NUMERICAL ANALYSIS OF AIRSIDE CHARACTERISTICS IN PLAIN AND
WAVY HEAT EXCHANGERS IN THE TURBULENT FLOW REGIME

by

Santhosh Kizhakkevedu Ramachandran

A thesis submitted in partial fulfillment
of the requirements for the degree

of

Master of Science

in

Mechanical Engineering

MONTANA STATE UNIVERSITY
Bozeman, Montana

November 2010

©COPYRIGHT

by

Santhosh Kizhakkevedu Ramachandran

2010

All Rights Reserved

APPROVAL

of a thesis submitted by

Santhosh Kizhakkeveedu Ramachandran

This thesis has been read by each member of the thesis committee and has been found to be satisfactory regarding content, English usage, format, citation, bibliographic style, and consistency and is ready for submission to the Division of Graduate Education.

Dr. Ruhul M. Amin

Approved for the Department of Mechanical and Industrial Engineering

Dr. Christopher H. Jenkins

Approved for the Division of Graduate Education

Dr. Carl A. Fox

STATEMENT OF PERMISSION TO USE

In presenting this thesis in partial fulfillment of the requirements for a master's degree at Montana State University, I agree that the Library shall make it available to borrowers under rules of the Library.

If I have indicated my intention to copyright this thesis by including a copyright notice page, copying is allowable only for scholarly purposes, consistent with "fair use" as prescribed in the U.S. Copyright Law. Requests for permission for extended quotation from or reproduction of this thesis in whole or in parts may be granted only by the copyright holder.

Santhosh Kizhakkevedu Ramachandran

November 2010

ACKNOWLEDGEMENTS

I would like to begin by expressing my deepest thank you to Dr. Ruhul Amin for being my mentor throughout the course of this research. It was Dr. Amin's knowledge and spirit that kept me motivated and determined throughout my course of study. I am greatly honored to be able to say that I was able to learn from a professor who expresses such love and devotion for the field of engineering.

I would like to acknowledge Dr. Alan George and Dr. Ahsan Mian for their excellent support as committee members. I thank you both for your time and input. Deep gratitude goes to Dr. Christopher Jenkins, Dr. Doug Cairns and Dr. Ladean Mckittrick for their help in making my graduate school life an educating one.

I would also like to thank Almighty, my friends, Ranjana, Umang, Jyoti and Ramana; Naren and Murali my great colleagues Barkan, Chantz, Cody, Danny, Donny, Eric, Issac, Jason, JJ, Kylan, Matt, Nate, TJ and Tony; my beloved family back in India; I would not be where I am today if it were not for their love and support. Also, I would like to thank my fiancée Aiswarya for being there for me during thick and thin at least over the phone.

I can only claim all of the error, for all of the success does not belong solely to me.

TABLE OF CONTENTS

1. INTRODUCTION	1
Literature Review	2
Motivation for Present Research	9
2. BACKGROUND	11
Heat Exchangers	11
Plate-fin Heat Exchanger	12
Louvered Fin:	12
Offset Strip Fin:	14
Perforated Fin:	15
Plain Fin:	16
Wavy Fin:	17
Computational Fluid Dynamics	18
3. NUMERICAL METHOD	25
Governing Equations	25
Turbulence	27
Turbulence Models	28
The Baseline (BSL) $k-\omega$ Model	29
Modeling Flow Near the Wall	35
Mathematical Design for Automatic Near Wall Treatment:	37
Scalable Wall-Functions:	38
Y Plus Definitions	39
Standard Wall Function Based	40
Scalable Wall Function Based	40
4. COMPUTATIONAL DOMAIN	43
Nomenclature	47
Boundary Conditions	50
Inlet Boundary Condition	51
Outlet Boundary Condition	52
Wall Boundary Conditions	52
Symmetry Boundary Conditions	53

TABLE OF CONTENTS-CONTINUED

Non-Dimensionalization of Governing Equations and Associated Boundary Conditions.....	56
Non-Dimensional Boundary Conditions:	58
Inlet Boundary Condition	58
Outlet Boundary Condition.....	59
Wall Boundary Condition	59
Symmetry Boundary Condition	59
5. GRID INDEPENDENCE	60
Mesh Generation	60
Mesh Optimization.....	61
Edge Spacing	62
Face Spacing.....	63
Fin Analysis	71
6. CODE VALIDATION.....	76
Plain Fin Staggered Configurataion	77
Wavy Fin Staggered Configuration.....	79
7. RESULTS AND DISCUSSION	81
Flow Distinction.....	83
Flow Distinction Between Laminar and Turbulent Flow	85
Flow Distinction Between Inline and Staggered Configuration.....	91
Number of Tube Rows Study.....	102
Plain Fin Staggered Configuration	102
Wavy Fin Staggered Configuration	106
Effects of Transverse Pitch	111
Wavy Fin Staggered Configuration	115
Effects of Fin Pitch.....	118
Effects of Wavy Angle.....	126
Effects of Wavy Height.....	131

TABLE OF CONTENTS-CONTINUED

8. CONCLUSIONS.....	136
REFERENCES.....	139

LIST OF TABLES

Table	Page
1. Geometrical parameters adopted for mesh optimization.	64
2. Different grid resolutions adopted for mesh optimization for the plain fin staggered configuration.....	65
3. Different grid resolutions adopted for mesh optimization for the wavy fin staggered configuration.....	68
4. Geometrical parameters used for fin analysis for constant temperature approximation	72
5. Physical properties and other paramaters adopted for fin analysis.....	72
6. Geometrical parameters for the fin configurations for the code validation	77
7. Geometrical Parameters for the plain fin configuration for the effect of tube row numbers study.....	102
8. Geometrical Parameters for the wavy-fin configuration for the effect of tube row numbers study.....	107
9. Geometrical parameters for the plain-fin staggered configuration for the effects of transverse pitch (Pt)	112
10. Geometrical parameters for the wavy-fin staggered configuration to study the effects of transverse pitch (Pt)	115
11. Geometrical parameters for the plain fin staggered configuration to study the effects of fin pitch (Fp)	119
12. Geometrical parameters for the wavy fin staggered configuration to study the effects of fin pitch (Fp)	123
13. Geometrical parameters for the wavy fin staggered configuration to study the effects of wavy angle (Wa)	127
14. Geometrical parameters for the wavy fin staggered configuration to study the effects of wavy height (Wh)	131

LIST OF FIGURES

Figure	Page
1. Isometric view of Louvered fin heat exchanger	13
2. Front view of a Louvered Heat Exchanger	14
3. An isometric view of Offset strip fin	15
4. Isometric view of perforated fin heat exchanger.	16
5. Three dimensional view of a plain-fin heat exchanger with inline tube arrangement.....	17
6. Wavy fin heat exchanger with tubes in inline arrangement.....	18
7. Main elements of CFD and its functions and interconnectivity [39].....	21
8. Algorithm adopted in the solver for general field solution process.....	23
9. Different layers in a steady turbulent fluid flow	36
10. Computational domain for plain heat exchanger with staggered tube arrangement.....	44
11. Simplified computational domain for plain heat exchanger with staggered tube arrangement.....	45
12. Computational domain for wavy heat exchanger with staggered tube arrangement.....	45
13. Simplified computational domain for wavy heat exchanger with staggered tube arrangement.....	46
14. Enhanced front view of wavy fin heat exchanger.....	46
15. Front view of the wavy fin heat exchanger with nomenclature.....	48
16. Top view of the wavy fin inline configuration with nomeclature.	49

LIST OF FIGURES-CONTINUED

Figure	Page
17. Computational domain along with boundary conditions for plain fin heat exchanger with staggered tube arrangement.....	51
18. Tetrahedral element incorporated for meshing	61
19. Mesh development using edge spacing.....	62
20. Mesh development using face spacing.	63
21. Temperature profiles at the outlet from wall plane to symmetry plane for different mesh configurations.	66
22. Velocity profiles at the outlet from wall plane to symmetry plane for different mesh configurations.	67
23. Temperature profile for wavy fin staggered configuration for grid independence.....	69
24. Velocity profile for wavy fin staggered configuration for grid independence.	70
25. Temperature distribution from fin base to fin tip.....	74
26. Comparison of friction factor (f) for the plain fin configuration to the experimental data by Wang et al. [3]	78
27. Comparison of Colburn factor (j) for plain fin staggered configuration with experimental data by Wang et al. [3]	78
28. Comparison of Friction factor (f) for wavy fin staggered configuration with experimental data by Wang et al. [4].....	79
29. Comparison of Colburn factor (j) for wavy fin staggered configuration with experimental data by Wang [4].....	80
30. The general trend of friction factor(f) and Colburn factor(j) against Reynolds number for flow in a circular pipes as mentioned by Kakac et al. [49]	84
31. Friction factor for plain and wavy fin staggered configuration for laminar and transitional regime.....	85

LIST OF FIGURES-CONTINUED

Figure	Page
32. Colburn factor for plain and wavy fin staggered configuration for the laminar and transitional regime	86
33. Streamlines on plain fin staggered configuration for $Re=2000$	87
34. Streamlines on plain fin staggered configuration for $Re=3000$	87
35. Streamlines on plain fin staggered configuration for $Re=5000$	88
36. Streamlines on plain fin staggered configuration for $Re=7000$	88
37. Temperature contour plot for plain fin staggered configuration for Reynolds number $Re=2000$	88
38. Temperature contour plot for plain fin staggered configuration for Reynolds number $Re=7000$	89
39. Turbulent viscosity for the case of wavy fin staggered configuration at $Re=2000$	89
40. Velocity contour lines for plain fin staggered configuration at Reynolds number, $Re=7000$	90
41. Temperature contour lines for plain fin staggered configuration at Reynolds number, $Re=7000$	91
42. Turbulent viscosity contour lines for plain fin staggered configuration at Reynolds number $Re=7000$	91
43. Computational domain for plain fin inline configuration with four tubes	92
44. Computational domain for plain fin staggered configuration with four tubes	92
45. Computational domain for wavy fin inline configuration with four tubes	92
46. Computational domain for wavy fin staggered configuration with four tubes	93
47. Streamlines in plain fin inline configuration at Reynolds number, $Re=2000$	93

LIST OF FIGURES-CONTINUED

Figure	Page
48. Streamlines in plain fin staggered configuratoom at Reynolds Number, $Re=2000$	94
49. Contour plot of temperature at symmetry plane for plain fin inline configuration.	94
50. Contour plot of temperature at symmetry plane in plain fin staggered configuration	95
51. Contour plot of temperature at symmetry plane for wavy fin inline configuration	95
52. Vector plot for wavy fin inline configuration	96
53. Vector plot for plain fin staggered configuration.	96
54. Enhanced vector plot for plain fin staggered configuration between second and third tube.	97
55. Friction factor comparison between inline and staggered configuration for the plain fin pattern.	98
56. Colburn factor comparison between inline and staggered configuration for the plain fin pattern.	99
57. Friction factor comparison between inline and staggered configuration for the wavy fin pattern.	100
58. Colburn factor comparison between inline and staggered configuration for the wavy fin pattern.	101
59. Computational domains used in plain fin staggered configuration to study the effects of tube rows using (a) one (b) two and (c) three tube rows.....	103
60. Computational domains used in plain fin staggered configuration to study the effects of tube rows using (d) four (e) five and (f) six tube rows	104

LIST OF FIGURES-CONTINUED

Figure	Page
61. Heat transfer coefficient against the number of tube rows for the plain fin staggered configuration.....	105
62. Pressure coefficient against the number of tube rows for the plain fin staggered configuration.....	106
63. Computational domains used in wavy fin staggered configuration to study the effect of tube rows using (a) one (b) two (c) three tube rows.....	108
64. Computational domains used in wavy fin staggered configuration to study the effect of tube rows using (d) four (e) five and (f) six tube rows.....	109
65. Heat transfer coefficient against number of tube rows for wavy fin staggered configuration.....	110
66. Pressure coefficient against the number of tube rows for the wavy fin staggered configuration.....	111
67. Effects of the friction factor for plain fin staggered configuration with change in transverse pitch(P_t).....	113
68. Effects of Colburn factor for plain fin staggered configuration with change in transverse pitch(P_t).....	114
69. Effects of the friction factor for wavy fin staggered configuration with change in transverse pitch(P_t).....	116
70. Effects of the Colburn factor for wavy fin staggered configuration with change in transverse pitch(P_t).....	117
71. Fin spacing for plain fin configuration	118
72. Effects of the friction factor for plain fin staggered configuration with change in fin pitch(F_p).....	120
73. Effects of the Colburn factor for plain fin staggered configuration with change in fin pitch(F_p).....	121

LIST OF FIGURES-CONTINUED

Figure	Page
74. Efficiency index for plain fin staggered configuration with change in fin pitch(Fp)	122
75. Effects of the friction factor for wavy fin staggered configuration with change in fin pitch(Fp).....	124
76. Effects of the Colburn factor for wavy fin staggered configuration with change in fin pitch(Fp).....	125
77. Efficiency index for wavy fin staggered configuration for change in fin pitch(Fp).	126
78. Wavy fin staggered configuration with the wavy angle ($Wa=17.5^\circ$)	127
79. Wavy fin staggered configuration with the wavy angle ($Wa=8.95^\circ$)	127
80. Wavy fin staggered configuration with the wavy angle ($Wa=32.21^\circ$)	127
81. Effects of the friction factor for wavy fin staggered configuration with change in wavy angle.....	128
82. Effects of the Colburn factor for wavy fin staggered configuration with change in wavy angle.....	129
83. Efficiency index for the wavy fin staggered configuration for change in wavy angle(Wa).....	130
84. Wavy fin staggered configuration with the wavy height ($Wh=0.7508\text{mm}$)	131
85. Wavy fin staggered configuration with the wavy height ($Wh=1.5\text{mm}$)	132
86. Wavy fin staggered configuration with the wavy height ($Wh=3.0032\text{mm}$)	132
87. Effects of the friction factor for wavy fin staggered configuration with change in wavy height	133
88. Effects of the Colburn factor for wavy fin staggered configuration with change in wavy height	134

LIST OF FIGURES-CONTINUED

Figure	Page
89. Efficiency index for the wavy fin staggered configuration for change in wavy height(Wh)	135

NOMENCLATURE

<u>Symbol</u>	<u>Description</u>
C_p	Dimensionless pressure coefficient
$\overline{C_p}$	Area averaged pressure coefficient
c	Specific heat at constant pressure
D	Tube Diameter
D_h	Hydraulic Diameter
F_1	Blending equation function
F_k	Flux for k- equation
F_p	Fin pitch
F_t	Fin thickness
F_U	Flux for momentum equation
f	Friction factor
H	Fin spacing
h	Local heat transfer coefficient

NOMENCLATURE-CONTINUED

<u>Symbol</u>	<u>Description</u>
$\bar{h}$	Average heat transfer coefficient
j	Colburn factor
j/f	Efficiency Index
k	Turbulence kinetic energy
L	Characteristic flow length
Δn	Normal distance between wall and first grid point
Nu	Nusselt number
$\overline{Nu}$	Average Nusselt number
P_l	Longitudinal pitch
P_t	Transverse pitch
p	Local dimensionless pressure
p_{in}	Inlet pressure
p_{∞}	Pressure in the free stream

NOMENCLATURE-CONTINUED

<u>Symbol</u>	<u>Description</u>
Pr	Prandtl Number
q''	Local heat flux
Re	Reynolds number based on hydraulic diameter, D_h
Re_D	Reynolds number based on tube diameter, D
Re_H	Reynolds number based on fin spacing, H
T	Temperature
T_b	Local bulk mean temperature
T_{in}	Inlet temperature
T_w	Wall temperature
U	Dimensionless velocity vector
U_{in}	Inlet velocity
u	Velocity component in x-direction
u_τ	Frictional velocity
u^+	Dimensionless near wall velocity

NOMENCLATURE-CONTINUED

<u>Symbol</u>	<u>Description</u>
u^*	Alternative velocity scale
v	Velocity component in y-direction
W_a	Wavy angle
W_h	Wavy height
w	Velocity component in z-direction
y^+	Dimensionless distance from the wall
α	k - ω turbulence model constant
β	k - ω turbulence model constant
Γ	Momentum diffusivity
Γ_{eff}	Effective momentum diffusivity
Γ_t	Turbulent diffusivity
δ	Kronecker delta function
ε	Turbulence dissipation rate
λ	Thermal conductivity

NOMENCLATURE-CONTINUED

<u>Symbol</u>	<u>Description</u>
ν	Kinematic viscosity
μ	Dynamic viscosity
μ_{eff}	Effective viscosity
μ_t	Turbulent viscosity
ρ	Density of the fluid
σ	Stress tensor
σ_k	k - ω turbulence model constant
σ_ω	k - ω turbulence model constant
τ	Shear stress
τ_{visc}	Viscous shear stress
τ_w	Wall shear stress
Φ	General scalar variable
ω	Turbulence frequency
θ	Dimensionless temperature

NOMENCLATURE-CONTINUED

<u>Symbol</u>	<u>Description</u>
θ_b	Dimensionless bulk mean temperature

ABSTRACT

Numerical investigation has been performed to study heat transfer and pressure drop characteristics of plain fin and wavy fin heat exchangers in the external airside. The characteristics were studied using the dimensional parameters, friction factor and Colburn factor. The flow rate was varied over the range of $2000 \leq Re \leq 7000$ in the turbulent and transitional regimes. The analyses were performed using a finite volume method. Comparisons with experimental data were performed to validate the numerical results. The geometrical parameters like the fin pitch, transverse pitch, wavy angle and wavy height were varied to study the effects of each individual parameter. Comparisons were also drawn to the laminar range, the effect of number of tube rows and the pattern in which the tube were arranged.

The investigation shows that the effect of the number of tube rows on heat transfer coefficient is less after the number of rows is increased beyond four. It is observed that the reducing the fin pitch increases the heat transfer and pressure drop in both the configurations. The increase in transverse pitch resulted in a decrease in thermal and hydraulic characteristics. Response to the variation of wavy angle and wavy height was similar, as the wavy angle and wavy height were increased, the number of corrugations also increased resulting in higher heat transfer and a higher pressure drop. The critical balance between high heat transfer and low pressure drop was analyzed using the efficiency index for each parameter variation. The tube layouts in the staggered form were observed to be better than the in-lined layout for both the configurations.

INTRODUCTION

Compact heat exchangers are widely employed in refrigeration, cryogenics, air-conditioning and many other applications. The compactness also makes it a very interesting option for aerospace applications. There are many different types of geometry for heat exchangers available and being used. The 'plate-fin and tube' geometry is one of the most common configurations. There are different types of plate-fin geometry, the most common being the plain fin, where the fins are parallel plates attached to a hot element in the form of tubes or some other shape. These fins act as a sink, absorbing the heat out of the hot element with the help of conductive heat transfer. And then dissipating this absorbed heat onto the outside environment which is at a lower temperature. Just like any efficient heat exchanger, the performance in the vague sense can be improved with the help of more surface area. It may not be always very efficient, but it does indeed increase the heat transfer. It has also been shown that the performance of heat exchangers can be greatly increased with the use of unconventionally shaped flow passages by Webb [1] and Wang et al. [2]. There are many different varieties of those unconventional shapes; the most popular one is the wavy-fin geometry. The wavy surface can lengthen the path of airflow and cause better airflow mixing. As the wavy surfaces increase the flow path, it also increases the above mentioned surface area, thereby aiding in better heat transfer. The better flow mixing can be attributed to the corrugations existing in the flow channel. However, the problem with the wavy fins is that it has a much higher pressure

drop compared to the plain fins, making plain fins the choice for selected applications. Hence for the present study, these two most commonly used geometries are selected.

These above mentioned heat exchangers are commonly operated with a hot liquid inside the tubes and air on the outside. The heat from the fluid is transferred to the fin by conductive heat transfer. The fins then dissipate the heat onto the environment by convective heat transfer. The dominant heat resistance of almost 80%-90% for an air-cooled heat exchanger is external, which is on the air side as mentioned by Wang et al. [3,4]. There is not much heat resistance in the tube side or the channel where liquid flow takes place. Hence, there should be a greater focus to reduce the heat resistance on the dominant air side. The flow of air between the fins is filled with obstructions in the form of tubes, and also the air is in constant contact with the fins. The same air is the carrier of heat from the fins and there by cooling the fins down. In order to design better heat exchangers and come up with efficient designs, a thorough understanding of the flow of air in these channels is required. Hence this study focuses on the heat transfer and friction characteristics of the air side in those two types of heat exchangers.

Literature Review

The heat exchangers have been used in practice for a long period of time. There are many different types based on application, construction and operation. This investigation is focused in the compact heat exchangers. In the last few decades, lot of work has been done to study the friction and heat transfer characteristics of the two heat

exchangers which comes under the umbrella of this current investigation. The following review on literature summarizes few journal articles and proceedings briefly.

The performance of plate fin and heat exchangers has been under study for the last few decades. In the seventies, Rich [5,6] investigated the effects of staggered configurations and the effects of fin pitch for fourteen different configurations. In his conclusions, he stated that the fin spacing had minimum effect on the heat transfer. He also stated that the pressure drop is independent of the number of rows. Also in the seventies, researchers [7,8] did correlations for the friction factor based on their experiments on heat exchangers for the first time. The correlations he put forth claimed accuracy of $\pm 35\%$. In the eighties, Webb, [1] based on five investigations, proposed correlations for Colburn factor (j) and friction factor (f) for plain fin heat exchangers. When compared with the geometries mentioned in their study, it was found that these correlations had higher accuracy than that of the McQuiston [8] correlation. Also in the eighties, Rosman [9] studied the effects of tube rows on the heat transfer performance. Most of their study was limited to low Reynolds number. Their conclusions stated significant improvement to the reports made by the authors in the seventies. In the case of wavy fin pattern, researchers [10,11] presented wet and dry surface data for two wavy finned coils, one with an inline tube arrangement, and the other one with a staggered arrangement. However the first comprehensive study related to the wavy fin pattern was conducted by Beecher and Fagan [12]. The experimental method consisted of heat exchangers with three rows and the tubes being arranged in a staggered manner. The studies conducted by them involved investigating the effects of air velocity and the

pattern of fin arrangement. The results were presented in terms of arithmetic mean temperature difference against Graetz number. The effect of removing the sharp edges by rounding them in a wall with corrugations was studied by Sparrow and Hossfeld [13]. The results indicated that, the rounding off of the walls helped to reduce the Nusselt number. The rounding off also aided in reducing the friction factor in the flow channel.

Seshimo and Fujii [14] provided test results for a total of 35 samples of plain fin heat exchanger. This study by Seshimo and Fujii [14] investigated the effect of fin length along with the tube row number on the heat transfer and friction performance of plain fin heat exchangers in the range of $300 \leq Re \leq 2000$. Kundu et al. [15] measured the heat transfer coefficient and pressure drop over an eight tube row in-lined tube array placed between two parallel plates using two different aspect ratios in the range of $220 \leq Re \leq 2800$. Kundu et al. [16] presented more results on flow between two parallel plates in a following publication. The results presented by Eckels and Rabas [17] for plain fin heat exchanger under wet conditions reported that the pressure drop increased under wet conditions. The results also pointed out increase in heat transfer under the same wet conditions. The report also suggests that the effect of wet conditions decreased as the Reynolds number was increased. Experimental studies conducted by Ali and Ramadhyani [18] on the convective heat transfer in the entrance region of two-dimensional corrugated channels. The Nusselt number in the corrugated channels were higher than those in the parallel-plate channels by approximately 140-240%, the equivalent increase in friction factor being 130-280%.

A multiple regression technique was developed by Webb [19] to provide correlations for heat transfer and flow friction data. The relations provided by Webb [19] could predict 80% of the wavy fin data with $\pm 5\%$ and 95% of data within $\pm 10\%$. Results of an experimental study to determine the Nusselt number and friction factor on the air side of wavy finned, chilled water cooling coil is reported by Mirth and Ramadhayani [20]. In this study, general correlations of the dry and wet surfaces were presented.

In the recent times, most extensive experimental data on this subject is reported by Wang et al. [21]. The report given by Wang et al. [3] presented the airside performance of 15 test cases of plain fin and tube heat exchangers. Several geometrical parameters including the number of tube rows, fin thickness were studied. In his study Wang et al. [3] supported his argument against the claim made by Webb [1] that the increase of Colburn factor (j) at higher number of tube rows may not truly be attributed to the experimental uncertainties. The paper also comments on the poor performance of the correlation equation provided, mentioning that it under-predicts the heat transfer performance. After reporting on plain fin heat exchangers, Wang et al. [4] presented results on wavy fin heat exchangers. The paper reported its results on pressure drop and friction characteristics on the air side of the heat exchanger. The study reported negligible relation between the tube rows and the friction factor (f); it also suggested correlation equations for the data set they produced. The same correlation was used to compare its performance against the data reported by other researchers. It claimed better predictive capability than many others using regression technique. The correlations were given to predict friction factor (f), and Colburn factor (j).

In a paper Kim et al. [22] presented results for plain fin tube heat exchangers and presented correlations for heat transfer and friction characteristics. More detailed study by Wang et al. [21] presented results with varying tube diameters and updated the correlation equations, with which he claimed higher accuracy. In his study, Wang et al. [21] presented correlation equations based on a total of 61 samples. The experimental values taken in the current investigation are from these publications.

In the studies undergone in the field of numerical analysis, researchers have come a long way. In the available literature, most of the early researchers have used two-dimensional and laminar techniques to analyze flow conditions in their numerical calculations. In the study of flow between the tube banks, Launder and Massey [23] has used hybrid polar Cartesian grid system. Using the same method, Fujii et al. [24], also studied the flow between the parallel plates cruising in between the tube banks. In an investigation conducted by Wung and Chen [25] employed the boundary fitted coordinate system to study the flow field and heat transfer for both staggered and in-lined tube arrays. For the first time, a researcher reported 2-D numerical results along with experimental data for the influence of fin spacing on the heat transfer and pressure drop over a four-row in-line cylinder between two parallel plates for $50 \leq Re \leq 500$, Kundu et al. [26]. These researchers were limited, because they were trying for solutions for a three dimensional problem with two dimensional approach. The authors noted in their study that the two dimensional field studies cannot sufficiently predict heat transfer between the fluid and the fin. The approximations used by the authors were to avoid the side wall effects; Zdravistch et al. [27] used Dirichlet and Neumann Boundary conditions at the

inlet and outlet for his study in the two dimensional flow field. The author specifically mentioned the importance of three dimensional simulations in case of side wall effects, which is exactly the case with heat exchangers which use fin and tube banks.

It is true that the flow channel consists of a complicated structure, hence numerical studies with two-dimensional approach is not sufficient. A simple fundamental model used by Yamashita et al. [28], consists of a pair of parallel plates and a square cylinder passing through the plates in the perpendicular direction. It can be assumed as an approximation of plate fin and tube. A circular tube was employed in the study by Bastini et al. [29]. They had assumed that the flow was fully developed and that it had periodic boundary conditions. The numerical study performed by Jang et al. [30] considered plate fin tube heat exchangers for both inline and staggered arrangements. The study used a variation of SIMPLE scheme and was limited to the laminar range. In a further study Jang did similar three-dimensional analysis on wavy fin and tube configuration. In a study Mendez et al. [31] investigated experimentally and numerically the effects of fin spacing using dye visualization technique and a general purpose fluid solver. In a more recent study, Tutar and Akkoca [32] studied the time dependent modeling of unsteady laminar flow and the heat transfer over multi-row plane fin heat exchanger. For the case of wavy fin configurations, McNab et al. [33] used a commercial software to model heat transfer and flow in an automotive radiator. The flow was considered over the fins, and no tubes were included for the study. Recently Amin and Panse [34] investigated the flow in transitional regime using a commercial code for both the plain fin tube configurations and wavy fin configurations. The study was conducted in the transitional regime of 1300

$\leq \text{Re} \leq 2000$. The study compared the applicability of two equation turbulence models for the transitional regime. The study focussed on the effects of parameters but was limited to the transitional regime. As these developments took place over the last three to four decades, simultaneous development took place in the computing power and the introduction of new numerical methods. The numerical methods developed by Wilcox [35] for solving the flow problems of turbulence were continually improved and many new models were introduced. Much improvement can be attributed to Menter [36] for the improvement of the two-equation model. The two-equation model was improved further by Grotjans [37]. Many commercially available computer codes such as ANSYS [38] incorporated these models. The commercial code ANSYS [38] gives the user the freedom to choose from many different types of models depending upon the application. A good review of the improvement of numerical methods and the current state of computational fluid dynamics can be found in Jiyan et al. [39].

Motivation for Present Research

Although the heat exchangers are widely used in industries, published results of air-side performance are very limited to certain geometries and in the laminar and transitional ranges of air flow. The effects of individual geometrical parameters are not completely known. The effect of variation in parameters may help in designing better heat exchangers with better heat transfer capabilities required by demanding applications of the new age.

Recently, Wang et. al provided extensive test results but these studies do not focus on the effects or influence of a particular geometrical parameter. The current study aims to rectify some of these deficiencies and focus on the parameter variation in the turbulent range of the flow. It is really time consuming, laborious and challenging to test all individual geometrical parameters and study their effects. Advanced computers of today with numerical simulations can help with this issue by solving the problems in finite volume method. The actual problem existing can be simplified into algebraic equations with the help of few assumptions and boundary conditions and achieve results with reasonable accuracy.

This aspect can help in running the numerical simulation for all the various geometrical parameters one is interested in. Also, there is still lot more to be understood about the turbulent flow, and solving it using numerical simulations is useful in studying the different turbulence models and its resourcefulness. Although only one tested turbulence model is used in this study, this model is known to provide reasonable accuracy with the experimental results. This correlation between the model and the

experimental results is the motivation for the present study. Thus a parametric study can be carried out to study the impact of individual geometrical parameters numerically. This is indeed difficult to perform experimentally, as the cost involved in it is not feasible. Also, minor changes are laborious and carrying out multiple tests under same conditions can get difficult.

BACKGROUND

Heat Exchangers

A heat exchanger is an efficient device for the transfer of energy in the form of heat from one medium to another medium which may or may not be in direct contact with each other. Heat exchangers are used in common household appliances like in computers, air conditioning equipment, refrigerators etc. and also play an important part in many industries, like manufacturing, food processing, chemical and petroleum industries to name a few. In the biological world, due to high surface area to volume ratio, ears, lungs and skin in living beings are major examples of heat exchangers. However, this study focuses on the design of common manufactured heat exchangers, whose classification is discussed below.

Following is the one type of classification for heat exchangers based on its construction.

1. Plate-fin heat exchanger.
2. Spiral heat exchanger.
3. Direct contact heat exchanger.
4. Shell and tube heat exchanger.

A heat exchanger is said to be of compact type, if it has a very high surface area to volume ratio. The plate-fin heat exchangers come under this category. More of this can be found in Kays and London [40].

Plate-Fin Heat Exchanger

In the case of a plate-fin heat exchanger, many layers of flat metal plates are arranged to create a series of finned chambers. The flow of fluids may differ based on the application. The fins allow the heat exchanger to withstand high pressures while providing stretched surfaces increasing its structural integrity.

The plates or the extended surfaces can be any one of the many different shapes. The common shapes are as follows.

1. Plain fin
2. Wavy fin
3. Louvered fin
4. Offset strip fin
5. Perforated fin

Let us look into the louvered, offset strip and perforated fin briefly before we move on to the fins which are of our interest.

Louvered Fin: In the case of the louvered fin heat exchanger, in order to break up the boundary layer formations louvered fins are used. This is advantageous as it provides high average heat transfer coefficients along the air side as compared to continuous fins as mentioned by Stephen and Thole [41]. This type of geometry is commonly applied in the case of automotive radiators. The majority of the studies mentioned by Guo [42] with louvered fins are in the laminar range. This is due to the fact that this type of geometry is more efficient in its heat transfer in the laminar range due to duct flow. As the Reynolds number goes higher, the flow became parallel to the louvers; as mentioned by

Achaichia and Cowell [43]. When this happens it is termed as boundary layer flow. This boundary layer flow normally happens when there are no louvered fins. In other words, at high Reynolds number, the influence of louvered fins is nullified.

In case of the flow at low Reynolds number, louvers act to interrupt the airflow and create a series of thin boundary layers that have lower thermal resistance. The geometry of the Louvered fins consist of louver angle patterns with angles ranging from 15° - 40° . Figure 1 and Figure 2 shows louvered fin.

Figure 1. Isometric view of Louvered fin heat exchanger

Figure 2. Front view of a Louvered Heat Exchanger

Offset Strip Fin: In the offset strip fin, a boundary layer on the short strip length is established, followed by its dissipation in what is termed as the wake region between the strips. Strip lengths generally vary between 3-6 mm, and the Reynolds number is kept within the laminar region [44]. As mentioned in the studies by Kays [40], the Colburn factor (j) of the offset strip fin is about 2.5 times higher than that of the plain fin for comparable geometries. Also the friction factor (f) is about 3 times higher than that of the plain fin. Shorter strip lengths can be used for much higher heat transfer ratio. Figure 3 shows an offset strip fin heat exchanger.

Figure 3. An isometric view of Offset strip fin

Perforated Fin: In the case of the perforated fin, a matrix of spaced holes is perforated in the fin. Then the fin is bent to form U-shaped flow channels. When the porosity becomes high for the fin, a higher rate of heat transfer occurs. This is due to the boundary layer dissipation in the wake region formed by the matrix of holes. The experimental studies by Kays [40] show that comparatively the performance of the perforated fin is less than that of an offset strip fin and thus the perforated fin is becoming obsolete these days. There are some recent studies on the turbulent region for the perforated fins by Shaeria et al. [45]. A simple image showing the perforated fins are given in Figure 4.

Figure 4. Isometric view of perforated fin heat exchanger.

Plain Fin: In the case of Plain fin, the geometry is simple. The setup consists of parallel plates which act as fins. The difference may be whether the tubes are present or not. In our study the tubes are present as shown in Figure 5. The bank of tubes shares common fins or extended surfaces as it is sometimes called. A hot liquid flows through the tubes, and a cool fluid, generally air flows through the channels formed by adjacent fins around the tube bank. The heat transfer between the gas and the fins and tube surfaces is determined by the flow structure of the gaseous fluid. Air is taken as the gaseous fluid in our study. If the tubes are not present, the different fluids flow through different layers in between the parallel plates thereby facilitating heat transfer. A three-dimensional image of plain fin-tube geometry is shown in Figure 5.

Figure 5. Three dimensional view of a plain-fin heat exchanger with inline tube arrangement

Wavy Fin: When the fins have periodic corrugations in their geometry in the form of a wave, then it is called a wavy fin. The wavy pattern may be smooth or of a herringbone pattern. These periodic corrugations having a definite angle of corrugation helps in having better mixing of flow, thereby higher heat transfer. These corrugations in a wavy fin help in increasing the flow length in a limited space than that of the plain fin. This type of geometry is being widely studied and used these days due to its attractive heat transfer performance as demonstrated in their studies conducted by Nishimura et al. [46] and Wang et al. [21]. The important parameters in the study of wavy fin are the wavy angle and the wavy height, which will be explored in detail in this study.

The basic ideas behind having different types of fins or extended surfaces are different. The most common one is to induce boundary layer separation in the channels, so as to provide better mixing, as is the case in wavy channels, and thereby increasing the heat transfer rate. Figure 6 shows a three dimensional image of a wavy fin heat exchanger.

Figure 6. Wavy fin heat exchanger with tubes in inline arrangement

Computational Fluid Dynamics

Computational fluid dynamics has matured to be used widely in industrial applications and academic research. Although the use of this field was not widely accessible a few decades ago, at present, it is an extensively accepted and adopted

methodology for solving intricate problems in modern day engineering arenas. System optimization and improving existing system designs are performed in almost all kinds of industries with the help of computer simulations. Computing simulations are widely used due to enhanced efficiency and lower operating costs

Computational fluid dynamics (CFD) is an area in fluid mechanics which deals with fluids that are in motion. The physical attributes of the fluid in motion or rather the flow can be expressed in terms of mathematical equations. The flow might also have many possible properties, like heat transfer, chemical process, phase changes, radiation, combustion, turbulence etc. The governing mathematical equations which are usually in partial differential form are discretized into algebraic equations. These equations are converted to software programs using high-level programming languages. The engineers use these generic software programs to their area of interest for research and development. These software programs solve the equations at rates that were unimaginable a few decades ago.

In actual practice, CFD allows the user to alternate the geometrical designs over various parameters which might be useful to analyze the flow physics. This is advantageous in the beginning stage of a fluid-system design to try different sample models. It is also useful to optimize an existing system which needs to be improved by trying out small changes numerically. Another major advantage is to perform these technological improvements requiring precise predictions of flow behaviors without the experimental development which might be very expensive to initiate. The idea of CFD is

not to replace empirical and experimental testing, but to provide an invaluable tool in terms of assisting research and analysis.

The success of CFD based research rests on many factors. Since the computer will generate results based on the input given to it and the conditions specified. Therefore, the success of any study being done rests on the quality of input data and the correctness of conditions specified. The major role therefore rests in the aptitude and knowledge of the user. The user needs to have thorough understanding of the flow physics and the fundamentals of numerical techniques and models.

The software package used for the current analysis is ANSYS workbench version 12.0. This package is owned and marketed by ANSYS Inc. [38]. The package consists of a three-dimensional modeling software, a meshing software, a pre-processing module, a numerical solver and a post-processing module. The main elements of a CFD are:

- Pre-processor
- Solver
- Post-processor

The steps involved are briefed in Figure 7 showing its main functions and its interconnectivity with other modules.

Figure 7. Main elements of CFD and its functions and interconnectivity [39]

These three modules constitute the core of CFD, the modeling software and the meshing software comes under the pre-processing stage. The geometrical modeling is performed in the first stage. In the second stage, meshing is performed, which create nodes, tetrahedral elements and other elements as required so that the equations can be solved in finite volume manner which is a type of finite difference method.

Some authors include the creation of geometry and mesh generation in the pre-processing stage itself. Other functions in the pre-processing stage include specifying of the boundary conditions at various locations in the geometry and specifying properties for the materials involved. The physical models like the existence of phase changes,

turbulence, combustion, radiation and other processes and conditions are also to be mentioned.

In the solver stage, the transport equations of mass, momentum and energy are solved with supporting physical models in the background. Also, the solution is initialized by specifying the convergence criteria, how the solution needs to be monitored, and also by the method in which the solution is controlled. A generic algorithm for any problem is shown in Figure 8, which is adopted by ANSYS [38].

Figure 8. Algorithm adopted in the solver for general field solution process

In the final post-processing stage, all the results can be obtained which includes but are not limited to obtaining the vector and contour plots of various variables. One can also obtain graphs for various variables against many parameters in the physical domain. Also these variables can be probed at a particular location which is of interest. All these features make post-processing and thereby CFD very useful.

NUMERICAL METHOD

In this chapter, the numerical method used for solving the problem mentioned in the introductory chapter will be explained in detail. To do that, the governing equations corresponding to the physical state are mentioned and also the turbulence model employed for the study is explained. The process employed in the finite volume method is explained in a systematic manner with the CFD techniques mentioned in the previous chapter.

Governing Equations

Governing equations are used to formulate a solution based on the physical state and the boundary conditions. These governing equations represent mathematical statements of the conservation laws of physics. Once these equations are formulated, these expressions are discretized using finite volume method to arrive at the solution. For the present problem, the working fluid is air, and the major mode of heat transfer is through convection. To arrive at the required governing equations we need to adopt certain physical laws and assumptions. The adopted physical laws are as follows:

- Mass is conserved for the fluid
- The rate of change of momentum equals the sum of forces acting on the fluid.
(Newton's second law)
- The rate of change of energy equals the sum of rate of heat addition and the rate of work done on the fluid. (First law of thermodynamics)

Along with the physical laws, following assumptions about the fluid and the analysis are made:

- The flow is three dimensional and steady-state in nature.
- The fluid is Newtonian and incompressible.
- No body or buoyant forces are acting.

Based on the above mentioned laws and assumptions, the necessary equations describing the three-dimensional flow are adopted as follows:

Continuity Equation:

$$\frac{du}{dx} + \frac{dv}{dy} + \frac{dw}{dz} = 0 \quad (1)$$

Momentum Equations:

x-component:

$$\rho \left(u \frac{\partial}{\partial x} u + v \frac{\partial}{\partial y} u + w \frac{\partial}{\partial z} u \right) = - \frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \quad (2)$$

y-component:

$$\rho \left(u \frac{\partial}{\partial x} v + v \frac{\partial}{\partial y} v + w \frac{\partial}{\partial z} v \right) = - \frac{\partial p}{\partial y} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \quad (3)$$

z-component:

$$\rho \left(u \frac{\partial}{\partial x} w + v \frac{\partial}{\partial y} w + w \frac{\partial}{\partial z} w \right) = - \frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \quad (4)$$

Energy Equation:

$$\rho C_p \left(u \frac{\partial}{\partial x} T + v \frac{\partial}{\partial y} T + w \frac{\partial}{\partial z} T \right) = \lambda \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) \quad (5)$$

These equations will be non-dimensionalized in the future sections. Since the present study is about flow in the turbulent range of $2000 \leq Re_H \leq 7000$, a turbulence model is employed. The turbulence model adopted is Baseline(BSL) $k-\omega$ model. The additional equations which are required to solve these turbulence models are explained in the following sections.

Turbulence

A turbulent flow is defined as flow with random variations of various flow quantities such as velocity and pressure. It is well known that, turbulence is a property of flow and not of the fluid. Most of the flows of engineering significance are turbulent in nature. The turbulent flow regime is, therefore, not just of theoretical interest but also important to solve everyday engineering problems. In heat exchangers, the turbulence is essential for good heat transfer and mixing. Hence, the study of turbulence in heat exchanger design is extremely important. Turbulence is very effective in moving the air against and then away from the exchange surface in a relatively brief time period. The air near the surface that has already experienced considerable heat transfer is swept away from the surface to be replaced by fresh air.

Turbulence Models

Turbulence consists of fluctuations in the flow field in time and space. It is a complex process, mainly because it is three dimensional, unsteady and consists of many scales. It can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number.

In principle, the unsteady Navier-Stokes equations describe both laminar and turbulent flows without the need for additional information. However, turbulent flows at realistic Reynolds numbers span a large range of turbulent length and time scales, and would generally involve length scales much smaller than the smallest finite volume mesh, which can be practically used in a numerical analysis. The Direct Numerical Simulation (DNS) of these flows would require computing power which is many orders of magnitude higher than available in the foreseeable future [35].

To enable the effects of turbulence to be predicted, a large amount of CFD research has concentrated on methods which make use of turbulence models. Turbulence models have been specifically developed to account for the effects of turbulence without recourse to a prohibitively fine mesh and direct numerical simulation.

There are many types of turbulence models. The major classification is:

- Eddy viscosity models
 - The zero-equation model
 - Eddy viscosity transport model (One-equation model)
 - Two-equation models

- Reynolds Stress Turbulence Models

Two-equation models which come under eddy-viscosity models are the most widely used models, and they are superior than the Reynolds-stress models. The important two-equation models are the following:

- k - ϵ model
- RNG k - ϵ model
- Wilcox k - ω model
- Baseline (BSL) k - ω model
- Shear stress transport model (SST)

The model chosen for the present study is the Baseline (BSL) k - ω model, since it was recommended by many authors including Menter [36], Amin and Panse [34]. In this study only Baseline (BSL) k - ω model will be explained. Interested readers can check the paper by Wilcox [35] for other models. The main advantage of this model is its treatment near the walls.

The Baseline (BSL) k - ω Model:

A baseline k - ω model is a type of eddy viscosity model. An eddy viscosity model is defined based on a theory that the turbulence consists of small eddies which are continuously forming and dissipating, and in which the Reynolds stresses are assumed to be proportional to mean velocity gradients. This hypothesis of eddy viscosity assumes that the Reynolds stresses can be related to the mean velocity gradients and eddy

viscosity by the gradient diffusion hypothesis, a similar analogy can be seen in the laminar Newtonian flow between the stress and strain tensors.

Similar to the eddy viscosity hypothesis, there is the eddy diffusivity hypothesis, which states that the Reynolds fluxes of a scalar are linearly related to the mean scalar gradient. This formulation expresses the turbulent fluctuation terms as functions of the mean variables if the turbulent viscosity, μ_t is known.

Considering these hypotheses, the effective viscosity μ_{eff} , and the effective diffusivity Γ_{eff} are defined for the eddy viscosity model as,

$$\mu_{eff} = \mu + \mu_t \quad (6)$$

$$\Gamma_{eff} = \Gamma + \Gamma_t \quad (7)$$

where, μ_t , is the eddy or turbulent viscosity and Γ_t is the eddy or turbulent diffusivity.

The formulation of the Baseline (BSL) $k-\omega$ model began with the $k-\omega$ model developed by Wilcox [35]. It solves two transport equations, one for the turbulent kinetic energy, ' k ', which is defined as the variance of fluctuations in the velocity. The second transport equation solves for the turbulent frequency, ω . The stress tensor is computed from the eddy-viscosity concept.

The problem with the other two-equation models, especially the Wilcox model, is its sensitivity to free-stream conditions. As the value specified for ω at the inlet is modified, a significant variation in the results of the model can be obtained. This is undesirable and in order to solve the problem, a blending between the $k-\omega$ model near the surface and the $k-\varepsilon$ model in the outer region was developed by Menter [36]. Here ε is the

turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate) in the k - ε model. It consists of a transformation of the k - ε model to a k - ω formulation and a subsequent addition of the corresponding equations. The Wilcox model is thereby multiplied by a blending function F_1 and the transformed k - ε model by a function $1-F_1$. F_1 is equal to one near the surface and decreases to a value of zero outside the boundary layer which is a function of the wall distance. At the boundary layer edge and outside the boundary layer, the standard k - ε model is therefore recovered [36].

Wilcox Model:

$$\begin{aligned} \rho \left[\frac{\partial k}{\partial t} + \left(u \frac{\partial k}{\partial x} + v \frac{\partial k}{\partial y} + w \frac{\partial k}{\partial z} \right) \right] \\ = \left(\mu + \frac{\mu_t}{\sigma k_1} \right) \left(\frac{\partial^2 k}{\partial x^2} + \frac{\partial^2 k}{\partial y^2} + \frac{\partial^2 k}{\partial z^2} \right) + P_k - \beta' \rho k \omega \end{aligned} \quad (8)$$

$$\begin{aligned} \rho \left[\frac{\partial \omega}{\partial t} + \left(u \frac{\partial \omega}{\partial x} + v \frac{\partial \omega}{\partial y} + w \frac{\partial \omega}{\partial z} \right) \right] \\ = \left(\mu + \frac{\mu_t}{\sigma \omega_1} \right) \left(\frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2} + \frac{\partial^2 \omega}{\partial z^2} \right) + \alpha_1 \frac{\omega}{k} P_k \\ - \beta_1 \rho \omega^2 \end{aligned} \quad (9)$$

Transformed k- ω model:

$$\begin{aligned} \rho \left[\frac{\partial k}{\partial t} + \left(u \frac{\partial k}{\partial x} + v \frac{\partial k}{\partial y} + w \frac{\partial k}{\partial z} \right) \right] \\ = \left(\mu + \frac{\mu_t}{\sigma k_2} \right) \left(\frac{\partial^2 k}{\partial x^2} + \frac{\partial^2 k}{\partial y^2} + \frac{\partial^2 k}{\partial z^2} \right) + P_k - \beta' \rho k \omega \end{aligned} \quad (10)$$

$$\begin{aligned}
& \rho \left[\frac{\partial \omega}{\partial t} + \left(u \frac{\partial \omega}{\partial x} + v \frac{\partial \omega}{\partial y} + w \frac{\partial \omega}{\partial z} \right) \right] \\
&= \left[\alpha_2 \frac{\omega}{k} P_k - \beta_2 \rho \omega^2 + \left(\mu + \frac{\mu_t}{\sigma \omega_2} \right) \left(\frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2} + \frac{\partial^2 \omega}{\partial z^2} \right) \right. \\
&\quad + 2 \rho \sigma \omega_2 \frac{1}{\omega} \left(\frac{\partial k}{\partial x} n_x + \frac{\partial k}{\partial y} n_y + \frac{\partial k}{\partial z} n_z \right) \left(\frac{\partial \omega}{\partial x} n_x + \frac{\partial \omega}{\partial y} n_y \right. \\
&\quad \left. \left. + \frac{\partial \omega}{\partial z} n_z \right) \right]
\end{aligned} \tag{11}$$

Now the equations of the Wilcox model are multiplied by function F_1 . The transformed k - ω equations by a function $1-F_1$ and the corresponding k and ω equations are added to give the BSL model:

$$\begin{aligned}
& \rho \left[\frac{\partial k}{\partial t} + \left(u \frac{\partial k}{\partial x} + v \frac{\partial k}{\partial y} + w \frac{\partial k}{\partial z} \right) \right] \\
&= \left(\mu + \frac{\mu_t}{\sigma k_3} \right) \left(\frac{\partial^2 k}{\partial x^2} + \frac{\partial^2 k}{\partial y^2} + \frac{\partial^2 k}{\partial z^2} \right) + P_k - \beta' \rho k \omega
\end{aligned} \tag{12}$$

$$\begin{aligned}
& \rho \left[\frac{\partial \omega}{\partial t} + \left(u \frac{\partial \omega}{\partial x} + v \frac{\partial \omega}{\partial y} + w \frac{\partial \omega}{\partial z} \right) \right] \\
&= \left[\alpha_3 \frac{\omega}{k} P_k - \beta_3 \rho \omega^2 \right. \\
&+ \left(\mu + \frac{\mu_t}{\sigma \omega_3} \right) \left(\frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2} + \frac{\partial^2 \omega}{\partial z^2} \right) \\
&+ (1 - F_1) 2 \rho \sigma \omega_3 \frac{1}{\omega} \left(\frac{\partial k}{\partial x} n_x + \frac{\partial k}{\partial y} n_y + \frac{\partial k}{\partial z} n_z \right) \left(\frac{\partial \omega}{\partial x} n_x \right. \\
&\left. \left. + \frac{\partial \omega}{\partial y} n_y + \frac{\partial \omega}{\partial z} n_z \right) \right]
\end{aligned} \tag{13}$$

The coefficients of the new model are a linear combination of the corresponding coefficients of the underlying models:

$$\Phi_3 = F_1 \Phi_1 + (1 - F_1) \Phi_2 \tag{14}$$

Where the model constants are given as, $\beta' = 0.09$; $\alpha_1 = \frac{5}{9}$; $\beta_1 = 0.075$; $\sigma_{k1} = 2$;

$\sigma_{\omega 1} = 2$; $\alpha_2 = 0.44$; $\beta_2 = 0.0828$, $\sigma_{k2} = 1$; $\sigma_{\omega 2} = 1/0.856$

Modeling Flow Near the Wall

Since the current geometry has few boundary conditions as walls in it, it is extremely important to pay close attention to the flow dynamics near the wall. Near a no-slip wall, there are strong gradients in the dependent variables. In addition, viscous effects on the transport processes are huge. The illustration of these processes within a numerical simulation puts forth the following major problems:

- To figure out how to account for viscous effects at the wall.
- To figure out a method by which to resolve the rapid variation of flow variables which occur within the boundary layer region.

Experiments and mathematical analysis have shown that the near-wall region can be subdivided into three layers. In the innermost layer, the so-called viscous sub-layer, the flow is almost laminar-like, and the (molecular) viscosity plays a dominant role in momentum and heat transfer. Further away from the wall, turbulence dominates the mixing process. This layer is known as the turbulent layer. Finally, there is a region between the viscous sub-layer and the turbulent layer called the buffer layer or the logarithmic layer, where the effects of molecular viscosity and turbulence are of equal importance. The Figure 9 shown below illustrates these subdivisions of the near-wall region.

Figure 9. Different layers in a steady turbulent fluid flow

We have to assume that the logarithmic profile reasonably approximates the velocity distribution near the wall; it provides a way to numerically compute the fluid shear stress as a function of the velocity at a given distance from the wall. This is known as a 'wall function' and the logarithmic nature in the layer is called the 'log law of the wall'.

Three approaches are commonly used to model the flow in the near-wall region:

- The wall function method
- The low-Reynolds number method
- Combination of wall function and low-Reynolds number method (automatic near wall treatment)

The wall function method uses empirical formulas that impose suitable conditions near to the wall without resolving the boundary layer, thus saving computational resources. A majority of the turbulence models can use this method. The major advantages of the wall function approach is that the high gradient shear layers near walls

can be modeled with relatively coarse meshes, yielding substantial savings in CPU time and storage. It also avoids the need to account for viscous effects in the turbulence model.

The viscous layers are also important for the study, which is the reason for employing $k-\omega$ model and a fine mesh near the walls which will be discussed in the next chapter. The low Reynolds number method resolves the details of the boundary layer profile by using very small mesh length scales in the direction normal to the wall which are called as inflation layers. These inflation layers are very thin in nature. The computations are extended through the viscosity-affected sub-layer close to the wall. The low- Re approach requires a very fine mesh in the near-wall zone and correspondingly a large number of nodes. Computer-storage and runtime requirements are higher than those of the wall-function approach and care must be taken to ensure good numerical resolution in the near-wall region to capture the rapid variation in variables.

To include the advantages of the two methods and eliminate the disadvantages of both the methods, an automatic near wall treatment is applied and is being used [47]. To explain the automatic near wall treatment, we have to have an understanding of the other two methods.

Mathematical Design for Automatic Near Wall Treatment:

The logarithmic relation for the near wall velocity is given by:

$$u^+ = \frac{U_t}{u_\tau} = \frac{1}{K} \ln(y^+) + C \quad (15)$$

Where y^+ and u_τ are defined as:

$$y^+ = \frac{\rho \Delta y u_\tau}{\mu} \quad (16)$$

$$u_\tau = \left(\frac{\tau_\omega}{\rho} \right)^{1/2} \quad (17)$$

u_τ is the frictional velocity, U_t is the known velocity tangent to the wall at a distance of Δy from the wall, y^+ is the dimensionless distance from the wall, τ_ω is the wall shear stress, K is the Von Karman constant and C is a log-layer constant depending on wall roughness.

Scalable Wall-Functions:

Equation (13) has an issue that it becomes singular at separation points where the near wall velocity U_t approaches zero. In the logarithmic region, an alternative velocity scale, u^* can be used instead of u^+ :

$$u^* = C_\mu^{1/4} k^{1/2} \quad (18)$$

This alternative velocity scale has the property that it does not go to zero if U_t goes to zero (in turbulent flow k cannot be completely zero). Based on this definition, the following explicit equation for the wall-shear-stress is obtained.

$$\tau_\omega = \tau_{visc} \left(\frac{y^*}{u^+} \right) \quad (19)$$

where:

$$\tau_{visc} = \frac{\mu U_t}{\Delta y} \quad (20)$$

$$y^* = \frac{\rho u^* \Delta y}{\mu} \quad (21)$$

A notable drawback of the wall-function method is that the predictions depend on the location of the point nearest to the wall and are sensitive to the near-wall meshing; it is to be noted that as mentioned by Grotjans and Menter [37], the refining of the mesh does not necessarily give a unique solution of increased accuracy.

The problem of inconsistencies in the wall-function in the case of fine grids can be overcome with the use of the scalable wall-function formulation developed by the computational fluid dynamics code CFX [38]. It can be applied on arbitrarily fine grids and allows one to perform a consistent grid refinement independent of the Reynolds number of the application. The basic idea behind the scalable wall-function approach is to limit the value of y^* used in the logarithmic formulation by a critical value. At that particular point, the surface coincides with the edge of the viscous sub-layer.

Yplus Definitions

The automatic wall treatment makes use of two different types of y^+ based on the wall functions. The two of them are,

- Standard wall function based
- Scalable wall function based

Standard Wall Function Based: “ y^+ ” that is based on the distance from the wall to the first node is the definition of this variable. This is the more standard y^+ which is mentioned in the scientific literature. It is defined as

$$y^+ = \frac{\sqrt{\tau_w/\rho} \cdot \Delta n}{\nu} \quad (22)$$

Where Δn is the normal distance between the wall and it is the first grid point off the wall.

Scalable Wall Function Based: Another variable which is used in the CFX Solver is the solver y^+ . It is based on a different definition for the distance and for a different velocity scale u^* which was already mentioned in the previous section. This variable is different and is required so as to make it suitable for separated flows which are singular in nature because of the logarithmic profile at locations where wall shear stress is zero. This variable is based on the distance from the edge to the center of the near wall control volume. Since the first control volume off the wall is half the distance to the first node, and the center of the control volume is half of that, Solver y^+ is typically 1/4 of the value of the standard y^+ . This variable is defined as,

$$\text{Solver } y^+ = \frac{u^* \cdot \Delta n/4}{\nu} \quad (23)$$

Thus the automatic near wall treatment also uses the scalable wall functions to solve the dynamics near to the wall. Thus, we have the advantages of the $k-\omega$ model of

being able to use the ω in the viscous sub layer and also the advantages of automatic near wall treatment, which enables the advantage of backward consistency.

Another important idea to note is the formulation involved to blend the wall value for ω between the logarithmic and the near wall formulation. The flux for the k -equation is artificially kept at zero and the flux in the momentum equation is computed from the velocity profile. The equations are as follows:

Flux for the momentum equation:

$$F_U = -\rho u_\tau u^* \quad (24)$$

Where,

$$u_\tau = \sqrt{\nu \left| \frac{\Delta U}{\Delta y} \right|} \quad (25)$$

$$u^* = \max(\sqrt{a_1 k}, u_\tau) \quad (26)$$

Flux for the k -equation is maintained zero:

$$F_k = 0 \quad (27)$$

In the ω -equation, an algebraic expression is specified instead of an added flux. It is a blend between the analytical expressions for ω in the logarithmic region:

$$\omega_1 = \frac{u^*}{a_1 K y} = \frac{1}{a_1 K \nu} \cdot \frac{u^{*2}}{y^+} \quad (28)$$

and the corresponding expression in the sub-layer:

$$\omega_s = \frac{6\nu}{\beta(\Delta y)^2} \quad (29)$$

with Δy being the distance between the first and the second grid point. In order to achieve a smooth blending and to avoid cyclic convergence behavior, the following formulation is selected:

$$\omega_\omega = \omega_s \left[1 + \left(\frac{\omega_I}{\omega_s} \right)^2 \right]^{1/2} \quad (30)$$

In the wall-function formulation, the first node is treated as being outside the edge of the viscous sub-layer. It is to be noted, that the physical location of the first grid point is always at the wall ($y=0$).

COMPUTATIONAL DOMAIN

To model and numerically evaluate the entire heat exchanger is not feasible. Hence, the area of interest is identified first as shown in Figure 10 and then a domain is sketched. This domain closely resembles the actual physical problem. This domain, which is solved using finite difference method, in the computational software is termed as the computational domain. In the domain, every plane represents a boundary, so that the software can evaluate it using the boundary conditions specified and the governing equations. The computational domain for this particular study differs slightly with the changing geometry but the basic boundary conditions remain the same. The devising of computational domain is shown in Figure 11. The boundaries of the computational domain include inlet and outlet on the left and right extremes. It also includes solid no slip walls in the form of tubes and one plane, the other three planes are symmetry planes. The symmetry function helps in reducing the computational domain to a great extent. So, using symmetry condition, only one half of a tube is being studied. The boundary conditions will be once again described in the next section. Figure 10 shows the top view of the computational domain used for plain fin heat exchanger with staggered tube arrangement.

Figure 10. Computational domain for plain heat exchanger with staggered tube arrangement

The section where there is a hatched line is used for the current investigation and is shown again in Figure 11. The two edges go exactly through the center of the tubes, so that the symmetry plane boundary condition can be applied which will be explained in a later section. This type of reduction helps in modeling the geometry. This reduction also reduces the computational costs and the time involved for numerical simulation without sacrificing the accuracy of the results.

Figure 11. Simplified computational domain for plain heat exchanger with staggered tube arrangement

Figure 12. Computational domain for wavy heat exchanger with staggered tube arrangement

Figure 12 shows the top view of a wavy fin heat with staggered tube arrangement, where the vertical lines are the wavy corrugations in the plate fin. The domain can be

simplified for the reasons stated earlier. The hatched lines show the simplified form which is used for the numerical investigation. The hatched section is shown separately in Figure 13.

Figure 13. Simplified computational domain for wavy heat exchanger with staggered tube arrangement

Figure 14 shows the front view of the wavy heat exchanger in an enhanced manner so as to understand the wavy angle and the wavy height. These two will be also used for the parametric study.

Figure 14. Enhanced front view of wavy fin heat exchanger

Nomenclature

This section describes the various nomenclature used for the geometry in both plain fin heat exchanger and wavy fin heat exchanger. The nomenclature is similar for both in-line and staggered types of arrangement of the tubes. The nomenclature is essential to understand as the same terms will be used in the results section. The front view of the wavy fin staggered configuration heat exchanger with perpendicular tubes with in-line arrangement is shown in Figure 15. Figure 16 shows the top view of the wavy fin heat exchanger with in-line arrangement. The complete domain is then simplified. The images below can be used to understand the nomenclature used. Longitudinal pitch is the distance between the center of two tubes lengthwise. Transverse pitch is the distance between tubes in the transverse direction. Fin thickness, as the name suggests is the thickness of each fin. Fin spacing is the distance between the fins. The fin pitch is the sum of fin thickness and fin spacing. Wavy angle is the angle the fins create with horizontal axis. The vertical distance the wavy corrugation extend is termed as wavy height.

Figure 15. Front view of the wavy fin heat exchanger with nomenclature.

Figure 16. Top view of the wavy fin inline configuration with nomenclature.

Boundary Conditions

The difference between different types of flow lies in its boundary conditions. Generally, any boundary condition can be specified in terms of Dirichlet and Neumann boundary conditions. For briefness, the Dirichlet boundary condition can be specified as a transport property ϕ for a particular physical quantity over a boundary, like

$$\phi = \text{an analytic function} \quad (31)$$

The Neumann boundary condition similarly is represented by prescribing a derivative of a transport property at a given boundary as

$$\frac{\partial \phi}{\partial n} = 0 \quad (32)$$

This section describes the different boundary conditions used in this study and each of the boundary condition is followed by a brief explanation. The computational domain with the corresponding boundary conditions is shown in Figure 17.

Figure 17. Computational domain along with boundary conditions for plain fin heat exchanger with staggered tube arrangement

Inlet Boundary Condition

The practice widely adopted for inlet boundaries is to set the transport quantities of a predetermined profile over a surface. The inlet acts as a mass source for the domain. The major considerations which are to be adopted are the flow direction, the flow velocity, physical properties and its variations.

For this study, a uniform flow with constant velocity derived from the Reynolds number of interest is taken. The derivation is as follows.

$$U_{in} = \frac{Re_H \cdot \mu}{\rho \cdot H} \quad (33)$$

Also, a constant temperature T_{in} is also assumed. The direction of the flow is also mentioned, by mentioning the velocity component in the x -direction as shown,

$$u = U_{in} ; v = w = 0 \quad (34)$$

Outlet Boundary Condition

At the downstream end of the computational domain, in order to maintain the conservation of mass, an outlet is specified. Here, the outlet is maintained at a distance seven times the diameter, D of the tube [30] so as to nullify the back pressure effect or reverse flow. This distance helps in flow expansion and interior solution stability by preventing the back pressure effect which will be caused in the simulation.

The outlet boundary condition is applied as Neumann boundary condition where the convective derivative normal to the boundary face is enforced to zero. The p_{∞} in terms of static pressure is specified as zero at the outlet. The values at this location will be evaluated later based on the stream-wise extrapolation of the transport quantities.

Wall Boundary Conditions

This type of boundary condition is employed for solid walls bounding the flow. For this particular study, this type of boundary condition exists physically in the form of two entities; the tubes and the fins. A constant wall temperature T_w at the fins and the tubes is also assumed. Since these are stationary in nature, the no-slip wall condition can be applied. In other words, the velocity at the surface is equal to zero.

$$u = v = w = 0 \quad (35)$$

This particular condition implies that the fluid flow comes to rest at the solid walls. This is applied for the four half tubes and the bottom wall which represents a fin in the computational domain.

Symmetry Boundary Conditions

These boundary conditions are applied because the geometry possesses symmetry or rather periodic repetitions. This condition allows the flow problem to be simplified to a fraction of the total domain.

Two major requirements of this condition are as follows:

- The normal velocity is zero
- The normal gradients for all transport quantities are zero.

The two center planes and one top plane are assumed to have this boundary condition. The advantage of having a small fraction also reduces the computational load to a very large extent. The pressure drop is expressed in terms of the dimensionless pressure coefficient, C_p , defined as

$$C_p = \frac{p_{in} - p_{\infty}}{\frac{1}{2}\rho(U_{in}^2)} \quad (36)$$

where, p_{in} is the pressure at inlet, p_{∞} is the pressure in the freestream or the pressure applied at the outle. The local heat transfer coefficient is defined as

$$h = \frac{q''}{T_w - T_b} \quad (37)$$

where q'' is the local heat flux and T_b is the local bulk mean temperature of the fluid. The local heat transfer coefficient can be expressed in the dimensionless form by the Nusselt number Nu , defined as

$$Nu = \frac{h \cdot H}{\lambda} = \frac{\partial \left(\frac{\Theta}{\Theta_b} \right)}{\partial n} \Big|_{wall} \quad (38)$$

Here, H is the channel height or fin spacing, Θ is the dimensionless temperature defined as

$$\Theta = \frac{(T - T_w)}{(T_{in} - T_w)} \quad (39)$$

and Θ_b is the local dimensionless bulk mean temperature defined as

$$\Theta_b = \frac{(T_b - T_w)}{(T_{in} - T_w)} \quad (40)$$

and n is the dimensionless unit vector normal to the wall. The average values of heat transfer coefficient $\bar{h}$, the average pressure coefficient $\bar{C}_p$ and the average Nusselt number $\bar{Nu}$ can be obtained by,

$$\bar{h} = \frac{\int h \cdot dA_s}{\int dA_s} \quad (41)$$

$$\bar{C}_p = \frac{\int C_p \cdot dA_s}{\int dA_s} \quad (42)$$

$$\overline{Nu} = \frac{\int Nu \cdot dA_s}{\int dA_s} \quad (43)$$

where dA_s is the infinitesimal area of the wall surface.

Pressure drop characteristics can be expressed in terms of another dimensionless term. This dimensionless term is known as the friction factor f . It is defined as

$$f = C_p \cdot \frac{H}{4L} \quad (44)$$

where L is the characteristic length. Similarly, the heat transfer characteristics can be expressed in terms of a dimensionless term known as the Colburn factor (j). It is given by,

$$j = \frac{\overline{Nu}}{Re_H \cdot Pr^{1/3}} \quad (45)$$

The non-dimensional parameters Reynolds number, Re_H , and Prandtl number, Pr , are defined as follows:

$$Re_H = \frac{\rho \cdot U \cdot H}{\mu} \quad (46)$$

$$Pr = \frac{\mu C_p}{\lambda} \quad (47)$$

Non-Dimensionalization of Governing Equations and Associated Boundary Conditions

For uniformity, the governing equations and the boundary conditions used for the current investigation are presented in terms of non-dimensional parameters as follows:

Space variables in dimensionless form:

$$x^* = \frac{x}{X} \quad (48)$$

$$y^* = \frac{y}{Y} \quad (49)$$

$$z^* = \frac{z}{Z} \quad (50)$$

Velocity components in dimensionless form:

$$u^* = \frac{u}{U_{in}} \quad (51)$$

$$v^* = \frac{v}{U_{in}} \quad (52)$$

$$w^* = \frac{w}{U_{in}} \quad (53)$$

Modified pressure in dimensionless form:

$$p'^* = \frac{p + \rho k}{\rho U_{in}^2} \quad (54)$$

where k is the turbulence kinetic energy.

Turbulent viscosity in dimensionless form:

$$\mu_t^* = \frac{\mu_t}{\mu} \quad (55)$$

The governing equations can be expressed in dimensionless parameters as follows:

Continuity Equation:

$$\frac{du^*}{dx^*} + \frac{dv^*}{dy^*} + \frac{dw^*}{dz^*} = 0 \quad (56)$$

Momentum Equations:

x-component:

$$\begin{aligned} & \left(u^* \frac{\partial}{\partial x^*} u^* + v^* \frac{\partial}{\partial y^*} u^* + w^* \frac{\partial}{\partial z^*} u^* \right) \\ &= -\frac{\partial p'^*}{\partial x^2} + \frac{1}{Re_H} \left(\frac{du^*}{dx^*} + \frac{du^*}{dy^*} + \frac{du^*}{dz^*} \right) \end{aligned} \quad (57)$$

y-component:

$$\begin{aligned} & \left(u^* \frac{\partial}{\partial x^*} v^* + v^* \frac{\partial}{\partial y^*} v^* + w^* \frac{\partial}{\partial z^*} v^* \right) \\ &= -\frac{\partial p'^*}{\partial y^2} + \frac{1}{Re_H} \left(\frac{dv^*}{dx^*} + \frac{dv^*}{dy^*} + \frac{dv^*}{dz^*} \right) \end{aligned} \quad (58)$$

z-component:

$$\begin{aligned}
& \left(u^* \frac{\partial}{\partial x^*} w^* + v^* \frac{\partial}{\partial y^*} w^* + w^* \frac{\partial}{\partial z^*} w^* \right) \\
& = -\frac{\partial p'^*}{\partial z^2} + \frac{1}{Re_H} \left(\frac{dw^*}{dx^*} + \frac{dw^*}{dy^*} + \frac{dw^*}{dz^*} \right)
\end{aligned} \tag{59}$$

Energy Equation:

$$\left(u^* \frac{\partial}{\partial x^*} \theta + v^* \frac{\partial}{\partial y^*} \theta + w^* \frac{\partial}{\partial z^*} \theta \right) = \frac{1}{Re_H \cdot Pr} \left(\frac{\partial^2 \theta}{\partial x^{*2}} + \frac{\partial^2 \theta}{\partial y^{*2}} + \frac{\partial^2 \theta}{\partial z^{*2}} \right) \tag{60}$$

Non-Dimensional Boundary Conditions:

Non-dimensional boundary conditions are described with respect to Figure 17. The non-dimensional boundary conditions applied in the present study are given as follows:

Inlet Boundary Condition:

$$u^* = 1 \tag{61}$$

$$\theta = 1 \tag{62}$$

$$v^* = w^* = 0 \tag{63}$$

Outlet Boundary Condition:

$$\frac{du^*}{dx^*} = \frac{dv^*}{dx^*} = \frac{dw^*}{dx^*} = 0 \quad (64)$$

Wall Boundary Condition:

$$u^* = v^* = w^* = 0 \quad (65)$$

$$\theta = 0 \quad (66)$$

Symmetry Boundary Condition: On x-y plane (the top plane):

$$\frac{du^*}{dz^*} = \frac{dv^*}{dz^*} = \frac{d\theta^*}{dz^*} = 0 \quad (67)$$

$$w^* = 0 \quad (68)$$

On x-z plane (tube center planes):

$$\frac{du^*}{dy^*} = \frac{dv^*}{dy^*} = \frac{d\theta^*}{dy^*} = 0 \quad (69)$$

$$v^* = 0 \quad (70)$$

GRID INDEPENDENCE

Mesh Generation

One important factor for optimal performance of turbulence models is the proper resolution of the boundary layer. In order to obtain a proper resolution, a good quality mesh should be applied. Meshing is the process in which the entire domain is subdivided into a number of smaller, non-overlapping subdomains in order to solve the flow physics within the domain geometry that has been created. This results in the generation of a mesh or grid of cells (control volumes) overlaying the entire computational domain. The essential fluid flows that are described in each of these cells are solved numerically so that the discrete values of the flow properties such as the velocity, pressure, temperature and other transport parameters which are of interest are determined.

The accuracy of a solution is governed by the number of cells in the mesh within the computational domain. The higher the number of cells, denser is the mesh. There by accurate results are obtained. However a large number of cells are accompanied by computational costs and a huge calculation turnover time. Thus considerable efforts are made to build a grid that produce accurate results with optimum computational cost.

Mesh Optimization

The meshing element used for this study is the tetrahedral element shown in Figure 18. The domain axes are x_1 , x_2 and x_3 . The nodes in the element are also shown in the same figure. Grid generation for this particular study started with a very coarse mesh all over the domain.

Figure 18. Tetrahedral element incorporated for meshing

This technique helped in running multiple test runs at a faster rate, thus being able to analyze the convergence or divergence behavior of the numerical code. After the initial test runs, the coarse mesh was replaced with a finer mesh. The finer mesh was kept non-uniform throughout the domain. Denser mesh was used near the wall regions, due to the fact that the model needs to account for high gradients existing which are prominent in the wall region. The meshing procedure is done in two steps, first the elements are

created in the surface and then the elements are projected across the volume. The tetrahedral elements of a mesh are hard to visualize. Hence, the mesh pattern in the surface is shown in this study. The non-uniform mesh was applied using two methods, namely:

- Edge spacing
- Face spacing

Edge Spacing

Edge spacing helps in specifying a mesh length scale on an edge and in the volume adjacent to an edge. Edge spacing option allows one to specify a minimum length scale and an expansion factor from that particular edge. This allows the elements to expand until the spacing reaches the default length scale of the domain or up to a certain radius as specified by the user. Figure 19 shows this feature.

Figure 19. Mesh development using edge spacing

Face Spacing

Similar to edge spacing, this feature allows one to expand the length scale at a desired rate to whatever radius of influence from a minimum length scale. Figure 20 shows an example of face spacing.

Figure 20. Mesh development using face spacing.

These two features are applied to very intricate regions where high mesh density is required. This feature is also applied on the edges next to the tubes where high gradients exist. These features allow having a non-uniform mesh on the domain, so that the total number of meshes in the grid can be controlled in a better manner. This also helps in proper usage of computer resources.

After the meshing was performed, the solver module was made to adapt the mesh as required for better and faster convergence after a certain number of iterations. The entire

grid was optimized based on the results of two important parameters. These parameters are:

- Temperature at the outlet
- Velocity at the outlet

Once the flow becomes a fully developed flow, at the outlet, the velocity and temperature profiles are taken to analyze the grid independence, so that the optimum mesh can be chosen with reasonable computational effort without sacrificing the accuracy of the results. The geometrical dimensions used for the mesh optimization is shown in Table 1.

Table 1. Geometrical parameters adopted for mesh optimization.

	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D (mm)	Wa (Degrees)	Wh (mm)
Plain Fin Staggered Configuration	22	25.4	3.00	0.13	9.5	-	-
Wavy Fin Staggered Configuration	19.05	25.4	3.53	0.12	9.525	17.5	1.5

Different grid resolutions are used for this study at $Re_H = 2000$. It is assumed to be valid for all other values of Reynolds number. The grid resolutions which are used are tabulated in Table 2.

Table 2. Different grid resolutions adopted for mesh optimization for the plain fin staggered configuration.

Plain Fin Staggered Configuration	Reynolds Number (Re_H)	Number of nodes	Number of tetrahedral elements
Mesh type A	2000	18,945	93,205
Mesh type B	2000	26,388	126,888
Mesh type C	2000	52,290	260,965

The plot obtained for the temperature distribution is shown in Figure 21. The horizontal axis shows the variation of length in Z-direction of the computational domain. The vertical axis shows the temperature. The plot is taken at the outlet location.

Figure 21. Temperature profiles at the outlet from wall plane to symmetry plane for different mesh configurations.

The maximum temperature difference between the mesh type B and the mesh type C is under 1%, which is sufficient enough to adopt mesh type B as the optimum one. Another variable which was selected for the study is velocity. The location chosen is again the outlet across the Z-axis of the computational domain. The velocity profile is shown in Figure 22.

Figure 22. Velocity profiles at the outlet from wall plane to symmetry plane for different mesh configurations.

Once the mesh was optimized for the plain fin configuration, a similar approach was done for the wavy fin staggered configuration. The approach was different due to a higher number of planes existing in the wavy fin staggered configurations. The corrugations existing in the wavy pattern which created these numerous planes on the surface should be properly meshed in order to obtain reasonable results. Hence, the number of nodes and tetrahedral elements applied for wavy fin staggered configuration is higher than that of the plain fin staggered configuration. Once the optimum mesh was chosen for the staggered configuration, it is considered good enough for the inline configuration as it contains lesser number of planes to address.

Table 3. Different grid resolutions adopted for mesh optimization for the wavy fin staggered configuration

Plain Fin Staggered Configuration	Reynolds Number (Re_H)	Number of nodes	Number of tetrahedral elements
Mesh type A	2000	20,945	100,703
Mesh type B	2000	35,423	185,645
Mesh type C	2000	59,290	306,965

Table 3 shows the grid resolutions adopted for the wavy fin staggered configuration for the mesh optimization study. Similar to the plain fin staggered configuration, the velocity and temperature were the variables chosen. The plot showing the variation of temperature at the outlet from the wall plane to symmetry plane is given in Figure 23. The plot showing the variation of velocity at the outlet from the wall plane to the symmetry place is given in Figure 24.

Figure 23. Temperature profile for wavy fin staggered configuration for grid independence.

Figure 24. Velocity profile for wavy fin staggered configuration for grid independence.

From the mesh optimization study, it can be concluded that the mesh type B is chosen for further study. This optimum mesh enables to perform the investigation with minimum computational resources without sacrificing the accuracy required for the study.

Fin Analysis

In the present study, the boundary condition for the fin is taken as no-slip wall boundary condition with a constant temperature (333K). This approximation may not be true in actual working conditions due to the conduction resistance in the fin. There is a temperature gradient existing from the base of the fin next to the tube to the fin tip which is farthest from the tube which is the coldest location in the fin. However, the constant temperature wall boundary condition for the fin can be a reasonable approximation. The close comparisons of numerical results with experimental results shown in the code validation given in the next chapter can be taken as required verification. In addition a fin analysis is performed to find out the temperature distribution, so as to further validate the constant temperature approximation which is applied on the fins as a boundary condition.

In this current fin analysis is performed considering the fin as an annular fin. The temperature for the base of the fin is the temperature of the tube surface which is assumed as a constant temperature (333K) in the current study.

An extreme case is considered for the fin analysis. A case with highest geometrical process parameters ensured that the constant temperature fin approximation would be effective for lower values of the geometrical process parameters as well. For the analysis, the Reynolds number is set at $Re_H = 7000$. This Reynolds number presents the most extreme scenario. The geometrical parameters selected are shown in the Table 4.

Table 4. Geometrical parameters used for fin analysis for constant temperature approximation

Annular fin analysis	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D1 (mm)	D2 (mm)
Plain Fin Staggered Configuration	25.0	25.4	3.0	0.13	9.5	9.5

Table 5. Physical properties and other paramaters adopted for fin analysis

Annular fin analysis	Reynolds Number Re_H	Thermal Conductivity of Al (W/m·K)	Temperature at the base (K)	Bulk Temperature (K)
Plain Fin Staggered Configuration	7000	235	333	303

The temperature distribution for an annular fin can be expressed in its general form as follows [48]:

$$\frac{d^2T}{dr^2} + \frac{1}{r} \frac{dT}{dr} - \frac{2h}{\lambda t} (T - T_{\infty}) = 0 \quad (71)$$

where r is the radial distance from the base of the fin, h is the heat transfer coefficient, λ is the thermal conductivity of the fin, Ft is the fin thickness and T_{∞} is the surrounding temperature.

Defining, $m^2 = \frac{2h}{\lambda t}$ and

$$\theta = (T - T_{\infty}) \quad (72)$$

Eq. (72) can be written as

$$\frac{d^2\theta}{dr^2} + \frac{1}{r} \frac{d\theta}{dr} - m^2\theta = 0 \quad (73)$$

Eq. (73) is a modified Bessel equation of order zero, and its general solution is of the form,

$$\theta(r) = C_1 I_0(mr) + C_2 K_0(mr) \quad (74)$$

where I_0 and K_0 are modified, zero-order Bessel functions of the first and second kinds, respectively. With the temperature at the base of the fin being known, C_1 and C_2 can be evaluated to yield a temperature distribution of the form,

$$\frac{\theta}{\theta_b} = \frac{I_0(mr)K_1(mr_2) + K_0(mr)I_1(mr_2)}{I_0(mr_1)K_1(mr_2) + K_0(mr_1)I_1(mr_2)} \quad (75)$$

Based on Equation, the temperature distribution for the annular fin case is evaluated. The temperature distribution for this extreme case considered is shown in Figure 25.

Figure 25. Temperature distribution from fin base to fin tip

Figure 25 shows that the temperature decreases from the base of the fin to the fin tip. The fin base temperature is 333 K. The temperature at the fin tip as evaluated from the Eq (75) was found to be 331.48 K. This analysis shows that the temperature difference between fin base and fin tip is 1.52 K. The inlet air temperature in the present investigation is set to be 303 K and the fin is set to be at a constant temperature (333 K) at the no-slip wall boundary. The maximum temperature difference (1.52 K) found between fin base and the fin tip can be considered negligible when compared with the temperature difference between inlet air flow (303 K) and the fin base (333 K). Furthermore, the annular fin case considered here is an extreme case of the present study. An extreme case with the highest transverse pitch ($Pt = 25.4$ mm) is used. Therefore, for all the other cases investigated in the present study, the temperature gradient between fin

base and fin tip would be even smaller. Hence, the constant temperature approximation is a valid approximation.

CODE VALIDATION

In order to verify the accuracy of this numerical investigation, the results were compared with the published experimental data by Wang et al. [3] for plain fin staggered configuration and with another paper by Wang et al. [4] for wavy fin staggered configuration. Once the numerical code was validated, the study was then extended to the configurations under parametric study in which there are no previous data available. The configurations were extended in such a way as to study the effect of variation of certain parameters which will be further explained in the results section.

The code validation of the numerical investigation was performed by comparing the pressure drop characteristics in the form of friction factor (f) and also comparing the heat transfer characteristics using the dimensionless term Colburn factor (j) from the sources mentioned above. The results were compared for the complete range of Reynolds numbers in which this study was conducted, that is $2000 \leq Re_H \leq 7000$. The geometrical parameters used for the code validation were obtained from the published results. These geometrical parameters are shown in Table 6.

Table 6. Geometrical parameters for the fin configurations for the code validation

	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D (mm)	Wa (Degrees)	Wh (mm)
Plain Fin Staggered Configuration	22	25.4	3.00	0.13	9.5	-	-
Wavy Fin Staggered Configuration	19.05	25.4	3.53	0.12	9.525	17.5	1.5

The turbulence model that was chosen for the investigation gave a very close match between the numerical results and experimental data. The following sections shows the comparison.

Plain Fin Staggered Configuration

Figure 26 and Figure 27 shows the comparison between the computed numerical values for the friction factor (f) and Colburn factor respectively with the published experimental data of Wang et al. [3] for plain fin staggered configuration. As mentioned, the comparison shows reasonable closeness between the two values. The maximum difference between the experimental data and the numerical results were less than 7% in the case of friction factor. The maximum difference in the case of Colburn factor was less than 12%.

Figure 26. Comparison of friction factor (f) for the plain fin configuration to the experimental data by Wang et al. [3]

Figure 27. Comparison of Colburn factor (j) for plain fin staggered configuration with experimental data by Wang et al. [3]

Wavy Fin Staggered Configuration

Similar to the plain fin staggered configuration,, the numeical results obtained for the wavy fin staggered configuration were also compared with the published experimental data by Wang et al. [4]. The maximum diference between the values when friction factor (f) was compared is below 8% and the maximum difference between the values of Colburn factor(j) was below 13%. The plots showing the comparison for friction factor is shown in Figure 28. The plot for Colburn factor is shown in Figure 29.

Figure 28. Comparison of Friction factor (f) for wavy fin staggered configuration with experimental data by Wang et al. [4].

Figure 29. Comparison of Colburn factor (j) for wavy fin staggered configuration with experimental data by Wang [4].

RESULTS AND DISCUSSION

There exists a crucial difference between the modeling of the physical phenomena of laminar and turbulent flow. For the turbulent case, the appearance of turbulent eddies occurs over a wide range of length scales. The $k-\omega$ model used in the current investigation to solve the turbulent problems does a commendable job by accommodating various turbulent quantities like the rate of dissipation of turbulent energy and turbulent viscosity. This model is widely used and is a tested turbulence model. Recently Amin and Panse [34] tested various turbulent models and its effectiveness in the transitional regime for similar geometries. The model's performance has been assessed against number of practical flows. It has achieved notable success in predicting thin shear layers, boundary layers without the need for case by case model adjustment [36].

The results mentioned in this chapter are obtained from the simulations run using ANSYS CFX, which is a software available for solving engineering problems. This software is developed by ANSYS [38]. In the post-processing module of the software, there is a declarative language that has been developed based on Fortran. It is different from an imperative language as it reads from the user 'what is to be done' rather than 'how it is to be done'. It can be used to create new expressions and create new constants which may be dimensional or dimensionless. It can also be used to add terms to the solved equations. This language, which is known as CFX expression language (CEL), has been used extensively in this study.

The post processor module also allows having various visualizations. The most prominent one used in this study are the two dimensional graphs that represent the

variation of one dependent transport variable against another independent variable. These plots are precise way to present quantitative numerical data. Another useful visualization method is to have vector plots. These plots can be used to display vector quantity at discrete locations whose orientation indicates the direction and its length or size is a measure of its magnitude. Vector plots are used mostly for displaying velocity in this study.

Another way of displaying results which is used in this study is by using contour plots. It is one of the most common ways for graphical representation of data. A contour line is a line in space where the line is a representation of some property which is constant throughout that line in the space. Similar to lines, there are contour surfaces. It is usually linearly scaled; it helps to display the variation of a particular variable over the flow field. In this study temperature is represented using this method.

One unique tool in the post-processor is to display the streamlines; it helps in understanding the nature of flow in two as well as three dimensions. By definition, streamlines are parallel to mean velocity vectors, where they trace the flow pattern using massless particles. The streamlines can be used to emphasize recirculation zones existing in the flow field. Few of these plots are presented in the parametric study.

Flow Distinction

The range of flow in the current investigation is in turbulent range. In order to appreciate the turbulent flow, there is a need to understand the difference between the flow in turbulent range and the laminar range. The Figure 30 shows the general trend of Colburn factor and friction factor put forth by Kakac et al. [49]. Though the diagram shows a general trend in a circular pipe, the same trend is observed in this study as well. From the plot it can be seen that the slope of the two parameters generally tend to reduce as the Reynolds number is increased. Both the friction factor and Colburn factor tend to be almost parallel to each other as seen in the figure.

Figure 30. The general trend of friction factor(f) and Colburn factor(j) against Reynolds number for flow in a circular pipes as mentioned by Kakac et al. [49]

The results reported by Jang, Amin and Panse [30] and [34] explore flow characteristics in the laminar and transition range. Those results that are available in the turbulent range given by Amano [50] and Wang et al. [4] does not incorporate the effect of various geometrical parameters which are essential in design and development. This study explores few of those particular parameters in order to aid designers to choose better geometry depending upon their requirements and constraints.

Flow Distinction Between Laminar and Turbulent Flow

This section discusses the differences in flow field as the Reynolds number is increased from laminar to the turbulent regime. The major impact of increasing Reynolds number is that the flow rapidly changes its course and becomes turbulent. In the case of laminar flow, the flow adjusts itself to the geometry, where as in the case of the turbulent flow, the flow becomes violent and a large amount of eddy dissipation takes place. Figure 31 and Figure 32 shows the trend of friction factor and Colburn factor for the laminar and transitional regime ($1000 \leq Re_H \leq 2000$).

Figure 31. Friction factor for plain and wavy fin staggered configuration for laminar and transitional regime

Figure 32. Colburn factor for plain and wavy fin staggered configuration for the laminar and transitional regime

The following discussion explains the flow dynamics with the increase in Reynolds number with the help of streamline plots and few isothermal contour plots. Faber [51] mentions that the curvature of a streamline is related to the pressure gradient acting perpendicular to the streamline. The radius of curvature of the streamline is in the direction of decreasing radial pressure. In other words, a streamline which is straight will have zero radial pressure. Also, a streamline having a circle with small radius will have a higher radial pressure than a circle with a higher radius.

Figure 33, Figure 34, Figure 35 and Figure 36 show the streamlines on a plain fin staggered configuration for $2000 \leq Re \leq 7000$. The location of the streamlines given is the symmetry plane. It is to be noticed that the velocity scales in the figures are different.

The flow field with highest Reynolds number shows high amount of recirculation. As the velocity of the fluid increases, it definitely aids better mixing but it does not absorb enough heat resulting in low Colburn factor which will be discussed in later sections. In all the cases the recirculations are found behind the tube. Maximum velocity is found where the fluid crosses a tube.

Figure 33. Streamlines on plain fin staggered configuration for Re=2000

Figure 34. Streamlines on plain fin staggered configuration for Re=3000

Figure 35. Streamlines on plain fin staggered configuration for $Re=5000$

Figure 36. Streamlines on plain fin staggered configuration for $Re=7000$

Figure 37. Temperature contour plot for plain fin staggered configuration for Reynolds number $Re=2000$

Figure 38. Temperature contour plot for plain fin staggered configuration for Reynolds number $Re=7000$

Figure 39. Turbulent viscosity for the case of wavy fin staggered configuration at $Re=2000$.

Figure 37 and Figure 38 show the contour plots of temperature for different Reynolds numbers. In high Reynolds number test cases, the fluid experiences higher temperature earlier in the flow field than for the cases with lower Reynolds numbers. This can be attributed to better flow mixing and higher recirculations existing. Figure 39 shows the variation of turbulent viscosity in the wavy fin staggered configuration.

Turbulent viscosity is higher where the shear forces are more dominant like near the pockets and edges existing in the domain.

The three flow variables, viz. temperature, velocity and turbulent viscosity can be used to explain the flow physics existing. The figures are given in Figure 40 Figure 41 and Figure 42. It can be observed that the velocity tends to be low in those regions where turbulent viscosity is higher due to existence of recirculations. The higher shear stresses existing in those regions bring down the velocity. As the velocity is lower, the fluid absorbs more heat from the walls as it is in contact with the wall for a longer time period. The huge temperature gradient existing in the domain on the far end (outlet side) of the tubes is due to the influx of fresh air flowing at higher velocities and the air moving away at lower velocities from the recirculation regions. This is shown in Figure 41. The high shear stresses existing in those regions also elevates the viscosity as shown in Figure 42.

Figure 40. Velocity contour lines for plain fin staggered configuration at Reynolds number, $Re=7000$.

Figure 41. Temperature contour lines for plain fin staggered configuration at Reynolds number, $Re=7000$.

Figure 42. Turbulent viscosity contour lines for plain fin staggered configuration at Reynolds number $Re=7000$.

Difference in Flow Pattern between Inline and Staggered Configuration

There are mainly two ways to arrange the tubes in a heat exchanger, either in an inline configuration or in a staggered configuration. The flow pattern is different in both the cases as the course for the flow is different. Figure 43 and Figure 44 show the inline and staggered arrangement of plain fin heat exchanger. Figure 45 and Figure 46 show the inline and staggered configuration for wavy fin respectively.

Figure 43. Computational domain for plain fin inline configuration with four tubes

Figure 44. Computational domain for plain fin staggered configuration with four tubes

Figure 45. Computational domain for wavy fin inline configuration with four tubes

Figure 46. Computational domain for wavy fin staggered configuration with four tubes

As expected, the staggered arrangement allows better mixing of flow due to obstacles. These obstacles in the form of tubes help in reorienting the flow. Figure 47 shows the streamlines for a plain fin inline configuration at $Re=2000$. The flow mixing is more prominent in the case of staggered configuration as shown in Figure 48 due to the presence of obstacles in the flow path. The temperature absorption by the fluid is also higher as the fluid comes in contact with hot tubes in a better manner.

Figure 47. Streamlines in plain fin inline configuration at Reynolds number, $Re=2000$.

In Figure 47 and Figure 48 it can be observed that recirculation zones with very low velocity exist in between the tubes in the case of inline configuration. It can be speculated that this might create stagnation even at high velocities of air flow. Whereas in

the case of staggered arrangement as shown in Figure 48, the flow gets mixed and reoriented at every stage because of the location of tubes which guides the air flow through the course. The location for Figure 47 and Figure 48 in the domain is the symmetry plane.

Figure 48. Streamlines in plain fin staggered configuratoom at Reynolds Number, $Re=2000$

Figure 49. Contour plot of temperature at symmetry plane for plain fin inline configuration.

Let us look into the temperature profile for these two arrangements at the same location. The contour plots shown in Figure 49 and Figure 50 depict the temperature profile at the symmetry plane for Reynolds number $Re=2000$. The heat absorption from the fin and the tubes are higher in the staggered configuration. The higher heat absorption

is clearly visible from the contour plots that the higher temperature bands appear much earlier in the domain in the staggered configuration than the inline configuration.

Figure 50. Contour plot of temperature at symmetry plane in plain fin staggered configuration

When the contour plots for plain fin configurations are compared with the wavy fin configurations, few interesting observations can be made. The contour plot for the wavy fin configuration is shown in Figure 51.

Figure 51. Contour plot of temperature at symmetry plane for wavy fin inline configuration

Lack of mixing of the fluids in the plain fin inline configuration as seen in Figure 47, does not occur in the case of wavy fin configuration. This is due to the fact that mixing occurs as a result of the wavy corrugations existing in the wavy pattern even in

inline configuration, as shown in the vector plot shown in Figure 52. The mixing is not very much hindered by the absence of staggered arrangement, the corrugations makes sure that some amount of mixing is taking place.

Figure 52. Vector plot for wavy fin inline configuration

The vector plot of plain fin staggered configuration and an enlarged view of the same plot are given in Figure 53 and Figure 54. The enlarged view is the view between the second and the third tube.

Figure 53. Vector plot for plain fin staggered configuration.

Figure 54. Enhanced vector plot for plain fin staggered configuration between second and third tube.

As one can see, the magnitude which is represented by the length of vectors, are higher where it crosses the tube. It is the lowest behind the tubes where it directly strikes the tube. At the same location, as the fluid strikes the tubes, it may create eddies and recirculate. As the fluid passes over the tubes, the reorientation can be viewed with the direction of vectors presented. It can be seen that the fluid particles are aligning to orient itself according to the domain geometry. The plots showing the trend of friction characteristics and the heat transfer characteristics comparing the inline and staggered configurations for both the plain wavy patterns are shown in Figure 55 and Figure 56.

Figure 55. Friction factor comparison between inline and staggered configuration for the plain fin pattern.

A trend mentioned in the analysis of contour and streamline plot is reflected in the graphs depicting the trend of friction factor and Colburn factor. Higher friction factor and Colburn factor is observed for the staggered configuration. This trend is aided by better flow mixing and higher pressure drop in the case of staggered configuration due to higher obstacles in the flow path. The tubes create boundary layer interruptions in the plain fin configurations. The high difference in the pressure drop and heat transfer characteristics make the tube layout an important area of consideration.

Figure 56. Colburn factor comparison between inline and staggered configuration for the plain fin pattern.

It was displayed in the plain fin configuration that higher friction factor and Colburn factor exists for staggered configuration than the inlined configuration. This trend is true for the wavy fin configuration as well. The wavy fin staggered configuration reports higher pressure drop and higher heat transfer characteristics than the inlined configuration. The results show that in the inline configuration, the pressure drop and thereby the friction factor is considerably lower than the the staggered configuration. The friction factor comparison is shown in Figure 57. Similarly a Colburn factor comparisn is shown in Figure 58.

Figure 57. Friction factor comparison between inline and staggered configuration for the wavy fin pattern.

Figure 58. Colburn factor comparison between inline and staggered configuration for the wavy fin pattern.

Effects of the Number of Tube Rows

In this section, the effect of number of tube rows in pressure drop characteristics and friction characteristics are being explored. The study is performed for both the plain fin staggered configuration and the wavy fin staggered configuration. The number of rows were varied from one to six for the study.

Plain Fin Staggered Configuration

As mentioned earlier, the study was performed for both plain and wavy fin staggered configuration. Plain fin staggered configuration is studied first. The geometrical parameters which were incorporated for this study are shown in Table 7.

Table 7. Geometrical Parameters for the plain fin configuration for the effect of tube row numbers study

	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D (mm)	Wa (Degrees)	Wh (mm)
Plain Fin Staggered Configuration	22	25.4	3.00	0.13	9.5	-	-

The studies shown by Tutar and Akkoca [32] point out that there is no significant difference in the pressure drop and friction characteristics once the number of tube rows was increased beyond four. This was also shown in the studies by Jang et al. [30]. In this study, the average heat transfer coefficient and coefficient of pressure are taken for comparison. The computational domain of the plain fin staggerd configuration having various number of tube rows are shown in the Figure 59 and Figure 60.

(a)

(b)

(c)

Figure 59. Computational domains used in plain fin staggered configuration to study the effects of tube rows using (a) one (b) two and (c) three tube rows

(d)

(e)

(f)

Figure 60. Computational domains used in plain fin staggered configuration to study the effects of tube rows using (d) four (e) five and (f) six tube rows

The heat transfer coefficient against the number of tube rows is shown in the Figure 61. There is an increase in the heat transfer coefficient as the number of tube rows are increased, this is due to increase in overall area of effectiveness. It can also be noted that the slope reduces as the number of tube rows are increased. However, with the

increase in the heat transfer and the number of tube rows, there is a subsequent increase in the pressure drop as shown in Figure 62. Figure 62 shows the pressure coefficient against the number of tube rows. The idea here is to create a balance between the pressure drop and heat transfer. Around four tubes, there seems to be a balance between the increase in pressure drop and the heat transfer coefficient. Also, four tubes is the number of tubes chosen for study by many scholars including Wang et al. [3] and Jang et al. [30].

Figure 61. Heat transfer coefficient against the number of tube rows for the plain fin staggered configuration.

Figure 62. Pressure coefficient against the number of tube rows for the plain fin staggered configuration.

Wavy Fin Staggered Configuration

Similar to the study on plain fin staggered configuration, the investigation was carried out for the wavy fin staggered configuration. The geometrical parameters of the study is shown in Table 8.

Table 8. Geometrical Parameters for the wavy-fin configuration for the effect of tube row numbers study

	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D (mm)	Wa (Degrees)	Wh (mm)
Wavy Fin Staggered Configuration	19.05	25.4	3.53	0.12	9.525	17.5	1.5

The computational domains for the study of tube rows for the wavy fin staggered configuration are shown in the Figure 64. In a similar manner to the plain fin staggered configuration, the average heat transfer coefficient and pressure coefficient are taken as the variables to be studied.

(a)

(b)

(c)

Figure 63. Computational domains used in wavy fin staggered configuration to study the effect of tube rows using (a) one (b) two (c) three tube rows

(d)

(e)

(f)

Figure 64. Computational domains used in wavy fin staggered configuration to study the effect of tube rows using (d) four (e) five and (f) six tube rows

The reason for an increase in average heat transfer coefficient can be attributed to the fact that, as the number of rows of tube increases, the amount of effective area that take part in the heat transfer also increases. Similar to plain fin staggered configuration, the heat transfer coefficient witnesses lesser increase with each tube row as the number

of tube rows are increased beyond three. Once this optimum number is reached, there seem to be a balance between the heat transfer rate and the increase in effective area for heat transfer. The pressure coefficient plot against the number of tube rows is shown in Figure 66. As seen before there is a subsequent increase in the pressure drop across the geometry as well with the increase in number of tube rows. The balance is found to be around where the number of tube rows are around four.

Figure 65. Heat transfer coefficient against number of tube rows for wavy fin staggered configuration.

Figure 66. Pressure coefficient against the number of tube rows for the wavy fin staggered configuration.

It can be said from this study that, the optimum number of tube rows is four. This is found true in both the plain fin and wavy fin configurations. The selection of four tube rows by various researchers in previous studies is also validated. Hence, the four tube row configuration is taken used for investigation carried out in the present study as well.

Effects of Transverse Pitch

Plain Fin Staggered Configuration: Once we have established the optimum nature of the four row configuration, we can explore other aspects of geometrical parameters. In this section, the effects of transverse pitch will be explored. For this study, three test cases are considered, with all the other parameters being kept constant. The other parameters need to be kept constant so as to isolate the effects of the parameter under study. This study is incorporated for both the plain fin staggered configuration and the wavy fin staggered configuration. Table 9 shows the geometrical parameters used to analyze the effects of transverse pitch in the plain fin staggered configuration. The effects of transverse pitch have not been studied numerically before, especially the effects of the transverse pitch in the turbulent region.

Table 9. Geometrical parameters for the plain-fin staggered configuration for the effects of transverse pitch (Pt)

Plain Fin Staggered Configuration	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D (mm)	Wa (Degrees)	Wh (mm)
Test Case 1	22	17.7	3.00	0.13	9.5	-	-
Test Case 2	22	21.0	3.00	0.13	9.5	-	-
Test Case 3	22	25.4	3.00	0.13	9.5	-	-

The effects are studied with the help of pressure drop characteristics represented by the friction factor (f) and heat transfer characteristics represented by Colburn factor (j). The plot showing the effects of friction characteristics is shown in Figure 67.

Figure 67. Effects of the friction factor for plain fin staggered configuration with change in transverse pitch(Pt)

It is evident from Figure 67 that the pressure drop is indeed affected by the value of transverse pitch. The trend indicates that the pressure drop reduces as the value of transverse pitch is increased. With the increase in transverse pitch, the flow mixing does not happen as much in the flow field. The increase in transverse pitch decreases the flow mixing and thereby the friction factor.

Figure 68. Effects of Colburn factor for plain fin staggered configuration with change in transverse pitch(P_t)

Few observations can be made from Figure 68. The trend indicates, there is a higher Colburn factor for low value of transverse pitch for each case of Reynolds number. Thus, the lowest value in the study, $P_t=17.7\text{mm}$, has the maximum Colburn factor value. This phenomenon can be attributed to the assumption that at low transverse pitch, the fluid tends to mix well and thus, has more absorption of heat from the walls. Another observation is that the difference in the value of Colburn factor for the cases of varied transverse pitch gets low as the Reynolds number goes up.

Wavy Fin Staggered Configuration

Similar to the study conducted on the plain fin staggered configuration, the study was extended to wavy fin staggered configuration. Like before, four tube rows were adopted as it is the optimum case as observed by Jang et al. [30] and Wang et al. [4] to name few. Other geometrical parameters are kept constant to study the effects of variation of the transverse pitch. Table 10 shows the geometrical parameters used for this study.

Table 10. Geometrical parameters for the wavy-fin staggered configuration to study the effects of transverse pitch (Pt)

Wavy Fin Staggered Configuration	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D (mm)	Wa (Degrees)	Wh (mm)
Test Case 1	19.05	17.7	3.53	0.12	9.525	17.5	1.5
Test Case 2	19.05	21.0	3.53	0.12	9.525	17.5	1.5
Test Case 3	19.05	25.4	3.53	0.12	9.525	17.5	1.5

To analyze the effects of the transverse pitch for the wavy fin configuration, the same approach of observing friction factor and Colburn factor were adopted. The Figure 69 shows the friction factor for three different test cases for the wavy fin staggered configuration. Figure 70 shows the variation of Colburn factor in the wavy fin staggered configuration as the transverse pitch is varied.

Figure 69. Effects of the friction factor for wavy fin staggered configuration with change in transverse pitch(Pt)

As can be seen from Figure 69, the effects of transverse pitch is minimal in the wavy fin staggered configuration. This might be due to the fact that the flow mixing is not greatly impacted by the transverse pitch in the wavy fin staggered configuration in contrast to the plain fin staggered configuration. However, lower value of transverse pitch showed the lowest friction factor.

Figure 70. Effects of the Colburn factor for wavy fin staggered configuration with change in transverse pitch(Pt)

In the case of Colburn factor for the wavy fin staggered configuration, lower the transverse pitch, higher the value of Colburn factor. This phenomenon can again be attributed to the same reason as in the case of plain fin staggered configuration. As there is an increase in transverse pitch, the flow mixing gets reduced. The test case of $P_t=17.7\text{mm}$ showed the maximum heat transfer.

Effects of Fin Pitch

In this section, the focus is on the influence of the distance between the fins on the heat transfer and friction characteristics of the fluid flow. The effect of fin spacing can be very critical. As it determines the number of fins which can be installed in a given space along the tubes. The effects were studied for three different cases with varying fin pitch. The fin pitch used in all the calculations is defined according to the Equation (76) and its relation to the overall setup is demonstrated in the Figure 71.

$$\text{Fin spacing} = [\text{Fin pitch} - \text{Fin thickness}] \quad (76)$$

Figure 71. Fin spacing for plain fin configuration

The effects are studied on plain fin staggered configuration first. Table 11 shows the geometrical parameters used for the study.

Table 11. Geometrical parameters for the plain fin staggered configuration to study the effects of fin pitch (Fp)

Plain Fin Staggered Configuration	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D (mm)	Wa (Degrees)	Wh (mm)
Test Case 1	22	25.4	3.00	0.13	9.5	-	-
Test Case 2	22	25.4	2.23	0.13	9.5	-	-
Test Case 3	22	25.4	1.70	0.13	9.5	-	-

The influence on friction factor is shown in the Figure 72. There is a significant effect of fin pitch on the friction factor as shown in the Figure 72. The fin spacing strongly influences the overall pressure drop and thereby the friction factor. The inference is that if the fin spacing is too small, the pressure drop tends to get higher and thereby the friction factor also increases. If the fin spacing is too large, the pressure drop gets lowered. However, having a high fin spacing will influence the number of fins a heat exchanger can have.

Figure 72. Effects of the friction factor for plain fin staggered configuration with change in fin pitch(Fp)

For small fin spacing, the flow is termed as Hele-Shaw type [52]. As the spacing increases, a separation zone gets formed behind the tubes and this happens especially at high Reynolds number.

The effect of fin pitch on Colburn factor is shown in Figure 73. A higher heat transfer is observed for lower value of in pitch. The effect subsides as the Reynolds number goes higher. Higher heat transfer at low fin spacing is due to the fact that the fluid comes in more intimate contact with the fins.

Figure 73. Effects of the Colburn factor for plain fin staggered configuration with change in fin pitch(Fp)

Multiple observations made in this study about the effects of fin pitch. Higher heat transfer is associated with a higher pressure drop. Let us take a look at the efficiency index of this particular study. Efficiency index is the ratio of Colburn factor to friction factor. It is one of the terms used to determine the efficiency of a heat exchanger. The plot of efficiency index against the Reynolds number for various values of fin pitch is given in Figure 74. The disadvantage of having higher pressure drop with decreasing value of fin spacing is not compensated with very high value of heat transfer characteristics as evident from the plot of efficiency index. So, a lower fin spacing for a marginal increase in the heat transfer will result in a very high pressure drop. The test

case with maximum fin spacing performed better with high efficiency index. The efficiency index plot also points that the efficiency decreases as the Reynolds number is increased.

Figure 74. Efficiency index for plain fin staggered configuration with change in fin pitch(F_p)

Let us take a look at the effects the fin pitch has on the wavy fin staggered configuration. The geometrical parameters are different than the plain fin staggered configuration. The geometrical parameters chosen are shown in Table 12.

Table 12. Geometrical parameters for the wavy fin staggered configuration to study the effects of fin pitch (Fp)

Wavy Fin Staggered Configuration	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D (mm)	Wa (Degrees)	Wh (mm)
Test Case 1	19.05	25.4	3.53	0.12	9.525	17.5	1.5
Test Case 2	19.05	25.4	2.34	0.12	9.525	17.5	1.5
Test Case 3	19.05	25.4	1.69	0.12	9.525	17.5	1.5

Similar to the trend observed in the plain fin staggered configuration, the wavy fin staggered configuration also encounters a similar effect on the pressure drop with decreasing value of fin pitch. The pressure drop getting higher can again be attributed to the higher amount of contact by fluid with the fin surface. The reason may be that the flow becomes more viscous in nature and the friction forces get higher. The trend of friction factor is shown in Figure 75.

Figure 75. Effects of the friction factor for wavy fin staggered configuration with change in fin pitch(F_p)

As shown in Figure 76, the friction factor increases with decreasing value of fin pitch. Therefore, there is a higher increase of pressure drop in the case of wavy fin staggered configuration than plain fin staggered configuration as the fin pitch is reduced.

The increase in heat transfer characteristics is not very high as the fin pitch is decreased. The Colburn factor has increased slightly with the decrease in fin pitch as shown in Figure 76. The impact of fin pitch cannot be cited just on the basis of Colburn factor. The efficiency index for wavy fin staggered configuration needs to be considered just like the case of plain fin staggered configuration.

Figure 76 Effects of the Colburn factor for wavy fin staggered configuration with change in fin pitch(F_p)

The effectiveness can only be found out with the help of a plot depicting efficiency index. The efficiency index for the wavy fin staggered configuration when the fin pitch is varied is shown in Figure 77. The effect of pressure drop and the increase in Colburn factor balances out for the values in between 2.34mm and 1.69mm. When the fin spacing goes above the value of 2.34mm, the pressure drop tends to dominate and thereby reduces the efficiency index of the heat exchanger.

Figure 77. Efficiency index for wavy fin staggered configuration for change in fin pitch(F_p).

Effects of Wavy Angle

In this section, the study focuses on the effect of wavy angle or the corrugation angle existing in the wavy fin staggered configuration. The three different test cases which are analysed involves three different angles and these are shown in Figure 78, Figure 79 and Figure 80. The geometrical parameters selected for this study are shown in Table 13. Since, the wavy angles occur in wavy fin staggered configuration, the plain fin staggered configuration will not be discussed.

Table 13. Geometrical parameters for the wavy fin staggered configuration to study the effects of wavy angle (Wa)

Wavy Fin Staggered Configuration	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D (mm)	Wa (Degrees)	Wh (mm)
Test Case 1	19.05	25.4	3.53	0.12	9.525	8.95	1.5
Test Case 2	19.05	25.4	2.34	0.12	9.525	17.5	1.5
Test Case 3	19.05	25.4	1.69	0.12	9.525	32.21	1.5

Figure 78. Wavy fin staggered configuration with the wavy angle ($Wa=17.5^\circ$)

Figure 79. Wavy fin staggered configuration with the wavy angle ($Wa=8.95^\circ$)

Figure 80. Wavy fin staggered configuration with the wavy angle ($Wa=32.21^\circ$)

As the wavy angle is increased, the number of corrugations for a fixed length also increases, thus increasing the total distance of flow. This has a major impact on the heat transfer and friction characteristics which is explained in this section. The pressure drop

characteristics are shown in terms of friction factor in Figure 81 and the heat transfer characteristics are shown in Figure 82.

Figure 81. Effects of the friction factor for wavy fin staggered configuration with change in wavy angle

Figure 82. Effects of the Colburn factor for wavy fin staggered configuration with change in wavy angle

The plots reveal that, as the wavy angle is increased, and as the number of corrugations increase, there is a simultaneous surge in the friction factor and the Colburn factor. The higher friction factor can be attributed to the increase in flow length and thereby a larger pressure drop. The higher Colburn factor is due to the increased amount of corrugations existing in the flow field. The lower angle see a simultaneous decrease in Colburn factor and friction factor due to the same above mentioned reasons. As the corrugations becomes less prominent for the case $Wa=8.95^\circ$, the values of Colburn factor and the friction factor is comparable to that of a plain fin staggered configuration in the case where the angle is reduced. The number of corrugations getting reduced can be cited as the reason for the close matching of values between a low wavy angle wavy fin

staggered configuration and plain fin staggered configuration. The efficiency index shown in Figure 83 gives a clear picture of the effectiveness when the wavy angle is varied. The friction factor given for the higher angle is compensated by the high value of Colburn factor with which it is associated. Thereby 32.21° angle has the highest efficiency from the study.

Figure 83. Efficiency index for the wavy fin staggered configuration for change in wavy angle(W_a)

Effects of Wavy Height

This section brings the focus on the effect of wavy height on the pressure drop and heat transfer characteristics. The geometrical parameters chosen for the investigation is shown in Figure 84, Figure 85 and Figure 86 shows the computational domain of three different test cases for the parametric study of wavy height.

Table 14 shows the computational domain of the three different test cases chosen with corresponding wavy height.

Table 14. Geometrical parameters for the wavy fin staggered configuration to study the effects of wavy height (Wh)

Wavy Fin Staggered Configuration	Pl (mm)	Pt (mm)	Fp (mm)	Ft (mm)	D (mm)	Wa (Degrees)	Wh (mm)
Test Case 1	19.05	25.4	3.53	0.12	9.525	17.5	0.7508
Test Case 2	19.05	25.4	2.34	0.12	9.525	17.5	1.5
Test Case 3	19.05	25.4	1.69	0.12	9.525	17.5	3.0032

Figure 84. Wavy fin staggered configuration with the wavy height (Wh=0.7508mm)

Figure 85. Wavy fin staggered configuration with the wavy height ($Wh=1.5\text{mm}$)

Figure 86. Wavy fin staggered configuration with the wavy height ($Wh=3.0032\text{mm}$)

Similar to the effect seen in the case of wavy angle, the wavy height also changes the flow length and the number of corrugations. The three different test cases show a similar trend which was observed in the case of variation in wavy angle. Figure 87 shows the friction factor characteristics and Figure 88 shows the Colburn factor for the wavy fin staggered configuration for the variation in wavy height.

Figure 87. Effects of the friction factor for wavy fin staggered configuration with change in wavy height

The friction factor is directly related to the increase in wavy height due to the increase in pressure drop which is further caused by the increase in flow length. Similar to the wavy angle (Wa) test case, the test case with the lower value of wavy height behaves more like the plain fin staggered configuration due to lesser corrugations and very small increase in the flow length. The same effect is seen in the case of Colburn factor also, where the increase in wavy height resulted in more corrugations and thereby high mixing in flow. Also the fluid is in more intimate contact with the fin surface resulting in higher heat transfer and subsequently higher Colburn factor.

Figure 88. Effects of the Colburn factor for wavy fin staggered configuration with change in wavy height

The efficiency index for the wavy fin staggered configuration for change in wavy height is shown in Figure 89. It can be noted that the efficiency index increases with the increase in wavy height.

The wavy angle and wavy height are similar in one approach. As the wavy angle or wavy height is increased, the number of corrugations also increases. However, the reduction of wavy angle and wavy height will smoothen the flow field due to a reduction in the corrugations. Hence, those cases where the wavy angle and wavy height are low behave more like plain fin case.

Figure 89. Efficiency index for the wavy fin staggered configuration for change in wavy height(Wh)

CONCLUSIONS

The conclusions obtained during the three-dimensional numerical investigation which were carried out on different configurations of plate fin and tube heat exchangers are presented. The study concentrated on the hydraulic and thermal characteristics of air side in the heat exchanger. The study focussed on the following range of $2000 \leq Re \leq 7000$ which falls in the turbulent regime of flow. The study was performed using the commercial code ANSYS CFX. The grid developed to solve the problem numerically was tested for grid independence, an optimized mesh was thus chosen for the analysis. The code was then validated by comparing the numerical results with published experimental data.

The configurations which were under study includes the plain fin and wavy fin heat exchangers, two different layouts of arrangement of tubes namely inline and staggered for both plain fin and wavy fin were then analysed. The two different arrangement were the inline arrangement and the staggered arrangement. The number of tube rows were varied for both the fin configurations to study the effects and find an optimum number of tube rows. For the staggered arrangement, the geometrical parameters of transverse pitch, longitudinal pitch and the fin pitch were varied for both plain fin and wavy fin configurations. Furthermore, the wavy angle and wavy height of the wavy fin configuration were varied. During the parametric study, only one parameter was varied, keeping the remaining constant. The thermal and hydraulic characteristics in the form of heat transfer and pressure drop were studied using the dimensionless coefficients namely, Colburn factor and friction factor.

The analysis in the turbulent regime was performed using the $k-\omega$ turbulence model which is widely used. The analysis was performed for seven test cases in the turbulent regime for each individual analysis. The following conclusions are made from the current study:

1. It was found that the effects of tube arrangement is very critical, as it influences both the pressure drop and heat transfer to a large extent. Staggered layout improves the heat transfer to a very large extent prompting to conclude that the staggered arrangement is indeed better than the in-lined arrangement. This effect was evident in both the plain fin configuration and the wavy fin configuration.
2. The wavy fin configuration showed consistent increase of pressure drop and heat transfer in both the in-lined and staggered layout compared to its counterpart, the plain fin configuration.
3. The increase in number of tube rows shows that the average heat transfer coefficient and pressure coefficient gets increased too. The rate of increase of heat transfer coefficient slowed down as the number of tube rows go beyond the optimum number of the tube rows which was found to be four. Beyond this critical value, the effective heat transfer area strikes a balance with the corresponding pressure drop associated.
4. The effect of increase in transverse pitch on the friction factor and Colburn factor on both types of configuration is that the friction factor gets reduced as the transverse pitch is increased. In the case of plain fin staggered configuration the

effect of variation of transverse pitch is more prominent than the wavy fin staggered configuration.

5. From the study, an increase in friction factor was found when the fin pitch was reduced in both plain fin staggered configuration and the wavy fin staggered configuration. However, the variation is higher in case of the plain fin staggered configuration. The Colburn factor increased as the fin pitch was reduced for both the configurations. Lowering of fin pitch beyond a critical value resulted in lower efficiency index for both wavy and plain fin staggered configuration.
6. The variation in wavy angle was studied for wavy fin staggered configuration. Higher wavy angle resulted in higher friction factor and a higher Colburn factor. This was due to the higher number of corrugations and better flow mixing aspect associated with it. The effective flow distance also got increased with higher wavy angle. The plots of efficiency index showed that the higher wavy angle has higher efficiency.
7. Similar to the study of wavy angle, it was found that higher the wavy height, more the number of corrugations in the flow path and subsequently a longer flow path. The friction factor and the Colburn factor was found to have increased with high values of wavy height. The efficiency index plot shows that higher efficiency is obtained as the wavy height is increased.

REFERENCES

1. Webb.R.L. Enhancement of single phase convective heat transfer. *Handbook of single phase convective heat transfer*. New York : John Wiley, 1987.
2. *Investigation of wavy fin-and-tube heat exchangers: A contribution to databank*. Wang, C. C., Lin, Y. T., Lee, C. J. and Chang, Y. J. 1999, Experimental Heat Transfer, Vol. 12, Number 1, pp. 73-89.
3. *Sensible heat and friction characteristics of plate fin-and-tube heat exchangers having plane fins*. Wang, C. C., Chang, Y. J., Hsieh, Y. C. and Lin, Y. T. 1996, International Journal of Refrigeration, Vol. 19, Number 4, pp. 223-230.
4. *Heat transfer and friction characteristics of typical wavy fin-and-tube heat exchangers*. Wang, C. C., Fu, W. L. and Chang, C. T. 1997, Experimental Thermal and Fluid Science, Vol. 14, Number 2, pp. 174-186.
5. *Effect of fin spacing on the heat transfer and friction performance of multi-row, smooth plate fin-and-tube heat exchangers*. Rich, D.G. 1973, ASHRAE Transactions, Vol. 79, Number 2, pp. 137-145.
6. *Effect of the number of tube rows on heat transfer performance of smooth plate fin-and-tube heat exchangers*. Rich, D.G. 1975, ASHRAE Transactions, Vol. 81, Number 1, pp. 307-319.
7. *Heat-transfer and flow- friction data for two fin- tube surfaces*. McQuiston, F. C. and Tree, D. R. 1971, Journal of Heat Transfer, ASME Transactions, Vol. 93, Number 2, pp. 249-250.
8. *Correlations for heat, mass and momentum transport coefficients for plate-fin-tube heat transfer surfaces with staggered tub*. McQuiston, F C. 1978, ASHRAE Transactions, pp. 294-309.
9. *Performance of one- and two-row tube and plate fin heat exchangers*. Rosman, E. C., Carajilescov, P. and Saboya, F. E. M. 1984, Journal of Heat Transfer, Transactions ASME, Vol. 106, Number 3, pp. 627-632.
10. *Transient testing technique for heat exchanger fin surfaceS*. Yoshii, T. 1972, Journal of the Institution of Telecommunication Engineers, Vol. 1, Number 3, pp. 51-58.

11. *Effects of dropwise condensate on wet surface heat transfer for air cooling oils.* Yoshii, T., Yamamoto, M. and Otaki, T. 1973, Proceedings of 13th International Congress of Refrigeration, pp. 285-292.
12. *Effects of fin pattern on the air side heat transfer coefficient in plate finned-tube heat exchangers.* Beecher, D. T. and Fagan, T. J. 1987, ASHRAE Transactions, pp. 1961-1984.
13. *Effect of rounding of protruding edges on heat transfer and pressure drop in a duct.* Sparrow, E. M. and Hossfeld, L. M. 1984, International Journal of Heat and Mass Transfer, Vol. 27, Number 10, pp. 1715-1723.
14. *Experimental study on the performance of plate fin and tube heat exchangers at low Reynolds numbers.* Seshimo, Y., and Fujii, M. 1991, ASME/JSME Thermal Engineering Joint Conference, Vol. 4, pp. 449-454.
15. *Heat transfer predictions in cross flow over cylinders between two parallel plates.* Kundu, D., Skeikh, A. H. and Lou, D. Y. S. 1991, Numerical Heat Transfer: An International Journal of Computation and Methodology; Part A: Applications, Vol. 9, Number 3, pp. 361-377.
16. *Heat transfer in crossflow over cylinders between two parallel plates.* Kundu, D., Skeikh, A. H. and Lou, D. Y. S. 1992, Journal of Heat Transfer, Transactions ASME, Vol. 114, Number 3, pp. 558-564.
17. *Dehumidification: on the correlation of wet and dry transport processes in plate finned-tube heat exchangers.* Eckels, P. W., Rabas, T. J. 1987, Journal of Heat Transfer, Transactions ASME, Vol. 109, pp. 575-582.
18. *Experiments on convective heat transfer in corrugated channels.* Ali, M. M. and Ramadhyani, S. 1992, Experimental Heat Transfer, pp. 175-193.
19. *Air-side heat transfer correlations for flat and wavy plate fin-and-tube geometries.* Webb, R. L. 1990, ASHRAE Transactions, Vol. 90, Number 2, pp. 445-449.
20. *Correlations for predicting the air-side nusselt numbers and friction factors in chilled-water cooling coils.* Mirth, D. R. and Ramadhyani, S. 1994, Experimental Heat Transfer, Vol. 7, Number 2, pp. 143-162.
21. *Empirical correlations for heat transfer and flow friction characteristics of herringbone wavy fin-and-tube heat exchangers.* Wang, C. C., Hwang, Y. M. and Lin, Y. T. 2002, International Journal of Refrigeration, Vol. 25, Number 5, pp. 673-680.

22. *Heat transfer and friction correlations for wavy plate fin-and-tube heat exchangers.* Kim, N. H., Yun, J. H. and Webb, R. L. 1997, Journal of Heat Transfer, Transactions ASME, Vol. 119, Number 3, pp. 560-567.
23. *Numerical prediction of viscous flow and heat transfer in tube banks.* Launder, B. E. and Massey, T. H. 1978, Journal of Heat Transfer, Transactions ASME, Vol. 100, Number 4, pp. 565-571.
24. *Numerical analysis of laminar flow and heat transfer of air in an in-line tube bank.* Fujii, M., Fujii, T. and Nagata, T. 1977, Numerical Heat Transfer, Vol. 7, Number 1, pp. 89-102.
25. *Finite analytic solution of convective heat transfer for tube arrays in crossflow: Part I. Flow field analysis.* Wung, T. S. and Chen, C. J. 1989, Journal of Heat Transfer, Transactions ASME, Vol. 111, Number 3, pp. 633-640.
26. *Pressure and heat transfer in cross flow over cylinders between two parallel plates.* Kundu, D., Skeikh, A. H. and Lou, D. Y. S. 1991, Numerical Heat Transfer: An International Journal of Computation and Methodology; Part A: Applications, Vol. 9, Number 3, pp. 345-360.
27. *Laminar and turbulent heat transfer predictions in tube banks in cross flow.* Zdravistch, F., Fletcher, C. A. J. and Behnia, M. Bali, Indonesia : s.n., 1994. Proceedings of the International Conference on Fluid and Thermal Energy Conversion. pp. 12-16.
28. *Fluid flow and heat transfer in a plate-fin and tube heat exchanger (Analysis of heat transfer around a square cylinder situated between parallel plates).* Yamashita, H., Kushida, G. and Izumi, R. 1986, Bulletin of the JSME, Vol. 29, Number 258, pp. 4185-4191.
29. *Numerical studies of a compact fin-tube heat exchanger.* Bastini, A. and Fiebig, M. and Mitra, N. K. 1991, Proceedings of EURO THERM Seminar No. 18, Design and Operation of Heat Exchangers, pp. 154-163.
30. *Numerical and experimental studies of three-dimensional plate-fin and tube heat exchangers.* Jang, J. J., Wu, M. C. and Chang, W. J. 1996, International Journal of Heat and Mass Transfer, Vol. 39, Number 14, pp. 3057-3066.
31. *Effect of fin spacing on convection in a plate fin and tube heat exchanger.* Mendez, R. R., Sen, M., Yang, K. T. and McClain, R. 2000, International Journal of Heat and Mass Transfer, Vol. 43, Number 1, pp. 39-51.

32. *Numerical analysis of fluid flow and heat transfer characteristics in three-dimensional plate fin-and-tube heat exchanger.* Tutar, M. and Akkoca, A. 2004, Numerical Heat Transfer; Part A: Applications, Vol. 46, Number 3, pp. 301-321.
33. *Numerical modeling of heat transfer and fluid flow over herringbone corrugated fins.* McNab, C. M., Atkinson, K. N., Heikal, M. R. and Taylor, N. 1996, Proceedings of the 11th International Heat Transfer Conference, Kyongju, Korea, Vol. 6, pp. 119-124.
34. *Numerical Investigation of Performance Characteristic of Plate and Wavy Fin Tube Heat Exchangers in Transitional Flow Regime.* Amin, M. R. and Panse, S. P. Sydney, Australia : s.n., 2006. Proceedings of the 13th International Heat Transfer Conference,.
35. *Multi-scale model for turbulent flows.* Wilcox, D. C. 1986, AIAA 24th Aerospace Sciences Meeting, American Institute of Aeronautics and Astronautics.
36. *Two-equation eddy-viscosity turbulence models for engineering applications.* Menter, F. R. 1994, AIAA Journal, Vol. 32, Number 8, pp. 1598-1605.
37. *Wall functions for general application CFD codes.* Grotjans, H. and Menter, F. R. 1998, ECCOMAS 98 Proceedings of the Fourth European Computational Fluid Dynamics Conference, pp. 1112-1117.
38. ANSYS. *ANSYS® Academic Research, Release 12.1, CFX Documentation and Help System, CFX, ANSYS, Inc.* 2009.
39. Jiyuan.T, Guan H.Y and Chaoqun.L. *Computational Fluid Dynamics: A practical Approach.* s.l. : Butterworth-Heinemann, 2007. 0750685638.
40. Kays, W. M. and London, A. L. *Compact Heat Exchangers.* New York : 1984, 1984.
41. *Optimization study relevant to louvered fin compact heat exchangers.* R.A. Stephen, K.A. Thole. 2003, International Journal of Heat Exchangers.
42. *Effect of Inlet Flow Angle on Performance of Multilouvered Fin Heat Exchangers.* Guo, D. and Tafti, D. K. 2003, American Society of Mechanical Engineers, Heat Transfer Division, (Publication) HTD, Vol. 374, Number 1, pp. 103-109.
43. *Heat transfer and pressure drop characteristics of flat tube and louvered plate fin surfaces.* Achaichia, A and Cowell, T. A. 2, April 1988, Experimental Thermal and Fluid Science, Vol. 1, pp. 147-157.

44. *3-D thermal-hydraulic analysis for louver fin heat exchangers with variable louver angle*. Jang, Ching Tsun Hsieh and Jiin Yuh. 2006, Applied Thermal Engineering, pp. 14-15.
45. *Heat transfer analysis of lateral perforated fin heat sinks*. M.R. Shaeria, M. Yaghoubi, and K. Jafarpura. 2009, Applied Energy Volume 86, Issue 10, pp. 2019-2029.
46. *Numerical flow analysis of pulsatile flow in a channel with symmetric wavy walls at moderate Reynolds numbers*. Nishimura, T., Yoshino, T. and Kawamura, Y. 1987, Journal of Chemical Engineering of Japan, Vol. 20, Number 5, pp. 479-485.
47. *Temperature and concentration profiles in fully turbulent boundary layers*. Kader.B.A. 1981, International journal of heat and mass transfer Voume 24, pp. 1541-1544.
48. Incropera F. P. and Dewitt, D.P. *Fundamentals of heat and mass transfer*. s.l. : John Wiley and Sons, Inc, 1996.
49. Kakac, S., Bergles, A. E. and Mayinger, F. *Heat Exchangers: Thermal-Hydraulic Fundamental and Design*. New York : McGraw-Hill, 1993.
50. *Numerical study of laminar and turbulent heat transfer in a periodically corrugated wall channel*. Amano, R. S. 1985, Journal of Heat Transfer, Transactions ASME, pp. 564-569.
51. Faber, T.E. *Fluid Dynamics for Physicists*. s.l. : Cambridge university press, 1995.
52. *Effect of fin spacing on convection in a plate fin and tube heat exchanger*. Ricardo Romero-Mendez, Mihir Sen, K.T. Yang, Rodney McClain. 1999, International journal of Heat and Mass transfer, pp. 39-51.
53. *Numerical study of periodically turbulent flow and heat transfer in a channel with transverse fin arrays*. Yuan, Z. X. 2000, International Journal of Numerical Methods for Heat and Fluid Flow, Vol. 10, Number 8, pp. 842-861.
54. *Numerical prediction of transitional characteristics of flow and heat transfer in a corrugated duct*. Yang, L. C., Asako, Y., Yamaguchi, Y. and Faghri, M. 1997, ASME Journal of Heat Transfer, Vol. 119, pp. 62-69.
55. *Heat transfer and friction characteristics of fin-and-tube heat exchangers*. Yan, W. M., Sheen, P. J. 2000, International Journal of Heat and Mass Transfer, Vol. 43, Number 9, pp. 1651-1659.

56. *Numerical prediction of laminar flow and heat transfer in wavy channels of uniform cross-sectional area.* Xin, R. C. and Tao, W. Q. 1988, Numerical Heat Transfer, Vol. 14, Number 4, pp. 465-481.
57. *Finite analytic solution of convective heat transfer for tube arrays in crossflow: Part II. Heat transfer analysis.* Wung, T. S. and Chen, C. J. 1989, Journal of Heat Transfer, Transactions ASME, Vol. 111, Number 3, pp. 641-648.
58. *Heat transfer and friction characteristics of plain fin-and-tube heat exchangers, Part II: Correlation.* Wang, C. C., Chi, K. Y. and Chang, C. J. 2000, International Journal of Heat and Mass Transfer, Vol. 43, Number 15, pp. 2693-2700.
59. *Heat transfer and friction characteristics of plain fin-and-tube heat exchangers, Part I: New experimental data.* Wang, C. C. and Chi, K. Y. 2000, International Journal of Heat and Mass Transfer, Vol. 43, Number 15, pp. 2681-2691.
60. *A comparative study of compact enhanced fin-and-tube heat exchangers.* Wang, C. C., Lee, W. S. and Sheu, W. J. 2001, International Journal of Heat and Mass Transfer, Vol. 44, Number 18, pp. 3565-3573.
61. *A numerical study of heat transfer and fluid flow in a plate fin- and- tube heat exchanger.* Tutar, M., Akkoca, A. and Serdar, O. 2001, American Society of Mechanical Engineers, Pressure Vessels and Piping Division (Publication) PVP, Vol. 431, Number 3, pp. 77-84.
62. Thomas, L. C. *Fundamentals of heat transfer.* s.l. : Prentice-Hall Inc., 1980.
63. *Heat transfer enhancement in a serpentine channel.* Snyder, B., Li, K. T. and Wirtz, R. A. 1993, International Journal of Heat and Mass Transfer, Vol. 36, Number 12, pp. 2965-2976.
64. *Local and average transfer coefficients for one-row plate fin and tube heat exchanger configurations.* Saboya, F.E.M. and Sparrow, E.M. 1974, Journal of Heat Transfer, Transactions ASME, Vol. 96, pp. 265-272.
65. *Experiments on the transfer characteristics of a corrugated fin and tube heat exchanger configuration.* Saboya, F.E.M. and Sparrow, E.M. 1976, Journal of Heat Transfer, Transactions ASME, Vol. 9, pp. 26-34.
66. *Direct simulation of turbulent flow and heat transfer in a channel Part II: A Green's function technique for wavy walls.* Rutledge, J. and Sleicher, C. A. 1994, Communications in Numerical Methods in Engineering, Vol. 10, Number 6, pp. 489-496.

67. *Turbulent flow in a channel with a wavy wall.* Patel, V. C., Chon, J. T. and Yoon, J. Y. 1991, Journal of Fluids Engineering, Transactions of the ASME, Vol. 113, Number 4, pp. 579-586.
68. *Laminar flow over wavy walls.* Patel, V. C., Chon, J. T. and Yoon, J. Y. 1991, Journal of Fluids Engineering, Transactions of the ASME, Vol. 113, Number 4, pp. 574-578.
69. *Numerical computation of turbulent flows.* Launder, B. E. and Spalding, D. B. 1974, Computer Methods in Applied Mechanics and Engineering, Vol. 3, Number 2, pp. 269-289.
70. *Microscopic phenomena and macroscopic evaluation of heat transfer from plate fins/circular tube assembly using naphthalene sublimation technique.* Kim, J. Y., and Song, T. H. 2002, International Journal of Heat and Mass Transfer, Vol. 45, Number 16, pp. 3397-3404.
71. *Heat transfer characterization of flat plain fins and round tube heat exchangers.* Kayansayan, N. 1993, Experimental Thermal and Fluid Science, Vol. 6, Number 3, pp. 263-272.
72. *Numerical study of the effect of interwall spacing on turbulent flow in a corrugated duct.* Ergin, S., Ota, M., Yamaguchi, H. and Sakamoto, M. 1996, American Society of Mechanical Engineers, Heat Transfer Division, (Publication) HTD, Vol. 333, Number 2, pp. 47-54.
73. *Finned tube heat exchanger: Correlation of dry surface heat transfer data.* Elmahdy, A. H. and Biggs, P. F. 1979, ASHRAE Transactions, Vol. 85, Number 2, pp. 262-273.
74. *Enhancement of convective heat transfer from small heat sources to liquid coolants using strip fins.* Brinkman, R., Ramadhyani, S. and Incropera, F. 1988, Experimental Heat Transfe, pp. 315-330.
75. *Improved subgrid-scale models for large-eddy simulation.* Bardina, J. and Ferziger, J. H. and Reynolds, W. C. 1980, AIAA, pp. 1280-1357.
76. *Finite-volume solutions for laminar flow and heat transfer in a corrugated duct.* Asako, Y., and Faghri, M. 1987, Journal of Heat Transfer, Transactions ASME, pp. 627-634.
77. *3-D thermal-hydraulic analysis for louver fin heat exchangers with variable louver angle.* Jang, Ching-Tsun Hseih and Jiin-Yuh. 2006, Applied Thermal Engineering, pp. 1629-1639.

78. *A numerical investigation of heat transfer enhancement in offset strip fin heat exchangers in self sustained oscillatory flows.* Sunden, Arash saidi and Bengt. 2001, International journal of Numerical methods for heat and fluid flow, volume 7, pp. 699-717.
79. *Heat Transfer analysis of lateral perforated fin heat sinks.* M.R Shaeri, M. Yaghoubiand, K Jafarpur. 2009, Applied Energy volume 86, issue 10, pp. 2019-2029.
80. Deen, William M. *Analysis of transport phenomena.* New York : Oxford university press, 1998.
81. *Air-side heat transfer and friction correlations for plain fin-and-tube heat exchangers with staggered tube arrangements.* Kim, N. H., Youn, B. and Webb, R. L. 1999, Journal of Heat Transfer, Transactions ASME, Vol. 121, Number 3, pp. 662-667.