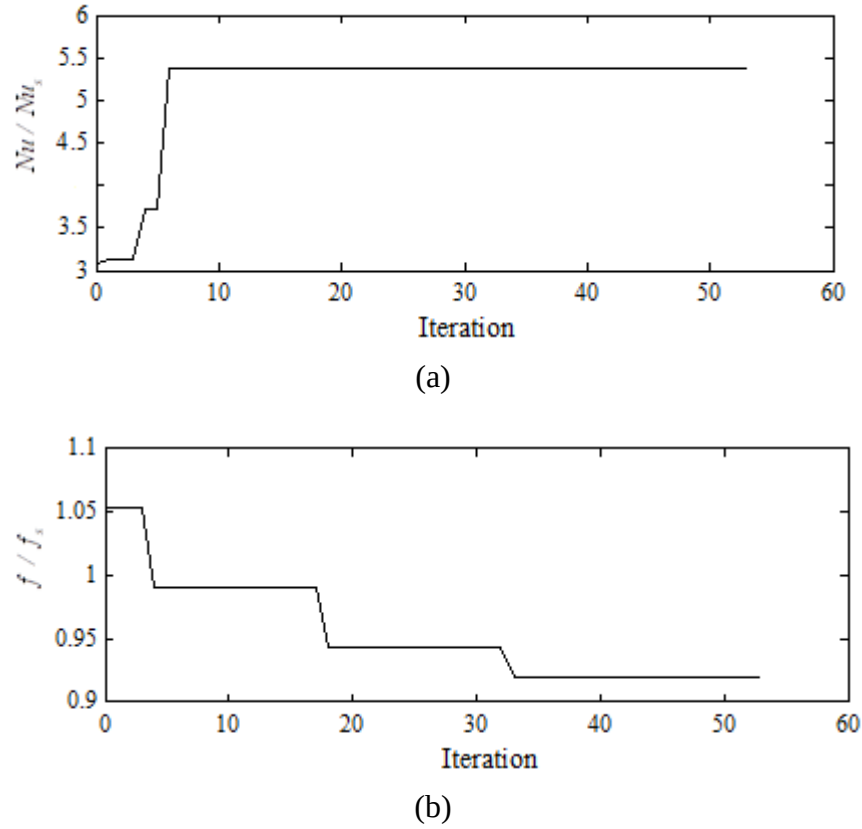


Figure 8.9. Convergence of objective functions (a) Nu/Nu_s and (b) f/f_s over iteration using of NSGAI algorithm

Figure 8.10 shows the distribution of Pareto optimal points using NSGA-II. The solid line shows the thermo-hydraulic efficiency index when heat transfer enhancement is exactly equal to the pressure drop increase. The optimal points on the Pareto front have a thermo-hydraulic efficiency index of greater than 1.3. Based on different PEC considered by engineers, different final solutions among the optimal points existing on the Pareto front can be selected. This study identified three optimal design points of interest on the Pareto front named A, B, and C aimed at minimizing pressure drop ($f/f_s = 0.92$), maximizing the thermo-hydraulic efficiency index ($\eta = 1.57$), and maximizing heat transfer enhancement ($Nu/Nu_s = 3.16$), respectively.

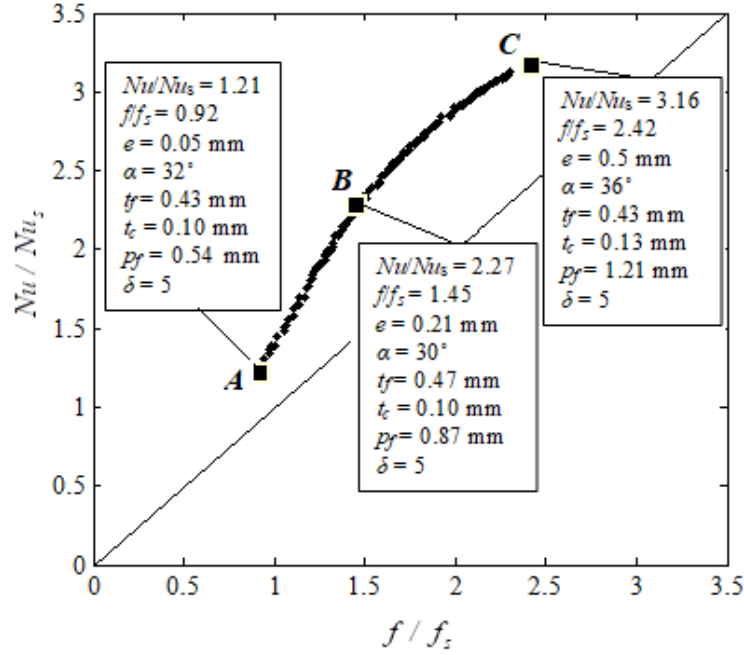


Figure 8.10. NSGA-II Pareto front after 52 iterations

In order to compare the results of this study with other numerical micro-fin tube studies, the micro-fin variables of design point *B* were used on a tube heat exchanger. The required information about the geometry, mesh, and simulation set up of the generated 3D micro-fin tube has been reported in Chapter 6. Table 8.3 shows the comparison of heat transfer enhancement, pressure drop increase, and the thermo-hydraulic efficiency index for the generated micro-fin tube of this study with the best reported micro-fin tubes in the literature. Accordingly, the thermo-hydraulic efficiency index of the micro-fin tube of this study is 18% greater compared to the optimum 2D micro-fin tube of Mann and Eckels (2019). It should be noted that in the study of Mann and Eckels (2019), solid-fluid interface assumed to have a constant temperature which yields unity fin efficiency and higher heat transfer enhancement. In this study, the fin efficiency is always less than one due to temperature distribution in the fins. The fin efficiency is defined as the ratio of the heat transfer rate of the actual fin to the heat-transfer rate from a conceptual fin with the same surface area, if the whole fin surface were at the same temperature as the fin base.

Table 8.3. Comparison of results of this study with other studies for the best micro-fin tube at $Re = 49,000$ and $D_i = 15.54$ mm with water flow

Dimension	Reference	e/D_i	Ns	p_f/e	α ($^\circ$)	t_c (mm)	δ	Nu/Nu_s	f/f_s	η
3D	Current Chapter	0.0032	51	2.63	30	0.10	5	2.23	1.54	1.45
	Chapter 6	0.0064	60	4.7	60	0.46	1	1.79	1.54	1.16
	Mann and Eckels [25]	0.018	61	2.11	51	-	-	2.18	1.76	1.23
2D	Chapter 6	0.0064	60	4.7	60	-	-	1.68	1.42	1.18
	Webb et al. [42]*	0.034	25	7.87	25	-	-	1.84	1.58	1.16

*Experimental study

Figure 8.11 shows the effects of decision variables on the enhancement of the optimal points on the Pareto front. There is a significant positive correlation between micro-fin height and pitch decision variables and the final enhancement. The strongest correlation ($r = 0.97$) is between micro-fin pitch and each of Nu/Nu_s and f/f_s ratios.

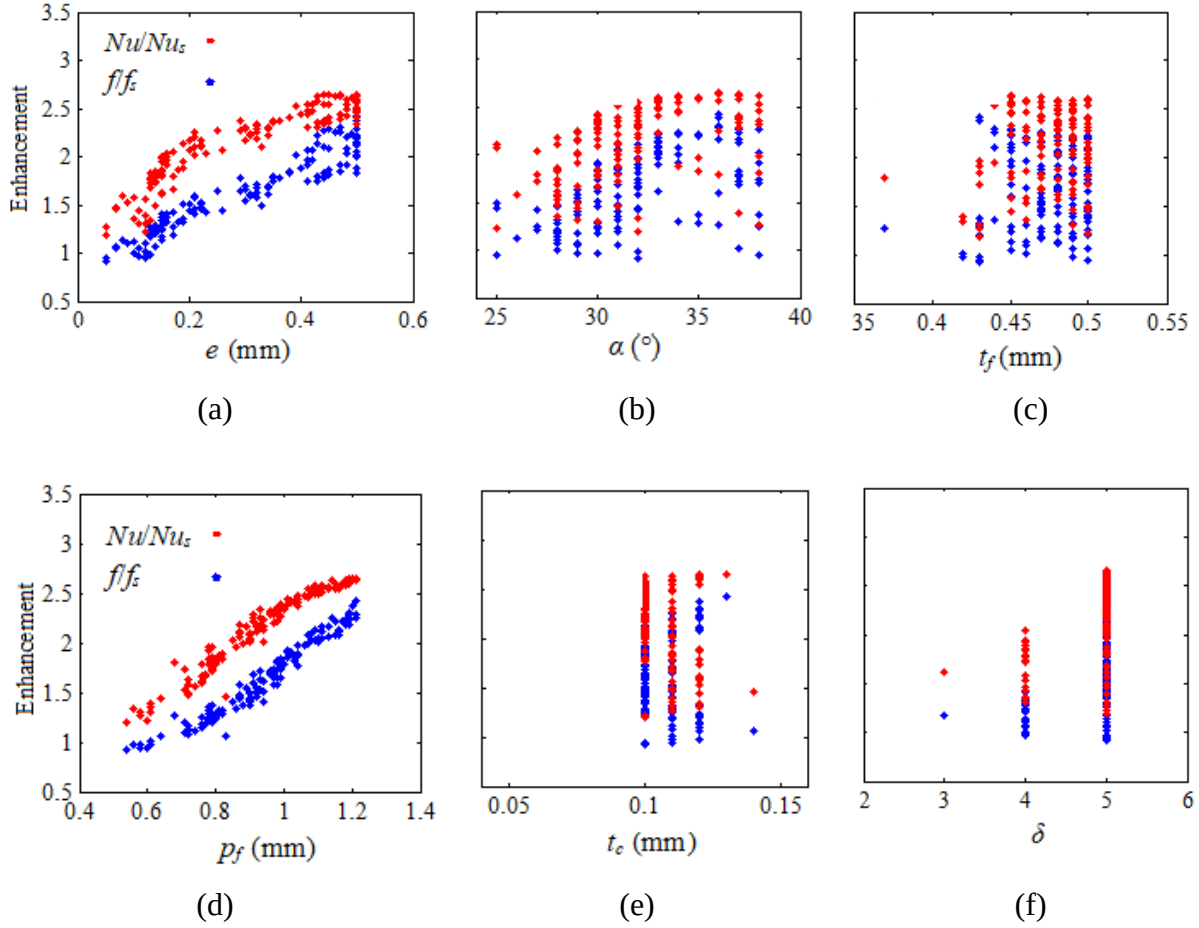


Figure 8.11. Parametrization of the Pareto Front

8.5.1 Velocity and Thermal Fields of the Optimal Design Points

Figure 8.12 shows velocity contours and streamlines above the micro-fin in the cross-midsection of the channels of design points A, B, and C. Like temperature gradient, design point A has the sharpest, and design point C has the mildest velocity gradient among optimal design points. The slope of velocity gradient proves that design points A and C have the lowest and highest f among the optimal design points. Micro-fin configuration of design point A increases Nu compared to the smooth surface and decreases f up to 8%. This characteristic of surface geometry has been widely discussed in the context of drag-reducing riblet surfaces addressed in Chapter 3.

With appropriate surface geometry design, the structure of riblets damps turbulence flow motion near the wall, which causes reducing surface skin friction. According to the results of design point A, this study numerically proves that a particular surface design can reduce friction drag and enhance heat transfer in the water flow regime, which has not been reported before.

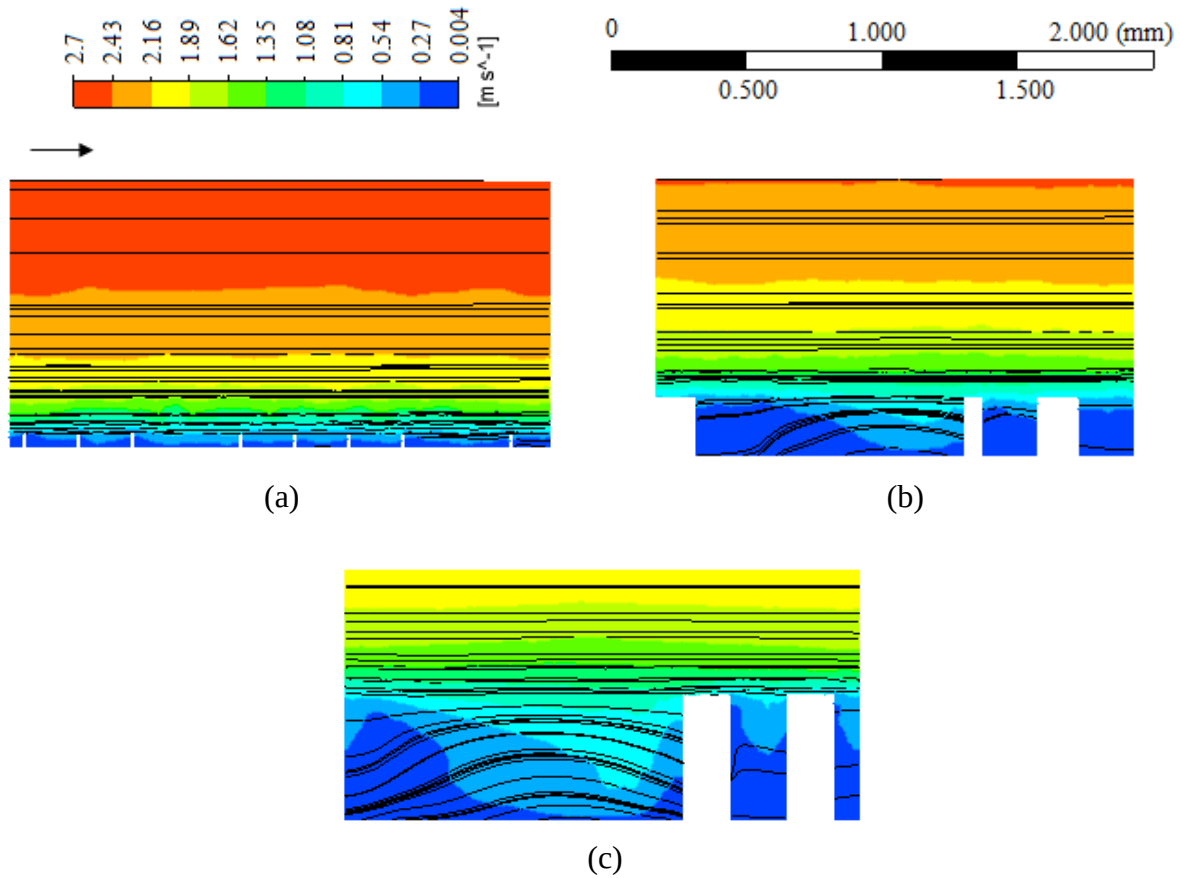


Figure 8.12. Velocity contours and stream-lines above the micro-fin surface at the cross-midsection of the channels with design points (a) A (b) B, and (c) C

Figure 8.13 shows velocity vectors in a micro-fin valley of design points A, B, and C. The feature of surface geometry of design point A damps the turbulence structures near the wall, and no stream-wise vorticity is evident in the micro-fin valleys. The vorticity size in the micro-fin valley of design point C is larger than design point B, which proves design point C has greater values of f and Nu than design point B.

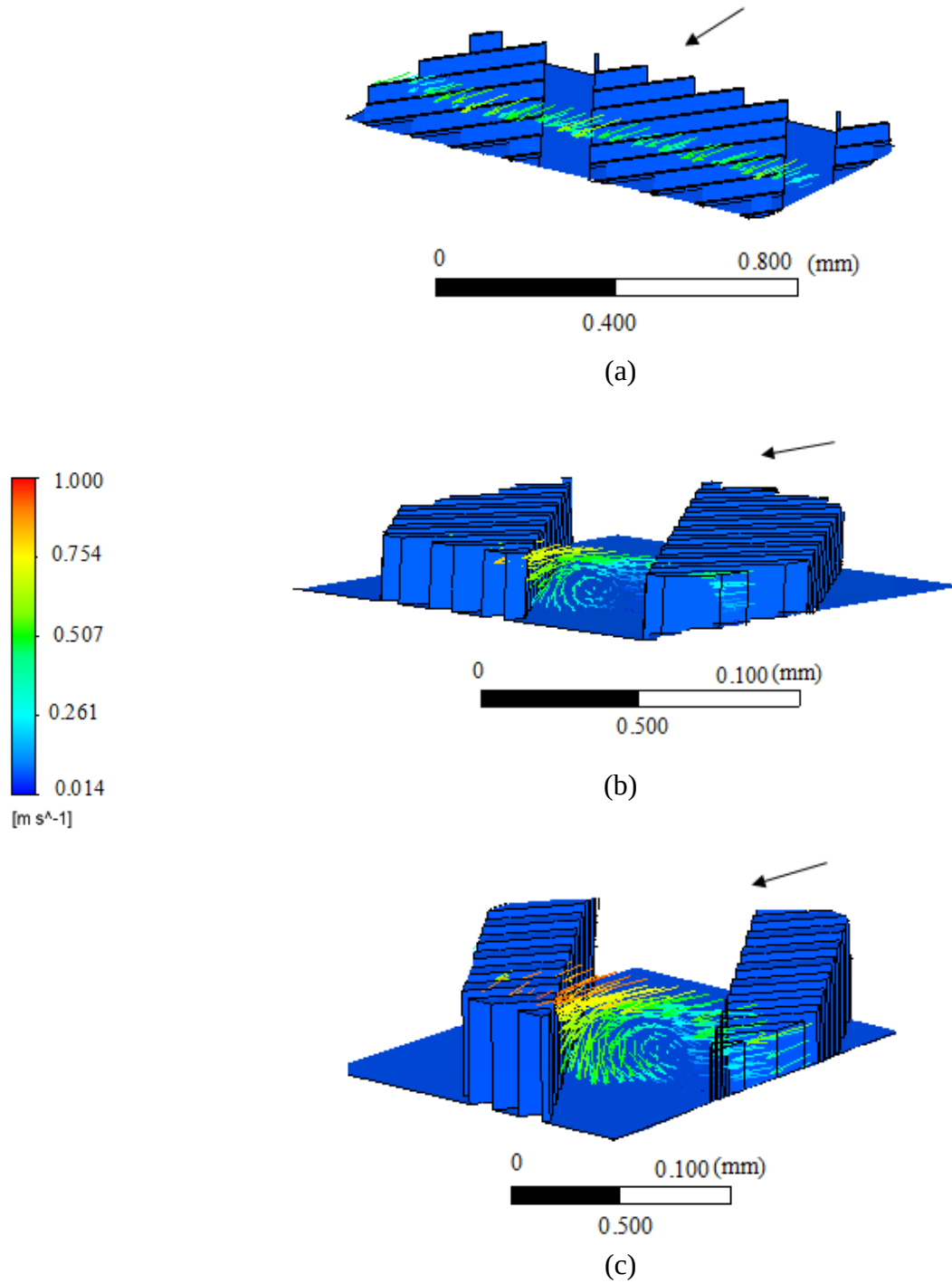


Figure 8.13. Velocity vectors in the micro-fin valleys of design points (a) A, (b) B, and (c) C

Figure 8.14 illustrates temperature contours at the solid-fluid interface for three design points of A, B, and C. Accordingly, design points A and C have maximum and minimum average

surface temperature of $38.6\text{ }^{\circ}\text{C}$ and $35.9\text{ }^{\circ}\text{C}$ compared to other optimal design points on the Pareto Front. The micro fin tip temperature goes down to $33\text{ }^{\circ}\text{C}$ in design point C. Unlike having the minimum fin efficiency of 80% among optimal design points, design point C has the maximum heat transfer enhancement.

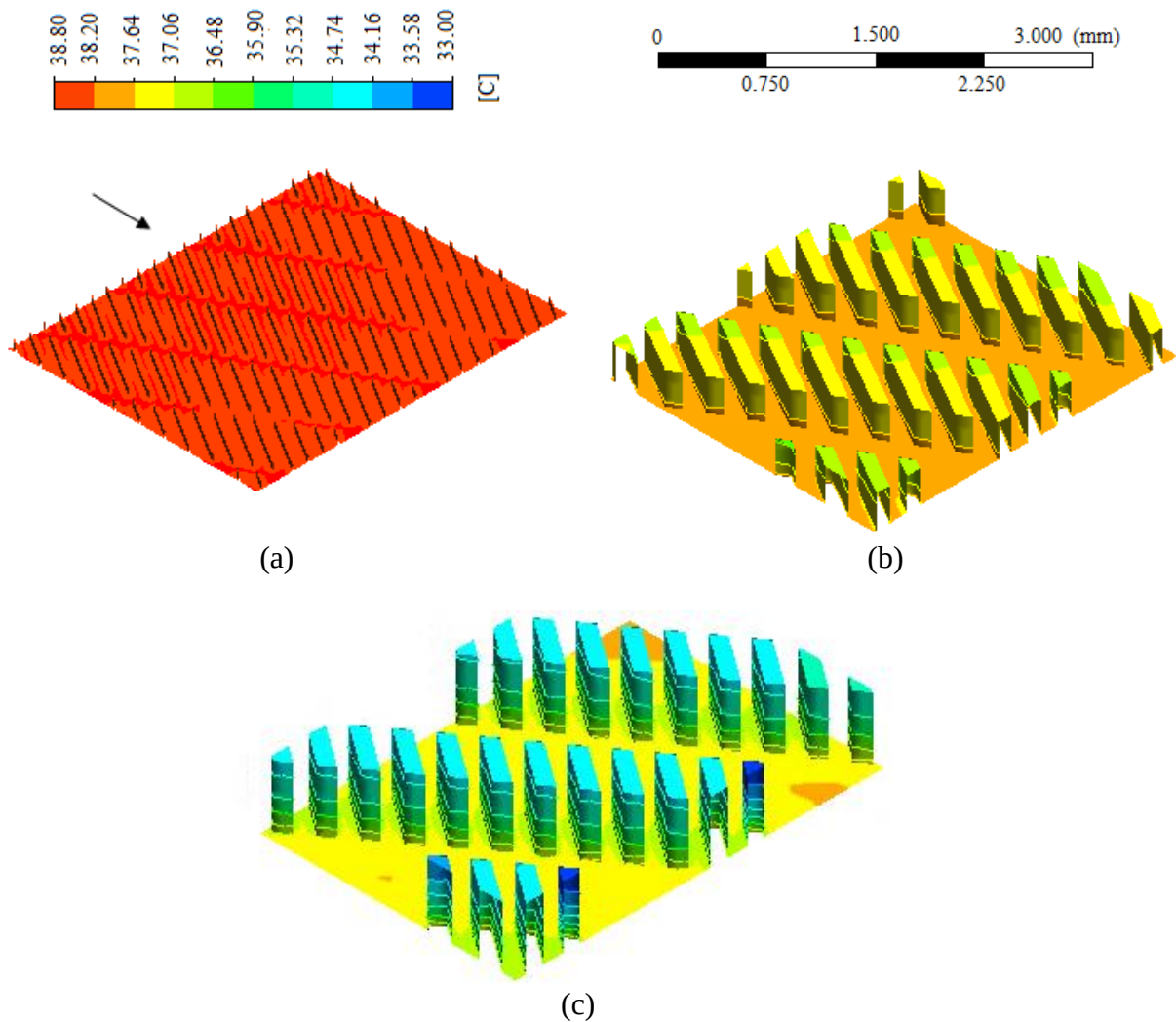


Figure 8.14. Temperature contours at a partial solid-fluid interface of design point (a) A (b) B, and (c) C

Figure 8.15 shows temperature contours above the micro-fin surface of design points A, B, and C at the cross-midsection of the channels. Design points A and C have the greatest and lowest

temperature gradient above the micro-fin surface. The value of micro-fin height determines the thickness of hot thermal layers above the micro-fins. The highest heat transfer location is downstream of the micro-fins along with the flow direction.

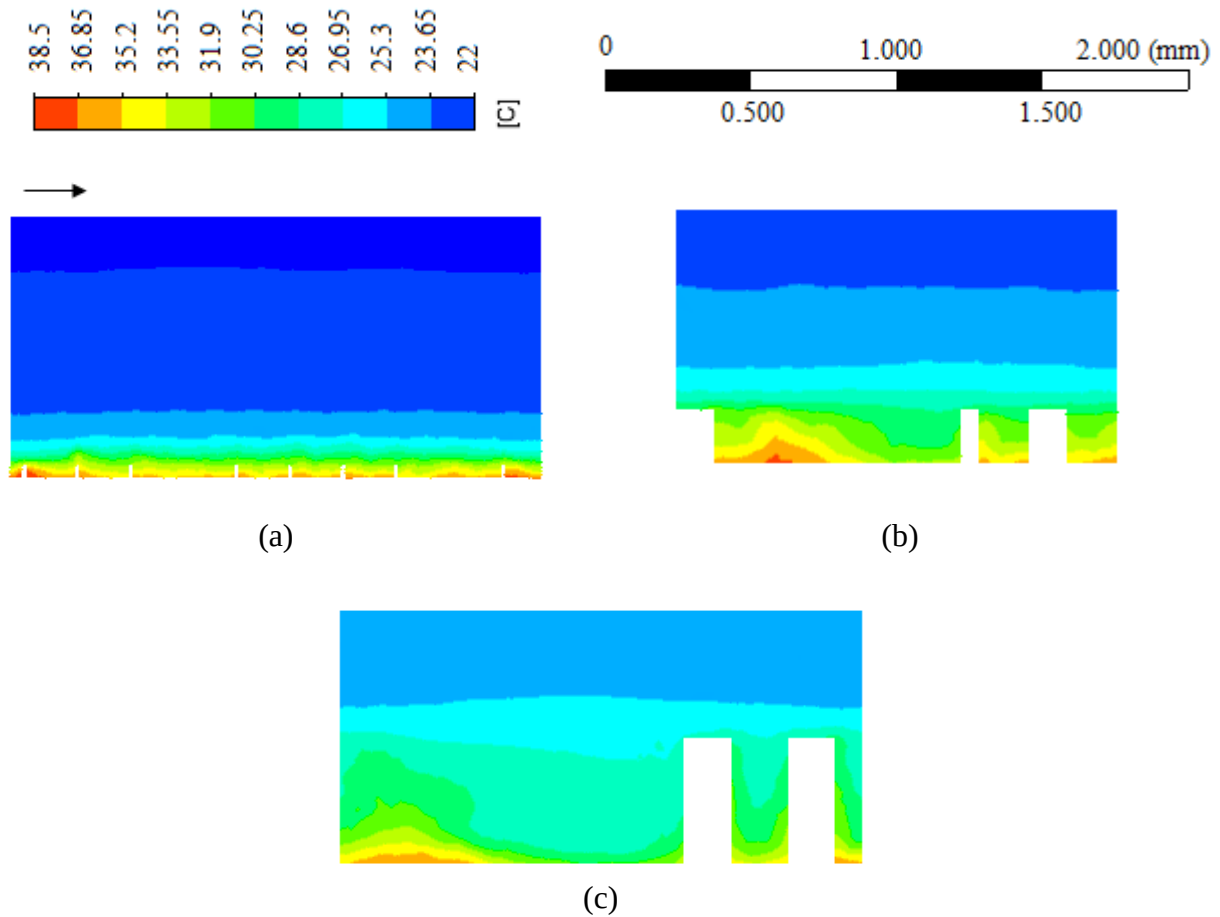


Figure 8.15. Temperature contours above the micro-fin surface at the cross-midsection of the channels with design points (a) *A* (b) *B*, and (c) *C*

9 Summary and Conclusion

This dissertation presented parametric studies and multi-objective shape optimization of micro-fins in turbulent flows and a review study on reducing drag and changing heat transfer by riblet surfaces. The objective functions of this study are maximizing heat transfer enhancement and minimizing friction factor increase. NSGAI was incorporated with CFD-based simulation to find the best set of micro-fin design variables for the given objective functions to employ a formal optimization methodology. Various design variables for 2D and 3D micro-fins were defined to realize the surface's full potential. The design variables were defined as micro-fin height, angle, thickness, pitch, length of discontinuity, and the number of discontinuities per micro-fin pitch for the 3D micro-fin optimization study. Different novel computational domains were defined to reduce the computational time of the simulation studies, consisting of two channels and a tube with periodic domains. A conduction model evaluating temperature distribution within the micro-fins was considered for the optimization and convective heat transfer modeling. In the simulations, fluid flow was steady state with a fixed Reynolds number, and a constant temperature boundary condition was specified at the channel and tube walls. For the first time, a data-mining model was built as a function of the micro-fin's design variables based on results obtained from numerical simulations to estimate heat transfer and pressure drop characteristics in 2D and 3D micro-fin tubes. The performance of the LS-SVR model was evaluated using the statistical criteria of *RMSE*, *MAE*, and *AAE*. The conclusions can be summarized as follows:

- 1- A transverse rectangular micro-fin with a pitch-to-height ratio of 3.5 (corresponds to a so-called *d*-type roughness) can enhance the efficiency factor by as much as 40% compared with a smooth surface.

- 2- By changing the micro-fin pitch, it was found that the heat transfer rate decreases and pressure drop increases for the transverse rectangular micro-fins with a pitch-to-height ratio of greater than 5, which demonstrated the breakdown of the Reynolds analogy.
- 3- The variable of pitch-to-height ratio had a significant impact on thermo-hydraulic performance.
- 4- Micro-fin height, helix angle, and the number of starts were similar for the best 2D and 3D micro-fin tubes.
- 5- The best 2D and 3D tubes enhanced heat transfer 68% and 80% rather than a smooth, respectively.
- 6- The best 2D and 3D tubes enhanced the thermo-hydraulic efficiency index up to 18% and 16% relative to a smooth tube, respectively.
- 7- Results of the LS-SVR model showed that the percentage of AAE between simulated and estimated j and f are 2.05% and 2.93% for 3D micro-fin tubes, respectively.
- 8- Statistical criteria revealed that the LS-SVR model is a suitable model as a surrogate of the CFD models to estimate the Colburn j factor and Fanning friction factor. The capability of the LS-SVR method to reduce computational time leads to future studies of micro-fins using advanced optimization algorithms to find the optimum micro-fin design variables.
- 9- The best micro-fin configuration of this study showed 18% improvement for the thermo-hydraulic efficiency index compared to the result of Mann and Eckels (2019).
- 10- This study outlined a particular micro-fin geometry that reduces drag by 8% and enhances heat transfer by 21% compared to a smooth surface in turbulent water flow.

The recommendations for future studies can be listed as follows:

- 1- Extend the LS-SVR model for estimating heat transfer and pressure characteristics in two-phase heat exchangers.
- 2- Set up experiment for the selected optimal points and validate the results.
- 3- Couple LS-SVR model as a surrogate of the CFD-based model with NASGA-II algorithm for micro-fin studies.
- 4- Use DNS to study the effects of riblet design variables and flow characteristic on drag reduction and heat transfer.
- 5- Using advanced data analytics technique such as machine learning methods to visualize and monitor the most effective patterns of coherent turbulent structures affect drag reduction and heat transfer on riblet surfaces from DNS results.
- 6- Clarify the physical mechanism of flow and heat transfer over riblet surfaces based on DNS results.

References

- Abdulbari, H.A., Yunus, R.M., Abdurahman, N.H., and Charles, A., 2013. Going against the flow—A review of non-additive means of drag reduction. *Journal of Industrial and Engineering Chemistry*, 19(1), pp.27-36.
- Abu-Hamdeh, N.H., Bantan, R.A. and Alimoradi, A., 2020. Heat transfer optimization through new form of pin type of finned tube heat exchangers using the exergy and energy analysis. *International Journal of Refrigeration*.
- Al-Sarkhi, A., 2010. Drag reduction with polymers in gas-liquid/liquid-liquid flows in pipes: a literature review. *Journal of Natural Gas Science and Engineering*, 2(1), pp.41-48.
- Aupoix, B., Pailhas, G. and Houdeville, R., 2012. Towards a general strategy to model riblet effects. *AIAA journal*, 50(3), pp.708-716.
- Aus der Wiesche, S., 2003. Heat Transfer and Drag Reduction in Flows Over Riblet Mounted Surfaces. In *ASME 2003 Heat Transfer Summer Conference*, pp. 449-457.
- Ayoobi, A., Ramezanizadeh, M. and Alhuyi-Nazari, M., 2020. Optimization of temperature distribution and heat flux functions for cylindrical and conical micro-fins by applying genetic algorithm. *Journal of Thermal Analysis and Calorimetry*, pp.1-12.
- Bandyopadhyay, P.R., 1986. Mean flow in turbulent boundary layers disturbed to alter skin friction. *Journal of Fluids Engineering*, 108(2), pp.127-140.
- Bannier, A., Garnier, E. and Sagaut, P., 2016. Riblets induced drag reduction on a spatially developing turbulent boundary layer. In *Progress in Wall Turbulence 2*, Springer, Cham.
- Barbier, C., Jenner, E. and D'Urso, B., 2012, November. Drag reduction with super-hydrophobic riblets. In *ASME International Mechanical Engineering Congress and Exposition (Vol. 45240, pp. 199-205)*. American Society of Mechanical Engineers.
- Bechert, D.W. and Bartenwerfer, M., 1989. The viscous flow on surfaces with longitudinal ribs. *Journal of fluid mechanics*, 206, pp.105-129.
- Bechert, D.W., Bartenwerfer, M., Hoppe, G. and Reif, W.E., 1986. Drag reduction mechanisms derived from shark skin. *AIAA journal*, 2, pp.1044-1068.
- Bechert, D.W., Bruse, M. and Hage, W., 2000. Experiments with three-dimensional riblets as an idealized model of shark skin. *Experiments in fluids*, 28(5), pp.403-412.
- Bechert, D.W., Bruse, M., Hage, W.V., Van der Hoeven, J.T. and Hoppe, G., 1997. Experiments on drag-reducing surfaces and their optimization with an adjustable geometry. *Journal of fluid mechanics*, 338, pp.59-87.
- Beigzadeh, R., Rahimi, M., Jafari, O. and Alsairafi, A.A., 2016. Computational fluid dynamics assists the artificial neural network and genetic algorithm approaches for thermal and flow modeling of air-forced convection on interrupted plate fins. *Numerical Heat Transfer, Part A: Applications*, 70(5), pp.546-565.
- Benhalilou, M. and Kasagi, N., 1999. Numerical prediction of heat and momentum transfer over micro-grooved surface with a nonlinear $k-\epsilon$ model. *International journal of heat and mass transfer*, 42(14), pp.2525-2541.

- Benhalilou, M., Anselmet, F., Liandrat, J. and Fulachier, L., 1991. Experimental and numerical investigation of a turbulent boundary layer over riblets. 8th Symposium on Turbulent Shear Flows, Technical University of Munich, Germany, 9-11 September.
- Benhamza, M.E. and Belaid, F., 2009. Computation of turbulent channel flow with variable spacing riblets. *Mechanics*, 79(5), pp.36-41.
- Benschop, H.O.G. and Breugem, W.P., 2017. Drag reduction by herringbone riblet texture in direct numerical simulations of turbulent channel flow. *Journal of Turbulence*, 18(8), pp.717-759.
- Bhatia, R.S. and Webb, R.L., 2001. Numerical study of turbulent flow and heat transfer in micro-fin tubes—part 1, model validation. *Journal of Enhanced Heat Transfer*, 8(5).
- Bilinsky, H.C. and Wilson, R.N., 2018. Maturation of Direct Contactless Microfabrication for Application of Drag-reducing Riblets. In 2018 AIAA Aerospace Sciences Meeting, Kissimmee, Florida, 8–12 January.
- Bilinsky, H.C., 2017. Riblet Microfabrication Method for Drag Reduction. In 55th AIAA Aerospace Sciences Meeting, Grapevine, Texas, 9-13 January.
- Bixler, G.D. and Bhushan, B., 2013a. Shark skin inspired low-drag microstructured surfaces in closed channel flow. *Journal of colloid and interface science*, 393, pp.384-396.
- Bixler, G.D. and Bhushan, B., 2013b. Fluid drag reduction with shark-skin riblet inspired microstructured surfaces. *Advanced Functional Materials*, 23(36), pp.4507-4528.
- Bozorg-Haddad, O., Soleimani, S. and Loáiciga, H.A., 2017. Modeling water-quality parameters using genetic algorithm–least squares support vector regression and genetic programming. *Journal of Environmental Engineering*, 143(7), p.04017021.
- Brennan, A. B., Baney, R. H., Carman, M. L., Estes, T. G., Feinberg, A. W., Wilson, L. H., & Schumacher, J. F. 2010. U.S. Patent No. 7,650,848. Washington, DC: U.S. Patent and Trademark Office.
- Brostow, W., 2008. Drag reduction in flow: Review of applications, mechanism and prediction. *Journal of industrial and Engineering Chemistry*, 14(4), pp.409-416.
- Byun, D., Kim, J., Ko, H.S. and Park, H.C., 2008. Direct measurement of slip flows in super-hydrophobic microchannels with transverse grooves. *Physics of Fluids*, 20(11), p.113601.
- Campet, R., Roy, P.T., Cuenot, B., Riber, É. and Jouhaud, J.C., 2020. Design optimization of an heat exchanger using Gaussian process. *International Journal of Heat and Mass Transfer*, 150, p.119264.
- Cavazzuti, M. and Corticelli, M.A., 2008. Optimization of heat exchanger enhanced surfaces through multiobjective genetic algorithms. *Numerical Heat Transfer, Part A: Applications*, 54(6), pp.603-624.
- Chen, H., Zhang, X., Ma, L., Che, D., Zhang, D. and Sudarshan, T.S., 2014. Investigation on large-area fabrication of vivid shark skin with superior surface functions. *Applied Surface Science*, 316, pp.124-131.
- Choi KS., Hamid S., 1991. Heat transfer study of riblets. *Fluid Mechanics and Its Applications*, 6. Springer, Dordrecht.

- Choi, H., Moin, P. and Kim, J., 1993. Direct numerical simulation of turbulent flow over riblets. *Journal of fluid mechanics*, 255, pp.503-539.
- Choi, K.S. and Orchard, D.M., 1997. Turbulence management using riblets for heat and momentum transfer. *Experimental Thermal and Fluid Science*, 15(2), pp.109-124.
- Choi, K.S., 1989. Near-wall structure of a turbulent boundary layer with riblets. *Journal of fluid mechanics*, 208, pp.417-458.
- Choi, K.S., 1993. Breakdown of the Reynolds analogy over drag-reducing riblets surface. In *Advances in Turbulence IV*, Springer, Dordrecht.
- Chu, D.C. and Karniadakis, G.E., 1993. A direct numerical simulation of laminar and turbulent flow over riblet-mounted surfaces. *Journal of Fluid Mechanics*, 250, pp.1-42.
- Ciri, U. and Leonardi, S., 2019. Heat transfer in a turbulent channel flow with super-hydrophobic or liquid-infused surfaces on one wall. 11th International Symposium on Turbulence and Shear Flow Phenomena (TSFP11) Southampton, UK, July 30 to August 2.
- Coles, D., 1956. The law of the wake in the turbulent boundary layer. *Journal of Fluid Mechanics*, 1(2), pp.191-226.
- Copiello, D. and Fabbri, G., 2009. Multi-objective genetic optimization of the heat transfer from longitudinal wavy fins. *International Journal of Heat and Mass Transfer*, 52(5-6), pp.1167-1176.
- Daguenet-Frick, X., Foucaut, J.M., Coudert, S., Toutant, A. and Olalde, G., 2012. Experimental analysis of the turbulent flow behavior of a textured surface proposed for asymmetric heat exchangers. *Flow, turbulence and combustion*, 89(1), pp.149-169.
- Dastmalchi, M., Sheikhzadeh, G.A. and Arefmanesh, A., 2017. Optimization of micro-finned tubes in double pipe heat exchangers using particle swarm algorithm. *Applied Thermal Engineering*, 119, pp.1-9.
- Dean, B. and Bhushan, B., 2010. Shark-skin surfaces for fluid-drag reduction in turbulent flow: a review. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 368(1929), pp.4775-4806.
- Dean, B. and Bhushan, B., 2012. The effect of riblets in rectangular duct flow. *Applied Surface Science*, 258(8), pp.3936-3947.
- Dean, B.D., 2011. The effect of shark skin inspired riblet geometries on drag in rectangular duct flow. Ph.D. Dissertation, The Ohio State University, United States.
- Deb, K., Pratap, A., Agarwal, S. and Meyarivan, T.A.M.T., 2002. A fast and elitist multiobjective genetic algorithm: NSGAII. *IEEE transactions on evolutionary computation*, 6(2), pp.182-197.
- Denkena, B., Köhler, J. and Wang, B., 2010. Manufacturing of functional riblet structures by profile grinding. *CIRP Journal of Manufacturing Science and Technology*, 3(1), pp.14-26.
- Dewan, A., Mahanta, P., Raju, K.S. and Kumar, P.S., 2004. Review of passive heat transfer augmentation techniques. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 218(7), pp.509-527.
- Drain, L.E. and Martin, S. 1985, Two-component velocity measurements of turbulent flow in a ribbed-wall flow channel, In *International conference on laser anemometry—advanced and application*, pp. 99-112.

- El-Samni, O.A., Chun, H.H. and Yoon, H.S., 2007. Drag reduction of turbulent flow over thin rectangular riblets. *International Journal of Engineering Science*, 45(2-8), pp.436-454.
- Enyutin, G.V., Lashkov, Y.A. and Samoilova, N.V., 1995. Drag reduction in riblet-lined pipes, *Fluid dynamics*, 30(1), pp.45-48.
- Favre-Marinet, M. and Tardu, S., 2013. *Convective heat transfer*. John Wiley & Sons.
- Fidanoglu, M., Komurgoz, G. and Ozkol, I., 2016. Heat transfer analysis of fins with spine geometry using differential transform method. *International Journal of Mechanical Engineering and Robotics Research*, 5(1), pp.67-71.
- Foli, K., Okabe, T., Olhofer, M., Jin, Y. and Sendhoff, B., 2006. Optimization of micro heat exchanger: CFD, analytical approach and multi-objective evolutionary algorithms. *International Journal of Heat and Mass Transfer*, 49(5-6), pp.1090-1099.
- García-Mayoral, R. and Jiménez, J., 2011. Drag reduction by riblets. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 369(1940), pp.1412-1427.
- Ghanadi Arab, M.M., Hajabdollahi, M. and Hajabdollahi, H., 2013. Multi-objective optimization of a fin with twodimensional heat transfer using NSGAI and ANN. *Journal of Applied Mechanical Engineering*, 2(2), pp.117-124.
- Ghanadi, F., Arjomandi, M., Zander, A.C. and Cazzolato, B.S., 2012. A review of skin friction drag reduction within the turbulent boundary layer. In *Proceedings: the 7th Australasian Congress on Applied Mechanics (ACAM 7)*, Adelaide, Australia, 9-12 December.
- Goldstein, D., Handler, R. and Sirovich, L., 1995. Direct numerical simulation of turbulent flow over a modeled riblet covered surface. *Journal of Fluid Mechanics*, 302, pp.333-376.
- Golovin, K.B., Gose, J.W., Perlin, M., Ceccio, S.L. and Tuteja, A., 2016. Bioinspired surfaces for turbulent drag reduction. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 374(2073), p.20160189.
- Grüneberger, R. and Hage, W., 2011. Drag characteristics of longitudinal and transverse riblets at low dimensionless spacings. *Experiments in Fluids*, 50(2), pp.363-373.
- Güngör, S., Ceyhan, U. and Karadeniz, Z.H., 2020. Optimization of heat transfer in a grooved pipe model by Stochastic Algorithms and DOE based RSM. *International Journal of Thermal Sciences*, 159, p.106634.
- Hamadneh, N.N., Khan, W.A. and Khan, I., 2020. Second Law Analysis and Optimization of Elliptical Pin Fin Heat Sinks Using Firefly Algorithm. *CMC-COMPUTERS MATERIALS & CONTINUA*, 65(2), pp.1015-1032.
- Han, X., Zhang, D., Li, X. and Li, Y., 2008. Bio-replicated forming of the biomimetic drag-reducing surfaces in large area based on shark skin. *Chinese Science Bulletin*, 53(10), p.1587.
- Hon, T.L. and Walker, J.D., 1987. An Analysis of the Motion and Effects of Hairpin Vortices (No. FM-11). LEHIGH UNIV BETHLEHEM PA DEPT OF MECHANICAL ENGINEERING AND MECHANICS.
- Hou, J., Hokmabad, B.V. and Ghaemi, S., 2017. Three-dimensional measurement of turbulent flow over a riblet surface. *Experimental Thermal and Fluid Science*, 85, pp.229-239.

- Jamshed, S., Qureshi, S.R. and Shah, A., 2019. Numerical Flow Analysis and Heat Transfer in Spirally Grooved Tubes in Heat Exchangers. *Iranian Journal of Science and Technology, Transactions of Mechanical Engineering*, 43(3), pp.477-486.
- Jasiński, P.I.O.T.R., 2011, August. Numerical optimization of flow-heat ducts with helical micro-fins, using Entropy Generation Minimization (EGM) method. In *Proceedings of WSEAS International Conferences (HTE'11)* (pp. 47-54).
- Ji, W.T., Jacobi, A.M., He, Y.L. and Tao, W.Q., 2015. Summary and evaluation on single-phase heat transfer enhancement techniques of liquid laminar and turbulent pipe flow. *International Journal of Heat and Mass Transfer*, 88, pp.735-754.
- Jiménez, J., 2004. Turbulent flows over rough walls. *Annu. Rev. Fluid Mech.*, 36, pp.173-196.
- Jin, Y. and Herwig, H., 2014. Turbulent flow and heat transfer in channels with shark skin surfaces: Entropy generation and its physical significance. *International Journal of Heat and Mass Transfer*, 70, pp.10-22.
- Juan, D. and Qin, Q.Z., 2014. Multi-objective optimization of a plain fin-and-tube heat exchanger using genetic algorithm. *Thermal Engineering*, 61(4), pp.309-317.
- Juan, D., Man-Ni, Y., and Shi-Fang Y., 2016. Correlations and optimization of a heat exchanger with offset fins by genetic algorithm combining orthogonal design. *Applied Thermal Engineering*, 107, pp.1091-1103.
- Jung, Y.C. and Bhushan, B., 2009. Biomimetic structures for fluid drag reduction in laminar and turbulent flows. *Journal of Physics: Condensed Matter*, 22(3), p.035104.
- Kahani, M., Ahmadi, M.H., Tatar, A. and Sadeghzadeh, M., 2018. Development of multilayer perceptron artificial neural network (MLP-ANN) and least square support vector machine (LSSVM) models to predict Nusselt number and pressure drop of TiO₂/water nanofluid flows through non-straight pathways. *Numerical Heat Transfer, Part A: Applications*, 74(4), pp.1190-1206.
- Kays, W.M. and Crawford, M.E., 1993, *Convective Heat and Mass Transfer*, McGraw-Hill. New York.
- Kazi, S.N., Togun, H. and Sadeghinezhad, E., 2012. Heat Transfer to Separation Flow in Heat Exchangers. In *An Overview of Heat Transfer Phenomena*. IntechOpen.
- Kim, H.M. and Kim, K.Y., 2004. Design optimization of rib-roughened channel to enhance turbulent heat transfer. *International Journal of Heat and Mass Transfer*, 47(23), pp.5159-5168.
- Kim, J.H., Jansen, K.E. and Jensen, M.K., 2004. Simulation of three-dimensional incompressible turbulent flow inside tubes with helical fins. *Numerical Heat Transfer, Part B*, 46(3), pp.195-221.
- Kim, T.W., 2014. Assessment of hydro/oleophobicity for shark skin replica with riblets. *Journal of nanoscience and nanotechnology*, 14(10), pp.7562-7568.
- Kline, S.J., Reynolds, W.C., Schraub, F.A. and Runstadler, P.W., 1967. The structure of turbulent boundary layers. *Journal of Fluid Mechanics*, 30(4), pp.741-773.
- Kobus, C.J. and Cavanaugh, R.B., 2006. A theoretical investigation into the optimal longitudinal profile of a horizontal pin fin of least material under the influence of pure forced and pure natural convection with a diameter-variable convective heat transfer coefficient.

- Kolar, V., 2007. Vortex identification: New requirements and limitations. *International journal of heat and fluid flow*, 28(4), pp.638-652.
- Koo, G.W., Lee, S.M. and Kim, K.Y., 2014. Shape optimization of inlet part of a printed circuit heat exchanger using surrogate modeling. *Applied thermal engineering*, 72(1), pp.90-96.
- Kordy, H., 2015. Process abilities of the riblet-coating process with dual-cure lacquers. *CIRP Journal of Manufacturing Science and Technology*, 11, pp.1-9.
- Kundu, B. and Das, P.K., 2002. Performance analysis and optimization of straight taper fins with variable heat transfer coefficient. *International journal of heat and mass transfer*, 45(24), pp.4739-4751.
- Kundu, B., and Das, P. K. 2005. Optimum profile of Thin Fins with volumetric heat generation: A unified approach. *ASME. J. Heat Transfer*, 127(8): 945–948.
- Launder, B. and Spalding, D., 1974. The numerical computation of turbulent flows, *Computer Methods in Applied Mechanics and Energy*, 3, 269–289.
- Launder, B.E. and Li, S.P., 1993. On the prediction of riblet performance with engineering turbulence models. *Applied scientific research*, 50(3-4), pp.283-298.
- Lee, C. and Choi, C.H., 2008. Structured surfaces for a giant liquid slip. *Physical review letters*, 101(6), p.064501.
- Lee, S.J. and Lee, S.H., 2001. Flow field analysis of a turbulent boundary layer over a riblet surface. *Experiments in fluids*, 30(2), pp.153-166.
- Leonardi, S., Orlandi, P., Djenidi, L. and Antonia, R.A., 2015. Heat transfer in a turbulent channel flow with square bars or circular rods on one wall. *Journal of Fluid Mechanics*, 776, p.512.
- Li, P. and Eckels, S.J. 2017. Measurements and Numerical Simulations of Heat-transfer and Pressure Drop in a Duct with Smooth Walls. In *ASME 2017 International Mechanical Engineering Congress and Exposition American Society of Mechanical Engineers*, November, pp. V008T10A004-V008T10A004.
- Li, P. and Kim, K.Y., 2008. Multi-objective optimization of staggered elliptical pin-fin arrays. *Numerical Heat Transfer, Part A: Applications*, 53(4), pp.418-431.
- Li, P., Campbell, M., Zhang, N. and Eckels, S.J., 2019. Relationship between turbulent structures and heat transfer in microfin enhanced surfaces using large eddy simulations and particle image velocimetry. *International Journal of Heat and Mass Transfer*, 136, pp.1282-1298.
- Li, P., Eckels, S.J., Mann, G.W. and Zhang, N. 2018. A Method of Measuring Turbulent Flow Structures with Particle Image Velocimetry and Incorporating Into Boundary Conditions of Large Eddy Simulations. *Journal of Fluids Engineering*, 140(7), p.071401.
- Li, W., Fu, P., Li, H., Li, G. and Thors, P., 2016. Numerical–theoretical analysis of heat transfer, pressure drop, and fouling in internal helically ribbed tubes of different geometries. *Heat Transfer Engineering*, 37(3-4), pp.279-289.
- Lindemann, A.M., 1985. Turbulent Reynolds analogy factors for nonplanar surface microgeometries. *Journal of Spacecraft and Rockets*, 22(5), pp.581-582.
- Lindqvist, K., Wilson, Z.T., Næss, E. and Sahinidis, N.V., 2018. A machine learning approach to correlation development applied to fin-tube bundle heat exchangers. *Energies*, 11(12), p.3450.

- Ling, H., Srinivasan, S., Golovin, K., McKinley, G.H., Tuteja, A. and Katz, J., 2016. High-resolution velocity measurement in the inner part of turbulent boundary layers over super-hydrophobic surfaces. *Journal of Fluid Mechanics*, 801, pp.670-703.
- Liu, H., Zhang, H., Yang, F., Hou, X., Yu, F. and Song, S., 2017. Multi-objective optimization of fin-and-tube evaporator for a diesel engine-organic Rankine cycle (ORC) combined system using particle swarm optimization algorithm. *Energy Conversion and Management*, 151, pp.147-157.
- Liu, K. N., Joseph, D. D., Riccius, O., Christodoulou, C. 1990. Drag reduction in pipes lined with riblets. *AIAA journal*, 28(10), pp.1697-1698.
- Liu, X. and Jensen, M.K., 2001. Geometry effects on turbulent flow and heat transfer in internally finned tubes. *J. Heat Transfer*, 123(6), pp.1035-1044.
- Liu, Z., Yue, Y., She, L. and Fan, G., 2019. Numerical analysis of turbulent flow and heat transfer in internally finned tubes. *Frontiers in Energy Research*, 7, p.64.
- Luo, L., Du, W., Wang, S., Wu, W. and Zhang, X., 2019. Multi-objective optimization of the dimple/protrusion channel with pin fins for heat transfer enhancement. *International Journal of Numerical Methods for Heat & Fluid Flow*.
- Luo, Y., 2015. Recent progress in exploring drag reduction mechanism of real sharkskin surface: A review. *Journal of Mechanics in Medicine and Biology*, 15(03), p.1530002.
- Luo, Y., Wang, L., Green, L., Song, K., Wang, L. and Smith, R., 2015. Advances of drag-reducing surface technologies in turbulence based on boundary layer control. *Journal of Hydrodynamics*, 27(4), pp.473-487.
- Madou, M.J., 2011. *Manufacturing techniques for microfabrication and nanotechnology*. CRC press, New York.
- Mann, G.W. and Eckels, S., 2019. Multi-objective heat transfer optimization of 2D helical micro-fins using NSGAI. *International Journal of Heat and Mass Transfer*, 132, pp.1250-1261.
- Marentic, F. J., & Morris, T. L. (1992). U.S. Patent No. 5,133,516. Washington, DC: U.S. Patent and Trademark Office.
- Martin, S. and Bhushan, B., 2014. Fluid flow analysis of a shark-inspired microstructure. *Journal of Fluid Mechanics*, 756, pp.5-29.
- Martin, S. and Bhushan, B., 2016a. Fluid flow analysis of continuous and segmented riblet structures. *Rsc Advances*, 6(13), pp.10962-10978.
- Martin, S. and Bhushan, B., 2016b. Modeling and optimization of shark-inspired riblet geometries for low drag applications. *Journal of colloid and interface science*, 474, pp.206-215.
- Maynes, D., Webb, B.W. and Davies, J., 2008. Thermal transport in a microchannel exhibiting ultra-hydrophobic microribs maintained at constant temperature. *Journal of Heat Transfer*, 130(2).
- Meng, Z., Shen, R. and Jiang, M., 2011. An objective penalty functions algorithm for multiobjective optimization problem. *American Journal of Operations Research*, 1(04), p.229.
- Mikk, I., 1980. Convective fin of minimum mass. *International Journal of Heat and Mass Transfer*, 23(5), pp.707-711.

- Mirzaei, M., Hajabdollahi, H. and Fadakar, H., 2017. Multi-objective optimization of shell-and-tube heat exchanger by constructal theory. *Applied Thermal Engineering*, 125, pp.9-19.
- Neumann, D. and Dinkelacker, A., 1991. Drag measurements on V-grooved surfaces on a body of revolution in axial flow. *Applied Scientific Research*, 48(1), pp.105-114.
- Ng, J.H., Jaiman, R.K. and Lim, T.T., 2016. Direct numerical simulations of riblets in a fully-developed turbulent channel flow: Effects of geometry. In *Advances in Computation, Modeling and Control of Transitional and Turbulent Flows* (pp. 155-165).
- Nguyen, Q.H., Ly, H.B., Ho, L.S., Al-Ansari, N., Le, H.V., Tran, V.Q., Prakash, I. and Pham, B.T., 2021. Influence of data splitting on performance of machine learning models in prediction of shear strength of soil. *Mathematical Problems in Engineering*, 2021.
- Nikuradse, J., 1933. Laws of Flow in Rough Pipes. VDI Forschungsheft, pp. 361. [English translation, NACA TM-1292 (1965).]
- Nobile, E., Pinto, F. and Rizzetto, G., 2006. Geometric parameterization and multiobjective shape optimization of convective periodic channels. *Numerical Heat Transfer, Part B: Fundamentals*, 50(5), pp.425-453.
- Okabayashi, K., Hirai, K., Takeuchi, S. and Kajishima, T., 2018. Direct numerical simulation of turbulent flow above zigzag riblets. *AIP Advances*, 8(10), p.105227.
- Okamoto, S., Seo, S., Nakaso, K. and Kawai, I., 1993. Turbulent shear flow and heat transfer over the repeated two-dimensional square ribs on ground plane.
- Orlandi, P., Sassun, D. and Leonardi, S., 2016. DNS of conjugate heat transfer in presence of rough surfaces. *International Journal of Heat and Mass Transfer*, 100, pp.250-266.
- Ozkol, I. and Komurgoz, G., 2005. Determination of the optimum geometry of the heat exchanger body via a genetic algorithm. *Numerical Heat Transfer, Part A: Applications*, 48(3), pp.283-296.
- Park, K. and Moon, S., 2005. Optimal design of heat exchangers using the progressive quadratic response surface model. *International Journal of Heat and Mass Transfer*, 48(11), pp.2126-2139.
- Patankar, S.V., Liu, C.H. and Sparrow, E.M., (1977), "Fully developed flow and heat-transfer in ducts having streamwise-periodic variations of cross-sectional area", *Journal of Heat-transfer*, 99(2), pp.180-186.
- Peabody, H. L. 2001. U.S. Patent No. 6,278,051. Washington, DC: U.S. Patent and Trademark Office.
- Peeters, J.W.R. and Sandham, N.D., 2019. Turbulent heat transfer in channels with irregular roughness. *International Journal of Heat and Mass Transfer*, 138, pp.454-467.
- Peng, H. and Ling, X., 2015. Predicting thermal-hydraulic performances in compact heat exchangers by support vector regression. *International Journal of Heat and Mass Transfer*, 84, pp.203-213.
- Perlin, M., Dowling, D.R. and Ceccio, S.L., 2016. Freeman scholar review: passive and active skin-friction drag reduction in turbulent boundary layers. *Journal of Fluids Engineering*, 138(9), p.091104.

- Pozarlik, A.K., Panara, D., Kok, J.B. and van der Meer, T.H., 2008, May. Heat Transfer in a Recirculation Zone at Steady-State and Oscillating Conditions—The Back Facing Step Test Case. In 5th European Thermal-Sciences Conference, EUROTHERM2008.
- Ranut, P., Janiga, G., Nobile, E. and Thévenin, D., 2014. Multi-objective shape optimization of a tube bundle in cross-flow. *International Journal of Heat and Mass Transfer*, 68, pp.585-598.
- Rao, R.V. and Patel, V., 2011. Design optimization of shell and tube heat exchangers using swarm optimization algorithms. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 225(5), pp.619-634.
- Rao, R.V. and Saroj, A., 2017. Economic optimization of shell-and-tube heat exchanger using Jaya algorithm with maintenance consideration. *Applied Thermal Engineering*, 116, pp.473-487.
- Razelos, P. and Imre, K., 1983. Minimum mass convective fins with variable heat transfer coefficients. *Journal of the Franklin Institute*, 315(4), pp.269-282.
- Reidy, L. and Anderson, G., 1988. Drag reduction for external and internal boundary layers using riblets and polymers. In 26th Aerospace Sciences Meeting, Reno, Nevada, 11-14 January.
- Robinson, S.K., 1991. Coherent motions in the turbulent boundary layer. *Annual Review of Fluid Mechanics*, 23(1), pp.601-639.
- Rohr, J.J., Andersen, G.W., Reidy, L.W. and Hendricks, E.W., 1992. A comparison of the drag-reducing benefits of riblets in internal and external flows. *Experiments in fluids*, 13(6), pp.361-368.
- Romans, T. and Hirt, G., 2010. Rolling of Drag-reducing Riblet-Surfaces. In 51st AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference 18th AIAA/ASME/AHS Adaptive Structures Conference, Orlando, Florida, 12- 15 April.
- Rout, S. K., Hussein, A. K., Mohanty, C. P. 2015. Multi-objective optimization of a three-dimensional internally finned tube based on Response Surface Methodology (RSM). *Journal of Thermal Engineering*, 1(2), pp.131-142.
- Sanaye, S. and Hajabdollahi, H., 2010. Thermal-economic multi-objective optimization of plate fin heat exchanger using genetic algorithm. *Applied Energy*, 87(6), pp.1893-1902.
- Saravi, S.S. and Cheng, K., 2013. A review of drag reduction by riblets and micro-textures in the turbulent boundary layers. *European Scientific Journal*, 9(33).
- Sareen, A., Deters, R.W., Henry, S.P. and Selig, M.S., 2014. Drag reduction using riblet film applied to airfoils for wind turbines, *Journal of Solar Energy Engineering*, 136(2), p.021007.
- Schlichting, H. and Gersten, K. 2016. *Boundary-layer theory*. 9th edition, Springer.
- Shi, H.N., Ma, T., Chu, W.X. and Wang, Q.W., 2017. Optimization of inlet part of a microchannel ceramic heat exchanger using surrogate model coupled with genetic algorithm. *Energy Conversion and Management*, 149, pp.988-996.
- Soleimani, S., Bozorg-Haddad, O., Saadatpour, M. and Loáiciga, H.A., 2016. Optimal selective withdrawal rules using a coupled data mining model and genetic algorithm. *Journal of Water Resources Planning and Management*, 142(12), p.04016064.
- Stalio, E. and Nobile, E., 2003. Direct numerical simulation of heat transfer over riblets. *International Journal of Heat and Fluid Flow*, 24(3), pp.356-371.

- Stenzel, V., Wilke, Y. and Hage, W., 2011. Drag-reducing paints for the reduction of fuel consumption in aviation and shipping. *Progress in Organic Coatings*, 70(4), pp.224-229.
- Sun, J. and Bhushan, B., 2019. Nanomanufacturing of bioinspired surfaces. *Tribology International*, 129, pp.67-74.
- Suykens, J. A. K., Van Gestel, T., De Brabanter, J., De Moor, B., and Vandewalle, J. 2002. Least squares support vector machines, World Scientific, Singapore.
- Takahashi, K., Nakayama, W. and Kuwahara, H., 1988. Enhancement of forced convective heat transfer in tubes having three-dimensional spiral ribs. *Heat transfer. Japanese research*, 17(4), pp.12-28.
- Tani, I., 1988. Drag reduction by riblet viewed as roughness problem. *Proceedings of the Japan Academy, Series B*, 64(2), pp.21-24.
- Tayal, M.C., Fu, Y. and Diwekar, U.M., 1999. Optimal design of heat exchangers: A genetic algorithm framework. *Industrial & engineering chemistry research*, 38(2), pp.456-467.
- Toms, B.A. (1949), in *Proceedings of the International Congress on Rheology*, North Holland, Amsterdam.
- Tullis, S. and Pollard, A., 1994. The time dependent flow over V-and U-groove riblets of different sizes. *Physics of Fluids*, 6(3), pp.1310-1314.
- Tummers, M.J. and Steunebrink, M., 2019. Effect of surface roughness on heat transfer in Rayleigh-Bénard convection. *International Journal of Heat and Mass Transfer*, 139, pp.1056-1064.
- Ullmann, A. and Kalman, H., 1989. Efficiency and optimized dimensions of annular fins of different cross-section shapes. *International journal of heat and mass transfer*, 32(6), pp.1105-1110.
- Vajdi Hokmabad, B. and Ghaemi, S., 2016. Turbulent flow over wetted and non-wetted super-hydrophobic counterparts with random structure. *Physics of Fluids*, 28(1), p.015112.
- Vapnik, V., Golowich, S. E. & Smola, A. 1996. Support vector method for function approximation, regression estimation, and signal processing. *Proceedings of the 9th International Conference on Neural Information Processing Systems Advances*, Denver, USA, 3–5 December.
- Vukoslavcevic, P., Wallace, J.M. and Balint, J.L., 1992. Viscous drag reduction using streamwise-aligned riblets. *AIAA journal*, 30(4), pp.1119-1122.
- Walsh, M. and Lindemann, A., 1984. Optimization and application of riblets for turbulent drag reduction. In *22nd Aerospace Sciences Meeting*, Reno, United States, 9-12 January.
- Walsh, M., 1982. Turbulent boundary layer drag reduction using riblets. In *20th aerospace sciences meeting*, Orlando, United States, 11-14 January.
- Walsh, M.J. and Weinstein, L.M., 1979. Drag and heat-transfer characteristics of small longitudinally ribbed surfaces. *AIAA journal*, 17(7), pp.770-771.
- Wang, J., Liu, Z., Yuan, F., Liu, W. and Chen, G., 2015. Convective heat transfer optimization in a circular tube based on local exergy destruction minimization. *International Journal of Heat and Mass Transfer*, 90, pp.49-57.

- Wang, J.J., Lan, S.L. and Chen, G., 2000. Experimental study on the turbulent boundary layer flow over riblets surface. *Fluid dynamics research*, 27(4), p.217.
- Wansaseub, K., Pholdee, N. and Bureerat, S., 2017. Optimal U-shaped baffle square-duct heat exchanger through surrogate-assisted self-adaptive differential evolution with neighbourhood search and weighted exploitation-exploration. *Applied Thermal Engineering*, 118, pp.455-463.
- Webb, R.L. and Kim, N.Y., 2005. *Enhanced heat transfer*. Taylor and Francis, NY.
- Webb, R.L., 2009. Single-phase heat transfer, friction, and fouling characteristics of three-dimensional cone roughness in tube flow. *International Journal of Heat and Mass Transfer*, 52(11-12), pp.2624-2631.
- Webb, R.L., Eckert, E.R.G. and Goldstein, R., 1971. Heat transfer and friction in tubes with repeated-rib roughness. *International Journal of Heat and Mass Transfer*, 14(4), pp.601-617.
- Webb, R.L., Narayanamurthy, R. and Thors, P., 1999. Heat transfer and friction characteristics of internal helical-rib roughness. *J. Heat Transfer*, 122(1), pp.134-142.
- West, N., Sammut, K. and Tang, Y., 2018. Material selection and manufacturing of riblets for drag reduction: An updated review. *Proceedings of the Institution of Mechanical Engineers, Part L: Journal of Materials: Design and Applications*, 232(7), pp.610-622.
- White, F., *Viscous Fluid Flow*, 3rd ed. McGraw Hill, New York.
- Xie, G.N., Sundén, B. and Wang, Q.W., 2008. Optimization of compact heat exchangers by a genetic algorithm. *Applied Thermal Engineering*, 28(8-9), pp.895-906.
- Zhao, D.Y., Huang, Z.P., Wang, M.J., Wang, T. and Jin, Y., 2012. Vacuum casting replication of micro-riblets on shark skin for drag-reducing applications. *Journal of Materials Processing Technology*, 212(1), pp.198-202.