NUMERICAL STUDY OF FORCED CONVECTION HEAT TRANSFER OVER CIRCULAR AND OVAL TUBE BANKS USING VORTEX GENERATORS

A Dissertation
by
Md. Rejaul Haque (Student ID: 0413102058)

Under the Supervision of
Dr. Md. Ashiqur Rahman
Assistant Professor

Thesis submitted to the
Department of Mechanical Engineering in Partial Fulfillment of the Requirements for the Degree of

MASTER OF SCIENCE
in
Mechanical Engineering

Defense Date: 19th July, 2016

BANGLADESH UNIVERSITY OF ENGINEERING AND TECHNOLOGY
RECOMMENDATIONS OF THE BOARD OF EXAMINERS

The thesis titled, *Numerical Study of Forced Convection Heat Transfer over Circular and Oval Tube Banks Using Vortex Generators*, submitted by Md.RejaulHaque, Roll no.: 0413102058, session: April, 2013 has been accepted as satisfactory in partial fulfillment of the requirement for the degree of Master of Science in Mechanical Engineering on 19th July 2016.

**BOARD OF EXAMINERS**

Dr. Ashiqur Rahman  
(Supervisor)  
Assistant Professor  
Chairman  
Department of Mechanical Engineering, BUET  
Dhaka, Bangladesh

Dr. Mohammad Ali  
(Ex-Officio) Member  
Head of the Department  
Professor  
Department of Mechanical Engineering, BUET  
Dhaka, Bangladesh

Dr. Md. Abdur Rashid Sarkar  
Member  
Professor  
Department of Mechanical Engineering, BUET  
Dhaka, Bangladesh

Dr. Mohammad Nasim Hasan  
Member  
Associate Professor  
Department of Mechanical Engineering, BUET  
Dhaka, Bangladesh

Dr. Hasan Mohammad Mostofa Afroz  
Member (External)  
Professor  
Department of Mechanical Engineering, DUET  
Gazipur-1700, Bangladesh
DECLARATION

This thesis work does not contain any material extracted from elsewhere or from a work published by anybody else. The work of this thesis has not been published elsewhere by the author for any degree or any other University or Institute of learning.

MD. REJAUL HAQUE
Roll No. 0413102058
Session: April 2013
DEDICATION

This dissertation is dedicated to my parents
ACKNOWLEDGMENT

I would like to express my deepest gratitude to my supervisor Dr. Ashiqur Rahman, Assistant Professor, Department of Mechanical Engineering, Bangladesh University of Engineering and Technology, Dhaka for his support, guidance, inspiration, constructive suggestions and close supervision throughout the entire period of my graduate study. The author acknowledges his gratefulness to Mechanical Engineering Department for providing required facilities. I would like to take the opportunity to thank all the members of examination committee for their constructive comments and criticism. Finally, I wanted to express my deepest gratitude to my parents for their unparallel love, dedication, and encouragement throughout my whole life. Without their effort none of this would have been possible.
## CONTENTS

<table>
<thead>
<tr>
<th>TITLE</th>
<th>I</th>
</tr>
</thead>
<tbody>
<tr>
<td>RECOMMENDATIONS</td>
<td>II</td>
</tr>
<tr>
<td>OF THE BOARD OF EXAMINERS</td>
<td>III</td>
</tr>
<tr>
<td>DECLARATION</td>
<td>IV</td>
</tr>
<tr>
<td>DEDICATION</td>
<td>V</td>
</tr>
<tr>
<td>ACKNOWLEDGEMENT</td>
<td>VI-VII</td>
</tr>
<tr>
<td>CONTENTS</td>
<td>VIII-IX</td>
</tr>
<tr>
<td>NOMENCLATURE</td>
<td>X-XIII</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>XIII-XV</td>
</tr>
<tr>
<td>LIST OF GRAPHS</td>
<td>XV</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>XVI-XVII</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td></td>
</tr>
</tbody>
</table>

### CHAPTER ONE

**INTRODUCTION**

| 1.1 | GENERAL | 1 |
| 1.2 | BACKGROUND OF HEAT TRANSFER ENHANCEMENT | 2 |
| 1.2.1 | ACTIVE HEAT TRANSFER TECHNIQUES | 3 |
| 1.2.2 | PASSIVE HEAT TRANSFER TECHNIQUES | 3 |
| 1.3 | VORTEX GENERATORS | 6 |
| 1.4 | OBJECTIVES | 10 |

### CHAPTER TWO

**LITERATURE REVIEW**

| 2.1 | 12 |

### CHAPTER THREE

**NUMERICAL METHOD AND MODELING**

| 3.1 | GOVERNING EQUATIONS | 22 |
| 3.2 | FLOW MODEL | 24 |
| 3.3 | FLOW SOLVERS | 26 |
| 3.4 | COMPUTATIONAL DOMAIN AND BOUNDARY CONDITION | 29 |
| 3.5 | BOUNDARY CONDITIONS | 33 |
| 3.6 | MESH GENERATION | 34 |
| 3.7 | GRID INDEPENDENCY | 35 |
| 3.8 | CALCULATED PARAMETERS | 35 |

### CHAPTER FOUR

**RESULTS AND DISCUSSIONS**

| 4.1 | VALIDATION | 39 |
| 4.2 | 2-D PRESENTATION OF THE PRESENT WORK | 41 |
| 4.2.1 | EFFECT OF TUBE BANKS | 42 |
| 4.2.2 | EFFECT OF NUMBER OF VORTEX GENERATORS | 44 |
| 4.2.3 | HEAT TRANSFER COEFFICIENT, PRESSURE DROP AND NUSSELT NUMBER FOR DIFFERENT REYNOLDS NUMBER | 45 |
| 4.2.4 | EFFECT OF ANGLE OF ATTACK ON TEMPERATURE DISTRIBUTION | 47 |
| 4.3 | 3-D PRESENTATION OF THE PRESENT WORK | 48 |
| 4.3.1 | EFFECT OF VORTEX GENERATOR CONFIGURATION | 48 |
| 4.3.2 | EFFECT OF REYNOLDS NUMBER | 53 |
| 4.3.3 | NUMBER OF VORTEX GENERATORS | 63 |
| 4.3.4 | EFFECT OF TUBE SHAPE (CIRCULAR/OVAL) | 68 |
| 4.3.5 | EFFECT OF ASPECT RATIO OF OVAL TUBE | 72 |

**CHAPTER FIVE**
CONCLUSION 75

**CHAPTER SIX**
FUTURE RECOMMENDATION 77

**REFERENCES** 78

**APPENDIX** 82
NOMENCLATURE

$A$  heat transfer surface area ($m^2$)
$A_{min}$ minimum flow area ($m^2$)
$c_p$ specific heat (J/kg K)
$D$ outer tube diameter (m)
$D_h$ hydraulic diameter
$f$ friction factor
$h$ heat transfer coefficient (W/m$^2$ K)
$H$ channel height (m)
$j$ Colburn factor
$L$ flow length (m)
$l$ rectangular winglet length (mm)
$Nu$ Nusselt number
$P$ pressure (Pa)
$Pr$ Prandtl number
$Q$ heat transfer capacity (W)
$Re$ Reynolds number
$T$ Temperature (K)
$T_{in}$ Inlet temperature (K)
$T_{w}$ wall temperature (K)
$U$ velocity in the $x$-direction (m/s)
$V$ velocity in the $y$-direction (m/s)
$W$ velocity in the $z$-direction (m/s)
$w$ span of rectangular winglet (mm)
$V_m$ mean velocity

Greek Symbols

$\alpha$ angle of attack (deg)
$\mu$ Dynamic viscosity (Pa.s)
$\rho$ density (kg/m$^3$)
$K$ thermal conductivity (W/m K)

Subscripts

in inlet parameter
$m$ mean value
out outlet parameter

Abbreviation

AR Aspect Ratio
CHE Compact Heat Exchanger
CFU Common Flow Up
CFD Common Flow Down
CFD Computational Fluid Dynamics
HE Heat Exchanger
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LVG</td>
<td>Longitudinal Vortex Generator</td>
</tr>
<tr>
<td>RVG</td>
<td>Rectangular Vortex Generator</td>
</tr>
<tr>
<td>TVG</td>
<td>Transverse Vortex Generator</td>
</tr>
<tr>
<td>VG</td>
<td>Vortex Generator</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>----------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Figure-1</td>
<td>Example of (a) twisted tapes (b) twisted coils</td>
</tr>
<tr>
<td>Figure-2</td>
<td>Wire coil insert</td>
</tr>
<tr>
<td>Figure-3</td>
<td>Commercial transport aircraft with VGs</td>
</tr>
<tr>
<td>Figure-4</td>
<td>Most common type of LVG</td>
</tr>
<tr>
<td>Figure-5</td>
<td>Vortex systems behind a delta winglet</td>
</tr>
<tr>
<td>Figure-6</td>
<td>Passively generated longitudinal vortices in CHE</td>
</tr>
<tr>
<td>Figure-7</td>
<td>The natural formation of a laminar horseshoe vortex,</td>
</tr>
<tr>
<td></td>
<td>(a) Configuration of the VG geometric proposed, (b) Schematic of the flow</td>
</tr>
<tr>
<td></td>
<td>motions across the vortex generators</td>
</tr>
<tr>
<td>Figure-8</td>
<td>Schematic of the large winglet and two kinds of delta winglet arrays</td>
</tr>
<tr>
<td>Figure-9</td>
<td>geometry and placements</td>
</tr>
<tr>
<td>Figure-10</td>
<td>Winglet locations and geometry</td>
</tr>
<tr>
<td></td>
<td>Winglet locations and geometry: (a) winglet-only case (b) circular cylinder</td>
</tr>
<tr>
<td>Figure-11</td>
<td>Heat exchanger geometry and vortex generator configuration</td>
</tr>
<tr>
<td>Figure-12</td>
<td>A vortex generator installed in front of an electronic module</td>
</tr>
<tr>
<td></td>
<td>Secondary velocity vectors at the cross section of x=21 mm for Re=3000</td>
</tr>
<tr>
<td>Figure-13</td>
<td>Winglet type vortex generator dimensions and the placement with respect to</td>
</tr>
<tr>
<td></td>
<td>the tube</td>
</tr>
<tr>
<td>Figure-14</td>
<td>Longitudinal vortices at the different cross-sections (x in m)</td>
</tr>
<tr>
<td>Figure-15</td>
<td>Louvered fin heat exchanger</td>
</tr>
<tr>
<td></td>
<td>Distributions of isovels in three x–z sections for modified case at Re=1500</td>
</tr>
<tr>
<td>Figure-16</td>
<td>Configuration of winglet-type VG on a fin surface: (a) common-flow down</td>
</tr>
<tr>
<td></td>
<td>and (b) common-flow-up</td>
</tr>
<tr>
<td>Figure-17</td>
<td>Cross-sectional view of the triangular duct between any two parallel plates</td>
</tr>
<tr>
<td>Figure-18</td>
<td>Flow chart of the Pressure-Based Solution Method</td>
</tr>
<tr>
<td></td>
<td>Schematic of the core region of a fin-and-tube heat exchanger with RWPs</td>
</tr>
</tbody>
</table>
Figure 23: Schematic of the tube region of a fin-and-tube heat exchanger (a) circular tube (b) oval tube

Figure 24: Schematic diagram of domain with CFU configuration of winglets (a) Circular tube (b) Oval tube

Figure 25: Schematic diagram of domain for Oval tubes with CFD configuration of winglets

Figure 26: Orthographic views of domain with CFD configuration of winglets (a) Top view (b) Front view

Figure 27: Dimensions of tubes and vortex generators (a) Circular tube and Oval tubes for aspect ratios of (b) AR=1.24 (c) AR=1.54 (d) AR=1.80 (e) AR=2.32

Figure 28: Position of vortex generators at (a) angle of attack=15º (b) angle of attack=25º

Figure 29: The mesh refined computational domain along with boundary conditions

Figure 30: The mesh refined computational domain

Figure 31: Verification of the presented numerical scheme with respect to Ramadhan et al. [40]

Figure 32: Different configurations for fin-and-tube heat exchangers a) Baseline case (circular tube banks) b) winglet at 30º angle of attack (oval tube-banks) c) winglet at 45º angle of attack (circular tube-banks)

Figure 33: Velocity distributions in the channel for the a) baseline case and b) the inline-7RWP (circular tube banks) case c) the inline-7RWP (oval tube banks) case for Re=1775

Figure 34: Temperature distributions in the channel for the a) baseline case and b) the inline-7RWP case c) the inline-7RWP (oval tube banks) case at Re=1775

Figure 35: Streamline distributions in the channel for the a) baseline case and b) the inline-3RWP case c) baseline case (oval tube banks) d) the inline-3RWP (oval tube banks) case at Re=1775

Figure 36: Temperature distributions in the half channel (symmetry plane) for
oval tube banks for the a) baseline case and b) the inline-3RWP c) the inline-7RWP case at $Re=1775$.

Temperature distributions in the half channel(symmetry plane) for circular tube banks for the a) baseline case and b) 30°angle of attack c) 45°angle of attack at $Re=1775$.

Figure-41

Longitudinal vortices for CFU configuration at the different cross-sections (x=80, 85,90, 95,100 in mm)

Figure-42

Longitudinal vortices for CFU configuration (Oval tubes AR=1.80) at the different cross-sections (x=80, 85, 90, 95,100 in mm) at angle of attack of 15°.

Figure-43

Vortices distribution for CFD configuration (Oval tubes AR=1.80) at the different cross-sections (x=80, 85, 90, 95,100 in mm) at angle of attack of 15° with 7VGs

Figure-44

Simulated vortices for (CFU+CFD) configuration at the different cross-sections (x=80, 85, 90, 95,100 in mm) at angle of attack of 15° with 7VGs

Figure-45

Staggered configuration creates vortices at the different cross-sections (x=80, 85, 90, 95,100 in mm) at angle of attack of 15° with 7VGs

Figure-46

Velocity contours for Circular tubes at X=98 mm for 7 vortex generator configuration at a) Re=500 b) Re=700 c) Re=850 at angle of attack of 15°

Figure-49

Temperature contours for oval tubes at Exit plane for 7 VGs at a) Re= 500 b) Re=700 c) Re= 850 at angle of attack of 15°

Figure-50

Streamline distribution for oval tubes at x=89.5mm for 7 vortex generators at a)Re=850 b) Re= 700 c) Re=500 at angle of attack of 15°

Figure-51

Temperature contours for Circular tubes at X=172mm for Re=850 at a) Baseline b)1VG c) 3VGs d) 7VGs for 15°angle of attack
Velocity contours for circular tubes at Reynolds number =850 and at a distance x=172mm for a) Baseline b) 1VG c) 3 VGs d) 7VGs for 15°angle of attack

Figure-61

Temperature contours for Circular tubes and oval tubes at exit plane for Re=500 at a) Baseline b) 1VG c) 3 VGs d) 7VGs at 15°angle of attack

Figure-62

Temperature contours for Circular tubes at a plane just past two tubes from the location of vortex generators for a) Baseline b) 1VGc) 3 VGs d) 7VGs at 15°angle of attack

Figure-63

Velocity contours for circular tubes at Reynolds number =850 for 3 vortex generator configuration at defined plane for 15°angle of attack

Figure-66

Temperature contours for circular tubes at Reynolds number =850 for 3 vortex generator configuration at defined plane for 15°angle of attack.

Figure-67

Temperature contours for Oval tubes at Reynolds number =850 for 3 vortex generator configuration at defined plane for 15°angle of attack

Figure-68

Velocity vectors for oval tubes at Reynolds number =850 for 7 vortex generator configuration at defined plane for 15°angle of attack

Figure-69

Temperature contours for oval tubes at Reynolds number =850 for 7 vortex generator configuration at defined plane for 15°angle of attack

Figure-70

Temperature contours for a) circular tubes and b) oval tubes at X=177.8mm for Re=850 at AR=1.24, AR=1.54, AR=1.8, AR=2.32

Figure-72

Streamline for a) circular tubes and oval tubes at b) AR=1.24, c)AR=1.54, d)AR=1.8, e)AR=2.34.

Figure-73
# LIST OF GRAPHS

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>31</td>
<td>Validation of the present paper with <em>Chu et al.</em>[31]</td>
<td>39</td>
</tr>
<tr>
<td>32</td>
<td>Validation of the present paper with <em>Chu et al.</em>[31]</td>
<td>40</td>
</tr>
<tr>
<td>39</td>
<td>Staggered configuration creates vortices at the different cross-sections (x=80, 85, 90, 95, 100 in mm) at angle of attack of 15° with 7VGs</td>
<td>45</td>
</tr>
<tr>
<td>40</td>
<td>Average Nusselt number variations with the <em>Re</em> number for both Circular and Oval tubes</td>
<td>47</td>
</tr>
<tr>
<td>47</td>
<td>Area weighted average value of a) heat transfer coefficient and the b) pressure drop for a range of Re numbers at 15° angle of attack</td>
<td>52</td>
</tr>
<tr>
<td>48</td>
<td>Performance parameter (<em>h/p</em> ratio) for a range of Re numbers at 15° angle of attack with 7VGs</td>
<td>53</td>
</tr>
<tr>
<td>52</td>
<td>Velocity distribution along the midline of YZ plane at a distance X=98mm for oval tubes (AR=1.80) at different Reynolds number and 15°angle of attack</td>
<td>55</td>
</tr>
<tr>
<td>53</td>
<td>Temperature distribution along the midline of YZ plane at a distance X=98mm for Oval tubes (AR=1.80) at different Reynolds number and 15°angle of attack.</td>
<td>56</td>
</tr>
<tr>
<td>54</td>
<td>Performance parameter (a) heat transfer coefficient and (b) pressure drop variations with Reynolds number for CFU configuration of VG at 15°angle of attacks</td>
<td>57</td>
</tr>
<tr>
<td>55</td>
<td>Variation of heat transfer coefficient and pressure drop with Reynolds number for different aspect ratios of Oval tubes at 15°angle of attack</td>
<td>58</td>
</tr>
<tr>
<td>56</td>
<td>Effect of Angle of attack on (a) heat transfer coefficient and (b)</td>
<td>60</td>
</tr>
</tbody>
</table>
pressure drop with Reynolds number for 7VGs.

Heat flux per unit pressure drop with Reynolds number at 7VGs for 15° angle of attack

Figure-57

Friction factor at various Reynolds number for different number of vortex generators at 15° angle of attack

Figure-58

Area goodness factor at various Reynolds number for different number of vortex generators

Figure-59

Velocity and Temperature distribution along the midline of YZ plane at a distance X=98mm for circular tube at different number of vortex generators

Figure-64

Percentage variation of heat transfer coefficient and pressure drop with Number of vortex generators

Figure-65

Velocity distribution along the midline of YZ plane at a distance X=98mm for circular tube at different aspect of ratios.

Figure-71

LIST OF TABLES

Table 1     Dimensions of the computational model     30
Table 2     Boundary condition of the computational model     33
Table 3     Grid independency test     35
ABSTRACT

The present work represents a 2-D and 3-D numerical investigation of forced convection heat transfer over circular and oval tube banks consisting of seven rows of tubes in an inline arrangement with rectangular longitudinal vortex generators (LVG) placed at the bottom of the channel. At first, the effects of common flow up and common flow down pairs of vortices produced by vortex generators (VG) with rectangular winglets restudied for a range of Reynolds numbers. The results show that effect of LVGs could effectively enhance the heat transfer of the heat exchanger in case of common flow up configuration of the winglets. Subsequently, the effects of Reynolds number (from 550 to 1775 in 2-D /from 500 to 850 in 3-D), the number of vortex generators (1, 3, and 7) and the angle of attack (30° and 45° in 2D, and 15° and 25° in 3D) of rectangular VGs on the heat transfer and fluid flow characteristics are examined. The characteristics of average Nusselt number, associated pressure drop, friction factor, area goodness factor, streamline distribution and temperature contours etc. are studied numerically by the aid of the computational fluid dynamics. In case of the 2-D analysis, heat transfer coefficient is increased by 80.5% for 7VGs and 45° angle of attack configuration compared to the baseline case. At the same time, circular tube banks give a nearly three times more pressure loss than that of oval tubes under the same operating conditions.

The 3-D cases are also evaluated and the circular tube (7VG) is found to exhibit an enhancement of heat transfer coefficient of 45% with an associated pressure drop penalty of 127%. In the case of oval tubes, an aspect ratio of 1.24 gives a remarkable heat transfer enhancement of 26.5% at a mere 69% increase in the pumping power than for oval tubes having aspect ratio of 2.34. For the Reynolds number ranging from 500 to 850, for the case of 1-RWP (Rectangular Winglet Pair), the increment of j/ f ratio is 27% over 7-RWP case. As number of VG increases, the area goodness factor reduces to ~0.18 from ~0.23. From the viewpoint of “area goodness,” the 1-RWP and 3-RWP case will require a smaller frontal area than the other cases and would be more efficient in compact heat exchanger design. However, in both cases, the streamline plot shows an excellent recirculation region near the VG’s position indicating that the addition of vortex generators intensifies the mixing between the hot and cold fluids and thereby enhances heat transfer significantly. The performance of the oval and circular tube banks with vortex generators under the same operating conditions is compared with a focus on finding the preferable configuration of vortex generators.
KEY WORDS-CFD, Compact heat exchanger, Vortex generators, Heat transfer enhancement, Tube banks, Aspect ratio, Angle of attack
CHAPTER ONE

INTRODUCTION

Heat exchangers find wide-spread deployment in diverse fields such as automobile industry, heating and air conditioning, power system, chemical engineering, electronic chip cooling and aerospace etc. The subject of heat transfer enhancement is of significant interest in developing compact heat exchanger to meet the objectives of high efficiency and low cost with the smallest volume and least weight possible. The present research focuses on heat transfer enhancement.

1.1 GENERAL

For the economic success of energy conversion systems, the efficiency of energy converters, cooling units and heat exchangers (HE) is one of the most important aspects. As a result, heat transfer enhancement has become a major concern in various engineering fields such as heating, ventilating, air conditioning, and refrigeration (HVAC&R) systems, automobiles, electronic cooling and chemical industry. A heat exchanger (HE) is a complex device that provides the thermal energy transfer between two or more fluids, which are at different temperatures and are in thermal contact with each other. HE is used either individually or as components of a large thermal system in a wide variety of commercial, industrial and household applications.

Considering its overwhelming area of application, an efficient heat exchange device will surely save much energy and associated costs. Designing heat exchanger devices to improve efficiency is not a new concept, rather a path explored by many researchers. But somehow engineers still able to find new ideas every now and then. So, the improvements in the performance of the HE have attracted many researchers for a long time as they are of great technical, economical, and not the least, ecological importance. Performance improvement becomes essential particularly in HE with gases because the thermal resistance of gases can be 10 to 50 times as large as that of liquids, which requires large heat transfer surface area per unit volume on the gas side. A large ratio area per unit volume defines a class of HE called by Compact Heat Exchanger (CHE).

The traditional methods of reducing the air-side thermal resistance are either increasing the surface area of the HE or reducing the thermal boundary layer thickness on the surface of the
HE. Increasing the surface area is effective but results in the increase of material cost and mass of the heat exchanger (an easier way to make it is increase the fin density of HE). This approach becomes unfeasible some application especially those which have not enough physical space, such as: automotive radiator, intercooler and condenser etc. Recently controlling the flow of the fluid in convective heat transfer became a priority Issue for enhancing the heat transfer.

Enhancing heat transfer by controlling fluid flow has a penalty in the form of increased pumping power. An efficient heat exchanger device should have a high heat transfer rate at minimum pumping power. There are several techniques by which this optimum state can be achieved. These techniques of heat transfer enhancement significantly reduce operating temperatures and thermally induced stresses. In order to enhance the heat transfer, passive devices like roughness elements, dimples or cavities are used, mostly in micro heat exchangers or other energy converters of micro-scales. On larger scale, other devices in use are guiding plates but the main goal of this thesis is to explore other method mentioned previously that reduces boundary layer thickness by generation of passive vortices. In this technique the flow field is altered by an obstacle that produces an oriented vortex in the direction of the flow. This changes the flow and, consequently, alters the local hydrodynamic and thermal boundary layer. The net effect of this manipulation is an average increase in the heat transfer, usually associated with penalty in pressure drop.

Overall, the present work is undertaken to compute, by Numerical Simulation, the enhancement heat transfer levels achievable in a Vortex Generators (VG) mounted fin-tube HE.

1.2 BACKGROUND OF HEAT TRANSFER ENHANCEMENT

The Heat transfer techniques generally use surface or geometrical modifications to the flow channel by incorporating inserts or additional devices. They promote higher heat transfer coefficients by disturbing or altering the existing flow behavior (except for extended surfaces) which also leads to increase in the pressure drop. In case of extended surfaces, effective heat transfer area on the side of the extended surface is increased. Passive techniques hold the advantage over the active techniques as they do not require any direct input of external power, rather they use from the system itself which ultimately leads to an increase in fluid pressure drop.
Various active mechanical enhancing methods are also available which can be used to enhance heat transfer. These techniques are more complex from the use and design point of view as the method requires some external power input to cause the desired flow modification and improvement in the rate of heat transfer. In these cases, external power is used to facilitate the desired flow modification and the concomitant improvement in the heat transfer.

1.2.1 Active heat transfer enhancement techniques

Mechanical aids consist of stirring the fluid or rotating the surfaces by mechanical means. Mechanical surface scrappers and rotating heat exchanger ducts are used to augment the heat transfer.

Low or high frequency surface vibrations are used to promote single phase heat transfer enhancement. A piezoelectric device may also be used to vibrate a heat transfer surface for augmentation of heat transfer. The fluid vibration is the more practical type of vibration enhancement due to the mass of the heat exchangers and they mostly applied for the single phase fluids.

Electrostatic field is used for dielectric fluids to cause proper bulk mixing of the fluid in the vicinity of the heat transfer surface. An electric field applied to dielectric fluid imposes a body force on the fluid which influences the fluid motion. Here a gas is supplied to a flow of liquid through the porous surface or the same fluid is injected upstream. The injected gas augments the single phase flow heat transfer. Suction involves vapor removal, in nucleate or film boiling, or fluid withdrawal through a porous heated surface and applied to only single phase fluids.

Jet impingement involves spraying a liquid on the hot surface which spreads as a thin film and gets evaporated. Single or multiple jets may be used for this purpose.

1.2.2 Passive heat transfer enhancement techniques

A porous coating on the base surface is an effective enhancement method for film condensation as condensate drainage is assisted by capillary flow within the porous coating, thinning of the condensate film thickness occurs. The temperature drop across a laminar condensate film depends on the condensation thermal resistance and
capillary assisted film thinning reduces the condensate thermal resistance. Surfaces may be made rough by machining or by restructuring the base surface or by placing some “roughness” adjacent to the surface. The knurled roughness on the vertical surface promotes the mixing in the condensate film.

Using a plain fin may increase the surface area but a special shape extended surface may increase heat transfer coefficient in addition to the area of heat exchanger. The extended surfaces for liquids typically use much smaller fin heights than that used for gases.

Some devices include a number of geometrical arrangements or tube inserts for forced flow that create rotating or secondary flow. Full length twisted tape inserts in Figure 1(a) or inlet vortex generator and axial coil in Figure 1(b) inserts with a screw type winding are some examples of swirl flow devices used for heat transfer enhancement.

![Figure 1](image_url)

*Figure 1 – Example of (a) twisted tapes (b) twisted coils [1]*

There are some devices that are inserted into the flow channel to improve energy transport at the heated surface indirectly as the displaced inserts mix the main flow in addition to that in the wall region heat transfer enhances. Displaced wire coil in Figure 2 insert is not attached to the wall of the tube and these devices periodically mix the gross flow structure but not affecting the main flow significantly.

Use of surface tension forces to affect condensate drainage is an effective enhancement technique. The surface tension devices do not increase the surface area of the base surface. Heat pipes typically use capillary wicking to transport liquid from
9. Delaying the boundary layer development;
10. Thermal dispersion;
11. Increasing the order of the fluid molecules;
12. Redistribution of the flow;
13. Modification of property of the convective medium;
14. Increasing the difference between the surface and fluid temperatures;
15. Increasing fluid flow rate passively;
16. Increasing the thermal conductivity of the solid phase using special nanotechnology fabrication.

1.3 VORTEX GENERATORS

Firstly, vortex generators (VGs) have been used successfully in many aeronautical applications for reduction of separation in both internal and external flows. The vortices created transfer low energy fluid from the surface into the mainstream and bring higher energy fluid from the mainstream down to the surface, where the higher kinetic energy level is able to withstand a greater pressure rise before separation occurs. A typical commercial transport aircraft with VGs built-on airplane wing is shown in Figure 3.

![Figure 3 - Commercial transport aircraft with VGs](image)

Especially for aircraft, controlling flow separation could result in an increase in system performance, with consequent energy conservation, as well as, weight and space savings. Regarding to industrial applications, VGs are beneficial to transfer high momentum
fluid from the outer flow into the boundary layer in order to increase the fluid temperature close to wall in a HE or to increase momentum to a boundary layer exposed to an adverse pressure gradient, in order to delay separation. In fact, both effects are probably occurring.

The first results about the influence of LVG on heat transfer were reported and the investigation includes the influence of a row of triangular winglets on the drag and heat transfer of a circular cylinder normal to an air stream and found that the cylinder drag was reduced due to the separation delay caused by the longitudinal vortices and that heat transfer was somewhat increased. It is also reported that local heat transfer results for a row of co-rotating and counter-rotating vortices generated by a row of delta winglets attached to one channel wall. For the generation of co-rotating vortices the winglets had all the same attack angle, while for the counter-rotating vortices the attack angle alternated for successive winglets. Heat transfer and pressure drop data were first measured for fin-tube geometry with longitudinal vortex generators in the form of very slender rectangular winglets. It is stated that triangular configurations and small attack angles gave disappointing performances in preliminary investigations.

Delta and rectangular wing, delta and rectangular winglet, trapezoidal delta wing and ribs are some types of VGs. A great deal of research use these simple devices, due to its efficient, low maintenance and operating cost in HE. Figure 4 shows the most common VGs applied on CHE.

![Figure 4 – Most common type of LVG](image)

There are two types of VGs: transverse vortex generator (TVG) and longitudinal vortex generator (LVG). The rotational direction of the TVG is normal to the main flow direction and the flow is 2-D, while the LVG have their rotating axes parallel to the main flow and the flow is 3-D as shown in Figure 5.
These imposed vertical motions outlined, provide heat transfer performance enhanced with relatively low pressure penalty. This is due to pressure drop associated with wall friction and related to the derivative of the streamwise velocity (the spanwise and normal velocities have little effect) as shown in Figure 8 and Figure 9. However, the spanwise and normal velocities play a significant direct role in convective heat transfer.

1.4 OBJECTIVES

As mentioned earlier in this chapter, saving thermal energy, reduction of material and cost reduction are key words nowadays regarding heat transfer optimization. According to this trend, the present study is focused on enhance of the heat transfer considering a fin-tube CHE with VGs and seven row tubes (commercial appliance). In order to study the influence of winglets on the heat transfer characteristics and flow structure for fin-and-tube heat exchangers, a comparative investigation for fin-and-tube heat exchangers with and without winglet for both Circular and Oval tube banks is planned to be performed. The level of heat transfer enhancement and pressure drop depends on the VG geometry (size, shape, angle of attack, aspect ratio etc.) and the Reynolds number. The winglet pairs are symmetrically mounted adjacent to the tubes. However, by considering all the factors, the main objectives of the present research work can be summarized as follows:

- To perform a 2D and 3D numerical study of flow structure and laminar convection heat transfer on compact heat exchangers (for both circular and oval tubes) with longitudinal vortex generators.
- To obtain the understanding of the heat transfer, flow structure and pressure drop encountered in a compact heat exchanger, with and without vortex generators and to validate the present scheme with the results presented in a well reputed Journal.
- To study the performance of the heat exchanger for winglets at various angles of attack and a range of Reynolds number etc.
- To compare the average increment of the heat transfer coefficient over the pressure drop for all the possible combinations and to obtain optimized configurations among them.
So, it can be said that the present study would investigate the effect of rectangular winglet vortex generators for flow through a compact heat exchanger, using different geometric configurations of tube and vortex generators. To fulfill the objectives, numerical investigation is performed to find the effect of most significant parameters of the heat exchanger on heat transfer characteristics.

This current investigation is expected to be a valuable addition to the existing literature and will provide a reliable design guideline in heat transfer enhancement for heat exchangers using vortex generators.
CHAPTER TWO

The early studies of LVGs focused on their application to straight channels, where the flow structures are relatively simple. While most early studies were performed experimentally, subsequent research made use of numerical simulation for the application of LVGs to fin-and-tube heat exchangers. These numerical investigations were made possible due to the rapid development of computer hardware and software. As a part of this present work, a detailed review on the germane published papers on heat transfer enhancement using numerical methods has been conducted.

2.1 LITERATURE REVIEW

As fluid passes across the LVG, strong secondary swirling flows are generated, and the tangential velocity of the vortices can be as high as twice the main flow velocity. The high-velocity swirling secondary flow not only promotes mixing of the fluids in the main flow and in the edge regions, but also can guide the high-energy fluid into the boundary layer to suppress and/or delay boundary layer separation, with a corresponding decrease of profile drag.

A numerical study was performed by Brockmeier et al. [8] to evaluate the impact of delta-wing and delta-winglet VG in flatplate channels. A delta wing with an aspect ratio of one was considered, with attack angles from 10° to 50° and Reynolds numbers from 1000 to 4000. With delta winglets at a 30° angle of attack, an average increase of 84% in the Nusselt number was predicted at a Reynolds number of 4000. No pressure drop predictions were presented. The results of this paper are very limited because of the small size of the computational domain. The channel length extended only 1.31 wing chord lengths downstream from the wing. Although detailed distributions of velocity, vorticity, and temperature were presented, these results are valid only for a distance downstream slightly larger than the wing chord itself. The incoming flow used in the computation was a fully developed, laminar channel flow, while the temperature field was modeled as developing.

Shizawa et al. [9] carried out an experiment of a single vortex in a pressure-driven 3-D turbulent boundary layer. The experimental results indicated that the rotating direction of vortex was very important in its interaction with the boundary layer. If the vortex induced a flow near the wall in the same direction as the transverse velocity of the boundary layer, the
perturbations induced by the vortex in the boundary layer would decay quickly. Otherwise, if the vortex induced a velocity near the wall in the opposed direction with the transverse flow in the boundary layer, then this boundary layer perturbations might persist. Therefore, it can be seen that the vortices fully changes the flow structure, and it should be considered carefully if a turbulence flow with LV is predicted or simulated by the traditional turbulence model.

Fiebeig [10] demonstrated that longitudinal and transverse vortices are formed if the angle is attack is very small and if the flow direction is perpendicular to the surface of vortex generators, respectively. Fiebig presented the idea of using delta winglet vortex generators in heat transfer enhancement. They considered heat exchanger elements with 3 tube rows and a delta-winglet pair downstream of each tube. While for an inline tube arrangement they found a 55–65% increase in heat transfer, with a pressure drop increase of 20–45%, for the staggered arrangement they found less enhancement and pressure loss. Both type of vortex enhances the heat transfer significantly.

Lee et al. [11] numerically studied the heat transfer characteristics and turbulent structure in a three dimensional turbulent boundary layer with longitudinal vortices. They indicated that the disturbance of the boundary layer caused the best heat transfer enhancement in the region where the flows are directed toward the wall but the vortex core is the region of relatively lower mixing.

Lau et al. [12] experimentally studied the momentum and heat transport in the turbulent channel flow with embedded longitudinal vortices. He showed the effects of the shapes of interrupted surfaces on the performance of the fin-and-tube heat exchanger used in home air conditioners. The scaled-up model experiments were conducted to evaluate the heat transfer coefficient and pressure drop, and the prototype experiments were also performed to examine the validity of the scaled-up experiments.

Sahaet al.[13] studied the vortex structures and kinetic energy budget in two-dimensional flow past a square cylinder at a Reynolds number of 100. The flow in the wake was found to be unsteady with a strong periodic component. Liouet al.[14] employed 12 different shaped vortex generators to experimentally study the heat transfer enhancement in a sharp turning two-pass square channel. Vasudevanet al. [15] numerically predicted the heat transfer enhancement of plate fin heat exchanger with triangular inserts between the plates. The study is carried out with the delta winglet mounted on the slant surfaces as well as punched from the surfaces with varying angles of attack of the winglet. Two different thermal boundary conditions are studied. Lozzaet al.[16] discussed the results of an extensive investigation
about the performance of various fin configurations and aimed to enhance the heat transfer capabilities of air-cooled condensers and liquid coolers. For louvered fins it was found that the louver height largely influences its performance and that the quality of the pressing process is essential to the achievement of the best results.

A three dimensional Numerical studies of oval tube and winglet pair were studied by Tiwari et al. [17]. Using common flow up (CFU) configuration Kwak et al. [18] measured the heat transfer coefficient and pressure drop in 2, 3, 4, and 5-row fin-tube heat exchangers with a DWVG in a range of $280 < \text{Re} < 2400$. Compared with heat exchangers with plain fins, those with DWVG fins showed a 10% increase in heat transfer performance of the j-factor for all heat exchangers, and a 0–10% increase of the f-factor for the 2, 4, and 5-row heat exchangers and for the 3-row heat exchanger, however, the f-factor dramatically decreased by 30–50%. O’Brien et al. [19], from an experimental study in Figure 10 reported that single winglet pair to the oval tubes yields 38% higher than the oval tube without winglet.

Figure 10 - Winglet locations and geometry [19]

In a later study for circular tubes, O’Brien et al. [20], found that heat transfer also increases for circular-winglet combination as shown in Figure 11. At low Reynolds number, it was reported to increase as a factor of 2 and at high Reynolds number; it was close to 50%.

It may be useful to indicate that the field synergy principle is a new idea or concept of the heat transfer enhancement mechanism; it is not the comparison criterion for which a number of comparison criteria can be found in [21].
From the results presented later, it can be found that any heat transfer enhancement by the LVG is always accompanied by a better synergy between velocity and temperature gradient, thus un-doubtfully demonstrated that the fundamental reason for LVG enhancement is in the improvement of the field synergy.

Sohal [22] performed an experimental study on a simulated fin-tube heat exchanger at Reynolds number range of 670–6300, the enhancement level is close to 50% they also found an average Nusselt number enhancement ratio of 35%.

The effect of the number of DWVG rows was investigated by Kwak et al. [23]. The heat exchanger with a single DWVG row experienced 10–30% larger heat transfer capacity and 34–55% less pressure drop and with two DWVG rows, however, heat transfer capacity and the pressure drop further increased by 6–15% and by 61–117%, respectively.

Sommers et al. [24] predicted the effect of frost on heat exchanger performance and suggested that vortex-induced flow suppresses dendritic frost growth as shown in Figure12. Pesteeiet al. [25] experimentally studied the effect of winglet location on heat transfer enhancement and the pressuredrop in a fin tube heat exchanger. The results showed that the average Nusselt number increases by about 46% while the local heat transfer coefficient increases by several times.
Sanders [26] tested the effects of winglets incorporated into a more realistic louvered fin geometry and measured heat transfer augmentation as high as 53% along the tube wall with 21% increase in pressure losses.

In case of louvered fin heat exchangers, Jacobi et al. [27] experimentally evaluated full-scale wind-tunnel testing of a compact heat exchanger typical to those used in automotive systems for both dry- and wet-surface conditions for a louvered-fin baseline and for a vortex-enhanced louvered-fin heat exchanger. An average heat transfer increase over the baseline case of 21% for dry conditions and 23.4% for wet conditions was achieved with a pressure drop penalty smaller than 7%.

Chomdee et al. [28] experimentally investigate the heat transfer enhancement by DWVGs in air cooling of a staggered array of rectangular electronic modules as given in Figure 13 the winglet VGs are placed in front of 3×5 modules with 20° attack angle. It could be seen that the VGs could enhance the adiabatic heat transfer coefficients, reduce the thermal wake functions and the module temperatures significantly.

Hiravennavar et al. [29] numerically studied a delta winglet pair of non-zero thickness in a hydrodynamically developed and thermally developing laminar channel flow and observed that the heat transfer is enhanced by 33% when single delta winglet is used and 67% when a winglet pair is used. The performance of the DWVG in flat finned tube heat exchangers was
examined by Allison et al. [30]. They compared the performance of two fin types, a plain fin with a DWVG and a louvered fin with CFU configuration. The j-factors and f-factors of the heat exchanger with the DWVG were 87% and 53% respectively, better than that of the heat exchangers with a louver fin. The air side heat transfer and friction characteristic of 5 types of fin was analyzed and reported by Tang et al. [2]. Tian et al. [4] reported the analysis for wavy fin which showed good augmentation over plain fin as shown in Figure 14. It was observed that with increase of angle of attack, both Nusselt number and friction factor increase but increasing row number results in a decrease of friction factor.

Figure 14- Secondary velocity vectors at the cross section of x=21 mm for Re=3000[4]. Better performance for staggered arrangement of tube bank compared to inline tube arrangement was reported by Chu et al.[31].

Figure 15 Winglet type vortex generator dimensions and the placement with respect to the tube [31]

However, the pressure drop penalty was found to increase due to the extra form drag induced by the staggered tubes. The analysis was conducted for circular tubes only as shown in Figure 15. Tao et al [32] presents numerical computation results on laminar convection heat transfer in a rectangular channel with a pair of rectangular winglets longitudinal vortex generator punched out from the lower wall of the channel. The effect of the punched holes and the
by less than 12%. For three DWVG rows, the heat transfer coefficient was augmented by 29.9–68.8% as the pressure drop penalty increased 26% at Re = 960 and 87.5% at Re = 220. Three-dimensional numerical study [34] was performed for heat transfer characteristics and fluid flow structure of fin-and-oval-tube heat exchangers with longitudinal vortex generators (LVGs). For Re (based on the hydraulic diameter) ranges from 500 to 2500, it was found that the average Nu for the three-row fin-and-oval-tube heat exchanger with longitudinal vortex generators increased by 13.6–32.9% over the baseline case as shown in Figure 18 and the corresponding pressure loss increased by 29.2–40.6%.

*Figure-18* Distributions of isovels in three x–z sections for modified case at Re = 1500 [34]

Hirokazu [35] indicated that continuous punched hole can lead to reduction of fin efficiency, thus the impact of the punching should be assessed. Moreover, if the back-row small winglet adjacent to the preceding one in a VG array, the vortices generated by different winglets may affect each other and the interactions of the vortices should be considered.

He *et al.* [3] observed that increasing Reynolds number gives higher heat transfer coefficient and lower friction factor as shown in Figure-19.
The solution of the complete Navier Stokes equation and the energy equation is carried out using the staggered grid arrangement as shown in Figure 20. The performance of the combination of triangular secondary fins and delta wing with stamping on slant surfaces has also been studied [36].

Eiamsa-arde et al. [1] investigated the heat transfer, flow friction and thermal performance factor characteristics in a tube fitted with delta winglet twisted tape, using water as working fluid at Reynolds number range of 3000–27000 with oblique delta-winglet twisted tape (O-DWT) and straight delta-winglet twisted tape (S-DWT) arrangements. Nusselt numbers from using the O-DWT and S-DWT with twist ratio of 3 are around 7–29% and 15.8–45.6% greater than those from the tapes with twist ratios of 4 and 5, respectively. In addition, friction factors for O-DWT and S-DWT with twist ratio of 3 are approximately 3.5–22.8% and 17.9–37.8%. The data calculated by them showed that the mean Nusselt number and friction factor for the ODWT are respectively, 4.2% and 7.8% higher than those for the SDWT. The enhancement of heat transfer coefficient for common flow up configuration of vortex generators was reported by Kannan et al. [37]. Aris et al. [38] experimentally and numerically studied the thermal-hydraulic performance of a heat pipe fin stack with shape memory alloy vortex generators in forced air convection. They achieved a heat transfer enhancement of 37% and a reduction in fan operating cost.

Ebrahimiet et al. [39] conducted a simulation study for both rectangular winglet and delta winglet pairs. The common flow up configuration along with rectangular winglet pair was found to give a higher heat transfer coefficient with the penalty of higher pressure drop than delta winglet pair. They found that the optimum result is obtained for delta winglet pair with common flow down configuration.
Ramadhan et al. [40] focuses on the Influence of the different parameters of VGs on heat transfer and fluid flow characteristics of three rows oval-tube banks. The characteristics of average Nu number and skin friction coefficient are studied numerically by the aid of the computational fluid Dynamics (CFD). The results showed increasing in the heat transfer and skin friction coefficient with the increasing of Re number and decreasing the relative distance of positions of LVGs.

Phadtare et al. [41] suggested that the VGs could be used to enhance turbine blade tip heat transfer cooling. They found that the heat transfer coefficient of the 2D VG might be a factor of 1.8 higher than that of the smooth tip. This arrangement achieved at the expenses of a penalty of pressure drop around 30% experimentally. They suggested that the VGs could be used to enhance blade tip heat transfer cooling.

No study could be located in the open literature comparing the effect of vortex generators on the heat transfer and pressure drop characteristics of circular and oval tube banks under the same operating conditions. In the present study, the heat transfer characteristics and fluid flow structure of fin-and-tube heat exchangers with vortex generators are explored for both circular and oval tube banks. A full scale model is considered in our numerical study that involves VGs in all the tube rows of a seven-row fin-and tube heat exchanger.
Z-momentum:

\[
\rho \left[ U \frac{\partial W}{\partial x} + V \frac{\partial W}{\partial y} + W \frac{\partial W}{\partial z} \right] = -\frac{\partial P}{\partial z} + \mu \left[ \frac{\partial^2 W}{\partial x^2} + \frac{\partial^2 W}{\partial y^2} + \frac{\partial^2 W}{\partial z^2} \right]
\]

3.1(d)

Energy equation:

\[
\rho c_p \left[ U \frac{\partial T}{\partial x} + V \frac{\partial T}{\partial y} + W \frac{\partial T}{\partial z} \right] = k_f \left[ \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right]
\]

3.1(e)

The five governing equations are presented here where \( U, V, W \) are the three velocity components along the three axis and \( P, T, \rho \) represents the pressure, temperature and density of the flowing fluid respectively.

### 3.1.1 Simulation Methods

Air flow is assumed as incompressible with constant properties and turbulent SST k-omega model is used as a turbulence model, considering complex flows (wake or separation) and anisotropic turbulence. A commercial CFD (Computational Fluid Dynamics) code (Ansys Fluent) was applied based on the Finite Volume Methodology. The Continuity, momentum and energy equations with the boundary condition equations are solved by using computational fluid dynamics software (FLUENT). The mesh around the tubes and the vortex generators are refined for the most corrected numerical data. The convective terms in the Navier-Stokes (X-momentum and Y-momentum) equations and energy equation are discretized with the second order upwind scheme. The velocity and pressure coupling is performed with SIMPLE algorithm. The Turbulent kinetic energy and specific dissipation rate is discretized with first order upwind scheme. The convergence criterion for the continuity, velocities, \( K, \omega \) is that the absolute criteria of residual is 0.001 and the convergence criterion for the energy is that the maximum value of the absolute criteria is \( 1.0 \times 10^{-6} \). According to literature, the generation of longitudinal vortices is a quasi-steady phenomenon. Consequently, due to the low inlet velocity and the small fin pitch (in 3-D), the flow in the channel of the compact heat exchanger is assumed to be laminar and steady. The Navier–Stokes and energy equations with the boundary condition equations are solved.
3.2 FLOW MODEL

Reynolds number for the heat exchanger flow in is 500,750 and 850 for 3-D and 530, 750, 1150, 1400 and 1775 for 2-D, which means that the flows can be laminar, transitional, or turbulent at different flow velocities. In this research, laminar model for 3D and turbulence model (SST k-omega) for 2D is considered.

Moreover, others methods could be used to predict the turbulence (besides RANS models). LES (Large Eddy Simulation) solves for the motion of the larger eddies while DNS (Direct Numerical Simulation, the most accurate and time-consuming/computationally expensive method of all for calculating turbulence) solves for all motion using the Navier-Stokes equations. LES and DNS are computationally very expensive due to requirements including very fine 3D mesh and transient behavior. Due to hardware requirements, LES and DNS were not used for this thesis.

In this thesis, the turbulence model for 2-D flow modeling evaluated is SST k-omega.

SST k-omega model is based on Standard k-omega model, which is based on the Wilcox (1994) k – ω model, which incorporates modifications for low-Reynolds-number effects, compressibility, and shear flow spreading. One of the weak points of the Wilcox model is the sensitivity of the solutions to values for k and ω outside the shear layer (free stream sensitivity). The standard k-omega model is an empirical model based on model transport equations for the turbulence kinetic energy (k) and the specific dissipation rate (ω), which can also be thought of as the ratio of ε to k, Wilcox (1994).

As the k – ω model has been modified over the years, production terms were added to both the k and ω equations, which improved the accuracy of the model for predicting free shear flows by combination of ω and ε equations. The idea of this new turbulence model is to retain the accurate formulation of the k – ω model in the boundary-layer region and to that add the advantage of the k – ε model in the region of free shear layers to reduce the sensitivity to the free stream flow. To achieve this idea the k – ε model is transformed into an equivalent k – ω formulation. This new method is called Shear-Stress Transport (SST) k – ω turbulence model.

Shear-Stress Transport (SST) k-omega model was proposed as a type of hybrid model, combining two models in order to better calculate flow in the near-wall region. It was designed in response to the problem of the k-epsilon model unsatisfactory near-wall performance for boundary layers with adverse pressure gradients. It utilizes a standard k-epsilon model to calculate flow properties in the free-stream (turbulent) flow region far from
the wall, while using a modified k-epsilon model near the wall using the turbulence frequency $\omega$ as a second variable instead of turbulent kinetic energy dissipation term $\varepsilon$. Since the air in this thesis is flowing between two flat fins very close to each other, it is expected that the boundary layer flow has a strong influence on the results, and properly modeling this near-wall flow could be important for accuracy of the calculations.

Shear-Stress Transport (SST) k-omega model was developed by Menter (1994). The SST k-omega model is similar to the standard k-omega model, but includes the following refinements, Ansys Fluent (2010):

- The standard k-omega model and the transformed k-epsilon model are both multiplied by a blending function and both models are added together. The blending function is designed to be one in the near-wall region, which activates the standard k-omega model, and zero away from the surface, which activates the transformed k-epsilon model.
- The SST model incorporates a damped cross-diffusion derivative term in the $\omega$ equation.
- The definition of the turbulent viscosity is modified to account for the transport of the turbulent shear stress.
- The modeling constants are different.
- The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm was devised by Patankar. Various forms probably are the most widely used of primitive-variable methods and are typically the core algorithm in most commercial engineering CFD codes. The method is very robust on coarse grids, but exhibits quite low asymptotic convergence rates. Hence, it is effective for calculating rough solutions to even very complex problems, but rapidly loses its effectiveness as grids are refined to achieve more spatial accuracy. Through the years numerous modifications were observed to the basic SIMPLE procedure, specifically intended to improve performance for time-dependent problems, such as SIMPLEC (SIMPLE-Consistent)
- The SIMPLE algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field. SIMPLE is the most applied, but many problems will benefit from the use of SIMPLEC. For relatively uncomplicated problems (laminar flows with no additional models activated), in which convergence is limited by the pressure-velocity coupling, is often obtained a converged solution more quickly using SIMPLEC. For complicated flows involving turbulence and/or additional physical models, SIMPLEC will improve convergence.
only if it was limited by the pressure-velocity coupling. Often, it will be one of the additional modeling parameters that limit convergence; in this case, SIMPLE and SIMPLECT will give similar convergence rates.

- SIMPLE algorithm is widely used in numerical simulation in compact heat exchanger. The SIMPLE algorithm gives a method of calculating pressure and velocities. The method is iterative, and when other scalars are coupled to the momentum equations the calculation needs to be done sequentially.

In this thesis, 3-Dimensional flow modeling is performed using the Laminar flow model.

### 3.3 FLOW SOLVERS

ANSYS Fluent allows us to choose one of the two numerical methods:

- Pressure-based solver
- Density-based solver

Historically speaking, the pressure-based approach was developed for low-speed incompressible flows, while the density-based approach was mainly used for high-speed compressible flows. However, recently both methods have been extended and reformulated to solve and operate for a wide range of flow conditions beyond their traditional or original intent.

In both methods the velocity field is obtained from the momentum equations. In the density-based approach, the continuity equation is used to obtain the density field while the pressure field is determined from the equation of state.

On the other hand, in the pressure-based approach, the pressure field is extracted by solving a pressure or pressure correction equation which is obtained by manipulating continuity and momentum equations.

Using either method, ANSYS Fluent will solve the governing integral equations for the conservation of mass and momentum, and (when appropriate) for energy and other scalars such as turbulence and chemical species. In both cases a control-volume-based technique is used that consists of:

- Division of the domain into discrete control volumes using a computational grid.
- Integration of the governing equations on the individual control volumes to construct algebraic equations for the discrete dependent variables (“unknowns”) such as velocities, pressure, temperature, and conserved scalars.
Linearization of the discretized equations and solution of the resultant linear equation system to yield updated values of the dependent variables.

The two numerical methods employ a similar discretization process (finite-volume), but the approach used to linearize and solve the discretized equations is different.

### 3.3.1 Pressure-Based Solver

The pressure-based solver employs an algorithm which belongs to a general class of methods called the projection method. In the projection method, the constraint of mass conservation (continuity) of the velocity field is achieved by solving a pressure (or pressure correction) equation. The pressure equation is derived from the continuity and the momentum equations in such a way that the velocity field, corrected by the pressure, satisfies the continuity. Since the governing equations are nonlinear and coupled to one another, the solution process involves iterations wherein the entire set of governing equations is solved repeatedly until the solution converges.

Two pressure-based solver algorithms are available in ANSYS Fluent. A segregated algorithm and a coupled algorithm. These two approaches are discussed in the sections below.

The pressure-based solver uses a solution algorithm where the governing equations are solved sequentially (i.e., segregated from one another). Because the governing equations are nonlinear and coupled, the solution loop must be carried out iteratively in order to obtain a converged numerical solution. In the segregated algorithm, the individual governing equations for the solution variables (for example, \(u, v, w, p, T, k, \varepsilon\) and so on) are solved one after another. Each governing equation, while being solved, is “decoupled” or “segregated” from other equations, hence its name. The segregated algorithm is memory-efficient, since the discretized equations need only be stored in the memory one at a time. However, the solution convergence is relatively slow, inasmuch as the equations are solved in a decoupled manner. With the segregated algorithm, each iteration consists of the steps outlined below:

1. Update fluid properties (for example, density, viscosity, specific heat) including turbulent viscosity (diffusivity) based on the current solution.
2. Solve the momentum equations, one after another, using the recently updated values of pressure and face mass fluxes.
3. Solve the pressure correction equation using the recently obtained velocity field and the mass-flux.
4. Correct face mass fluxes, pressure, and the velocity field using the pressure correction obtained from Step 3.

5. Solve the equations for additional scalars, if any, such as turbulent quantities, energy, species, and radiation intensity using the current values of the solution variables.

6. Update the source terms arising from the interactions among different phases (for example, source term for the carrier phase due to discrete particles).

7. Check for the convergence of the equations.

These steps are continued until the convergence criteria are met in Figure 21.

Figure 21 - Flow chart of the Pressure-Based Solution Method
3.3.2 Pressure-Velocity Coupling Scheme

Pressure-velocity coupling is achieved by deriving an additional condition for pressure by reformatting the continuity equation. The pressure-based solver allows to solve flow problem in either a segregated or coupled manner. ANSYS Fluent provides the option to choose among five pressure-velocity coupling algorithms: SIMPLE, SIMPLEC, PISO, Coupled, and (for unsteady flows using the non-iterative time advancement scheme (NITA) Fractional Step (FSM). All the aforementioned schemes, except the “coupled” scheme, are based on the predictor-corrector approach. The SIMPLE algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field.

3.3.3 Discretization of Momentum Equation (Second-Order Upwind Scheme)

When second-order accuracy is desired, quantities at cell faces are computed using a multidimensional linear reconstruction approach. In this approach, higher-order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about the cell centroid.

3.4 COMPUTATIONAL DOMAIN AND BOUNDARY CONDITION

3.4.1 Circular and Oval Tubes

The geometry of a seven-row fin-tube heat exchanger is considered for the computational domain. Fin spacing and tube size are similar to those of fin tube heat exchangers that are widely used in industrial system. The computational domain is considered as a channel as shown in Figure 22.
Figure 22 - Schematic of the core region of a fin-and-tube heat exchanger with RWPs.

The fin thickness is very small and is neglected. In case of oval tubes, the cross-sectional area of the tube is same to that of circular tubes as shown in Figure 23 and major and minor radius are calculated based on different aspect ratios of the oval tubes. The position of the winglet is fixed with respect to the center of the circular and oval tube. The dimensions are listed here in Table 1 and is figure out in Figures 24-26.

Figure 23 - Schematic of the tube region of a fin-and-tube heat exchanger (a) circular tube (b) oval tube

<table>
<thead>
<tr>
<th>Parameter</th>
<th>2-D Case</th>
<th>3-D Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Channel Dimension (mm)</td>
<td>H=177.8 L=177.8</td>
<td>H=3.63 W=25.4 L=177.8</td>
</tr>
<tr>
<td>Circular tube Radius (mm)</td>
<td>r=4.59</td>
<td>r=5.34</td>
</tr>
<tr>
<td>Oval tube Radius (mm)</td>
<td>a=6 b=3.5</td>
<td>a=5.98, b=4.8; a=6.62, b=4.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>a=7.17, b=4; a=8.13, b=3.5</td>
</tr>
</tbody>
</table>

Table 1 - Dimensions of the computational model
Figure 24- Schematic diagram of domain with CFU configuration of winglets (a) Circular tube (b) Oval tube

Figure 25- Schematic diagram of domain for Oval tubes with CFD configuration of winglets
3.4.2 Number of Vortex Generators and Size

The number of vortex generators is varied for different cases. We considered no vortex generators as baseline case and 1Vg, 3Vg, 7Vg for other three different cases. The length (L) of rectangular winglets for 2-D analysis is $L = 6.5\text{mm}$, thickness $\partial = 0.25\text{mm}$. Again, for 3-D case, length $L= 10.67 \text{mm}$, width $w=2.18 \text{mm}$ and thickness $\partial = 0.18\text{mm}$. The winglets can be arranged as Common Flow Up (CFU) and Common Flow Down (CFD) configuration.

3.4.3 Aspect Ratio of Oval Tubes

The ratio of major radius to minor radius of the oval tube is termed as Aspect ratio. Numerical investigation was done for different aspect ratios of the oval tubes for the stated conditions. Most of the results are considered for AR=1.8 of oval tubes with comparison of circular tube as shown in Figure 27.
**Figure 27**-Dimensions of tubes and vortex generators (a) Circular tube and Oval tubes for aspect ratios of (b) AR=1.24 (c) AR=1.54 (d) AR=1.80 (e) AR=2.32

### 3.4.4 Angle of Attack

Each vortex generator attached on the bottom fin could rotate around the axis. For 2-D cases, the angles of attack $\alpha$ of the winglets are varied by $30^\circ$ and $45^\circ$ and for 3-D, and the angles of attack $\alpha$ of the winglets are varied by $15^\circ$ and $25^\circ$ as shown in Figure 28.

![Figure 28](image)

**Figure 28**-Position of vortex generators at (a) angle of attack=15° (b) angle of attack=25°

### 3.5 Boundary Conditions

The boundary conditions are listed in Table 2:

<table>
<thead>
<tr>
<th>Segment</th>
<th>Boundary conditions</th>
<th>Fluid</th>
<th>Energy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity</td>
<td>310.6K</td>
<td></td>
</tr>
<tr>
<td>Tube wall</td>
<td>No slip</td>
<td></td>
<td>291.7K</td>
</tr>
<tr>
<td>Side walls</td>
<td>Symmetry</td>
<td></td>
<td>Symmetry</td>
</tr>
<tr>
<td>Top Wall</td>
<td>Symmetry</td>
<td></td>
<td>Symmetry</td>
</tr>
<tr>
<td>Bottom wall</td>
<td>Periodic</td>
<td></td>
<td>Periodic</td>
</tr>
<tr>
<td>Winglets</td>
<td>No slip</td>
<td></td>
<td>adiabatic</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure</td>
<td></td>
<td>adiabatic</td>
</tr>
</tbody>
</table>
**Table 2- Boundary condition of the computational model**

**3.6 Mesh Generation**

Mesh generation is the process of discretizing the whole domain into a set of sub domain. Numerical Results are greatly mesh sensitive. The entire domain is subdivided into elements as shown in Figure 29 and 30. The interpolation (shape functions) used for the pressure variable should be different from that used for velocities. The pressure variable should be interpolated with functions that are one order less than those used for the velocity field and the approximation may be discontinuous.

![Figure 29- The mesh refined computational domain along with boundary conditions](image)

![Figure 30- The mesh refined computational domain](image)
3.7 Grid Independency

In order to validate the independency of solution on the grid, different grid systems are investigated and the change in the averaged Nu number is less than 0.88% among the different grid systems as seen from table 3. For the present study, the final grid number is selected to be 389669 cells. Similar validations are also conducted for other cases.

<table>
<thead>
<tr>
<th>Number of Cells</th>
<th>Average Nusselt number (Nu)</th>
</tr>
</thead>
<tbody>
<tr>
<td>12247</td>
<td>38.1</td>
</tr>
<tr>
<td>17328</td>
<td>37.8</td>
</tr>
<tr>
<td>33330</td>
<td>34.6</td>
</tr>
<tr>
<td>167842</td>
<td>33.9</td>
</tr>
<tr>
<td>389669</td>
<td>33.8</td>
</tr>
</tbody>
</table>

Table 3-Grid independency test

3.8 Calculated Parameters

a) Reynolds number (Re)

Reynolds number is the ratio of inertia force to viscous force. It is a dimensionless number and is used as a criterion for a flow to be laminar or turbulent. Reynolds number based on hydraulic diameter of the domain can be calculated by the following equation.

\[
Re = \frac{\rho V_m D_h}{\mu}
\]

Where, \( \rho \) is the fluid density, \( V_m \) is the mean flow velocity, \( D_h \) is the hydraulic diameter and \( \mu \) is the dynamic viscosity of the flowing fluid. For 2-D case Reynolds number are 530, 750, 1150, 1400, 1775 where mean velocity is 0.63ms\(^{-1}\), 0.85ms\(^{-1}\), 1.1ms\(^{-1}\), 1.6ms\(^{-1}\), 2ms\(^{-1}\) respectively. On the other hand, Reynolds numbers 500(1.15ms\(^{-1}\)), 700(1.61ms\(^{-1}\)) and 850(1.95ms\(^{-1}\)) are considered for 3-D simulation.
b) Nusselt number\( (Nu) \)

The local Nusselt number based on diameter of the tube can be defined as:

\[
Nu = \frac{hD}{K}
\]

Where \( h \)=local heat transfer coefficient and \( k \)= thermal conductivity of the flowing fluid.

c) Pressure\( (P) \)

Pressure drop is hydraulic loss due to the roughness of the surface over which the fluid is flowing. Higher pressure drop requires higher pumping energy for flowing of a fluid. The mean pressure of a cross section and pressure drop is defined as:

\[
\bar{P} = \frac{\iiint p\,dA}{A}\]

where, \( dA \) is the elemental surface area

\( \Delta P = \bar{P}_{in} - \bar{P}_{out} \) Pressure drop, \( \bar{P}_{in} \)=Pressure at inlet and \( \bar{P}_{out} \)=Pressure at outlet

d) Temperature\( (T) \)

The mean temperature of a cross section and the log mean temperature is defined as:

\[
\bar{T} = \frac{\iiint uT\,dA}{A}
\]

where, \( dA \) is the elemental surface area
\[ A = \text{Total surface area of the selected surface} \]

\[
\Delta T = \frac{(T_w - \overline{T}_{in}) - (T_w - \overline{T}_{out})}{\ln \left( \frac{(T_w - \overline{T}_{in})}{(T_w - \overline{T}_{out})} \right)}
\]

Where, \( T_w = \text{Tube wall temperature}, \) \( T_{in} = \text{air inlet temperature}, \) \( T_{out} = \text{air outlet temperature}. \)

e) Heat flux \((Q)\)

The local heat transfer is defined as:

\[
Q = m C_p (\overline{T}_{out} - \overline{T}_{in})
\]

Where, \( m = \text{mass flow rate}, \) \( C_p = \text{specific heat of the flowing fluid} \)

f) Heat transfer coefficient \((h)\)

The local heat transfer coefficient is defined as:

\[
h = \frac{Q}{A \Delta T}
\]

g) Friction factor coefficient \((f)\)

For the calculation of area goodness factor \((j/f)\), we have to determine the friction factor \((f)\) from the simulated pressure drop results.

\[
f = \frac{\Delta P}{\rho V_m^2 \frac{2}{A A_{\text{min}}}}
\]

Where, \( A_{\text{min}} = \text{minimum cross flow area} \)

h) Stanton number \((St)\)
The Stanton number, $St$, is a dimensionless number that measures the ratio of heat transferred into a fluid to the thermal capacity of the fluid. It arises in the consideration of the geometric similarity of the momentum boundary layer and the thermal boundary layer, where it can be used to express a relationship between the shear force at the wall and the total heat transfer at the wall. It is used to characterize heat transfer in forced convection.

\[
St = \frac{h}{\rho V_m C_p}
\]  \hspace{1cm} \text{3.8(j)}

i) Colburn factor($j$)

To check the effectiveness of overall enhancement phenomenon (in terms of area goodness factor), the Colburn factor ($j$) deduced from the following equation:

\[
j = St \cdot \Pr^{2/3}
\]  \hspace{1cm} \text{3.8(k)}

Where, $\Pr =$ Prandlt number

[38]
CHAPTER FOUR

RESULTS AND DISCUSSIONS

In this chapter, the results of heat transfer phenomenon in a rectangular channel with rectangular vortex generators are evaluated and presented. The validation of the scheme, grid independency test, averaged Nusselt number, mean temperature; pressure drop and total heat flux etc. are compared with and without vortex generators in the channel at the prescribed angle of attack for different Reynolds numbers. The use of vortex generators increases the heat transfer coefficient compared to the case without vortex generators considerably.

4.1 VALIDATION

In order to validate the reliability of the numerical method being used, the numerical simulation is conducted for a heat exchanger with the same geometrical configurations as presented in Chu et al.[31]. The predicted results are compared with the numerical results from Chu et al.[31]. The overall pressure loss penalty $P$ and the air-side heat transfer coefficient $h_{air}$ are shown in Figure 31. The average discrepancy between the predicted

![Figure 31- Validation of the present paper with Chu et al.[31].](image-url)
pressure loss and the numerical values is less than 1%, and the average discrepancy between the predicted air-side heat transfer coefficient and the numerical values is less than 20% as seen in Figure 32. The good agreement between the predicted and numerical results indicates that the numerical model is reliable to predict heat transfer characteristics and flow structure in compact heat exchangers.

Figure 32 - Validation of the present paper with Chu et al.[31].
In order to verify the reliability of the numerical method being used, the numerical simulation is conducted for a fin-and-tube heat exchanger with the model presented in above Figure 33. The peak velocity is 16.4% lower in case of baseline model (a), the peak velocity is 3.6% lower in case of 30º angle of attack (b), the peak velocity is 2.03% higher in case of 45º angle of attack (c). The good agreement among the findings ascertains that our numerical model is reliable to predict heat transfer characteristics and flow structure in compact heat exchangers.

4.2 TWO DIMENSIONAL PRESENTATION OF THE PRESENT WORK

In order to study the influence of winglets on the heat transfer characteristics and flow structure for fin-and-tube heat exchangers, a comparative investigation for fin-and-tube heat exchangers with and without winglet for both Circular and Oval tube banks is performed. The winglet pairs are symmetrically mounted adjacent to the tubes. The angle of attack $\alpha$ is set at 30º and 45º. The $Re$ based on the channel height ranges from 550 to 1775. Figure 34 shows different configurations for fin-and-tube heat exchangers with and without winglet pairs.
Figure 34- Different configurations for fin-and-tube heat exchangers a) Baseline case (circular tube banks) b) winglet at 30º angle of attack (oval tube-banks) c) winglet at 45º angle of attack (circular tube-banks)

4.2.1 Effect of Tube Banks

Velocity Contours

Figure 35- Velocity distributions in the channel for the a) baseline case and b) the inline-7RWP (circular tube banks) case c) the inline-7RWP (oval tube banks) case for $Re=1775$ at 45º angle of attack.

A high velocity region is formed along the side of the tube as seen from Figure 35(a). Comparing Figure 35(a) with Figure 35(b), it is observed that the velocity distribution and the vortices structure for the inline-7RWP case are different from those for baseline case. The winglet pair develops length-wise vortices that delay the boundary layer separation from each tube. As a result, the reduction of the recirculation region accelerates the local flow velocity.
and hence the disruption of the thermal boundary layer intensifies the mixing between fluids. A reduced size of the recirculation region (marked by circle) is noticed in Figure 35(b). Figure 35(c) represents the velocity distribution characteristics for oval tube and shows a much lower velocity distribution than that of Figure 35(a) and Figure 35(b). A yellow region in the color code near the oval tube surface and the region occupied between the two tubes show a little mixing than circular tubes.

**Temperature Contours**

![Temperature Contours](image)

*Figure 36-* Temperature distributions in the channel for the a) baseline case and b) the inline-7RWP case c) the inline-7RWP (oval tube banks) case at $Re=1775$ at $45^\circ$ angle of attack

The local temperature distributions for the baseline case and the inline-7RWP case at $Re=1775$ are illustrated in Figure 36. From a comparison of Figure 36(a) with Figure 36(b), the temperature distribution in the vicinity of the inlet region is almost identical for the baseline the inline-7RWP cases. As the air approaches the RWPs, the longitudinal vortices are generated and the heat transfer is significantly enhanced. The enhancement of heat transfer is notable even far downstream of RWPs because the longitudinal vortices persist for several winglet chords and the red region eventually diminishes along the channel length, thereby ensuring a lower outlet temperature (marked by circle in b) and a better enhancement. At the same time, the wake zone behind each tube reduces by adding winglets (marked by rings). Thus; the temperature behind the RWPs for the inline-3RWP case is considerably lower than that in the corresponding region for the baseline case. The results indicate that the RWPs can significantly enhance the heat transfer with moderate pressure loss penalty. For
both the baseline and enhanced configurations, the variations in the temperature distribution are almost identical up to the location of the first tube.

Streamline Distributions

Figure 37- Streamline distributions in the channel for the a) baseline case and b) the inline-3RWP case c) baseline case (oval tube banks) d) the inline-3RWP (oval tube banks) case at Re=1775.

In case of the baseline and the inline-3RWP, the recirculation region is seen at the rear of the tube for both circular and oval tubes. In case of oval tubes, the ring is quite small compared to that of circular tubes. In case of winglet attachment, more fluid tends to go through the tube surface, thereby creates partial vacuum at the rear end of the winglet, and causes a great recirculation zone which thereby disrupt the thermal and velocity boundary layers and intensifies the mixing between the hot and cold fluids.

2.2 Effect of Number of Vortex Generators
Figure 38- Temperature distributions in the half channel(symmetry plane) for oval tube banks at angle of attack of 45º for the a) baseline case and b) the inline-3RWP  c) the inline-7RWP case at \(Re=1775\).

The fact that when the winglets are at a subsequent tube positions, it tends to rush more fluids on the second tube wall, thereby intensifying the heat transfer as temperature differences increases between the tube wall and flowing fluid can be observed from Figure 38.

4.2.3 Heat Transfer Coefficient, Pressure Drop and Nusselt Number for Different Re Numbers

![Graph showing heat transfer coefficient and pressure drop for different Re numbers](image)

Figure 39-Performance parameters (a) heat transfer coefficient and the (b) pressure drop for a range of \(Re\) numbers for both Circular and Oval tubes
The variation in the air-side heat transfer coefficient, $h$ for a range of $Re$ number is noticed in Figure 39. The heat transfer coefficients, $h$ for both the baseline case and the enhanced cases (both circular and oval tube banks) increase with increasing Reynolds number as is observed in Figure (a). With a higher value of Reynolds number, the thermal boundary layer thickness decreases and the degree of fluid mixing increases. Consequently, a global augmentation in heat transfer is observed with the increase in Reynolds number and angle of attack. Comparing with the baseline case, the heat transfer coefficient for the inline-3RWP case is improved by 38.1–43.6% over the range of the Reynolds number considered for angle of attack of 45°. The heat transfer coefficient for the inline- 7RWP case is increased from 75% at $Re =550$ to 80.5% at $Re =1775$ compared with the baseline case. The inline-7RWP case improves the heat transfer coefficient, $h$ from 58.5% to 84.1% over the baseline case for oval tube banks. The heat transfer coefficient is increased from 52.5% at 30 degree angle of attack to 74.4% at 45 degree angle of attack for $Re =550$. Hence, it can be concluded from Figure 39(a) that heat transfer coefficient increases with the increase of winglet number, orientation of the winglet and the Reynolds numbers.

The heat transfer augmentation is usually accompanied by additional pressure penalty. Comparing with the baseline case, the inline-3RWP case increase in pressure drop by 75Pa over the range of the Reynolds number considered in Figure 39(b). The inline-7RWP case increase in pressure drop by 210 Pa, and 58 Pa for the inline-7RWP case (oval tube banks). The incremental pressure drop is mainly due to the additional formed drag induced by the winglet vortex generators and the circular tube bank shows a nearly 3 times more pressure drop than that of oval tubes under the same conditions. So, from the point of view of pressure drop penalty, the oval tubes with winglet pair can be regarded as the best combination.

It was also found that the heat transfer performance parameter Nusselt number increases with the increase of Reynolds number as shown in Figure 40. The highest value of Nusselt number is found around 40 for 7VG at $Re=1775$. The influence of vortex generators actually reduces the size of wake region behind each tube and flow will be accelerating at this narrow region and hence increases the Nusselt number.


**Figure 40-** Average Nusselt number variations with the $Re$ number for both Circular and Oval tubes

### 4.2.4 Effect of Angle of Attack

**Figure 41-** Temperature distributions in the half channel (symmetry plane) for circular tube banks for the a) baseline case and b) 30° angle of attack  c) 45° angle of attack at $Re=1775$.

The angle of attack has a great impact on temperature distribution. The red portion indicating the inlet higher temperature persist throughout the bottom of the channel far to outlet for baseline case showing a poor heat transfer while increasing the angle of attack of the vortex generator, it diminishes near to 5th tube showing a lower outlet temperature. Thereby, the mixing is intensified and the heat transfer is enhanced.
4.3 THREE DIMENSIONAL PRESENTATION OF THE PRESENT WORK

4.3.1 Effects of Vortex generators Configuration

Velocity Vectors for Common Flow Up (CFU) and Common Flow Down (CFD) Configurations

*Figure -42* Longitudinal vortices for CFU configuration at the different cross-sections (x=80, 85, 90, 95, 100 in mm) at angle of attack of 15º with 7VGs

*Figure -43* Longitudinal vortices for CFU configuration (Oval tubes AR=1.80) at the different cross-sections (x=80, 85, 90, 95, 100 in mm) at angle of attack of 15º with 7VGs.
In CFU configuration, the transverse distance between the leading edges of the winglet pair is more than the distance between the trailing edge; hence, the left winglet forms vortices rotating in a counterclockwise direction while the right winglet forms vortices rotating in a clockwise direction as shown in Figure 42 and in Figure 43. The vortices create the up wash flow between the vortices while the down wash flow on the outer region of vortices towards the lower channel wall. Also, the distance between the vortex cores decreases, hence, the interaction between the counter rotating vortex pair increases considerably. It is also observed that the core of the vortices in CFU gradually moves from the lower wall to upper wall as the flow moves downstream. The thinning of the thermal boundary layer is on the outer region of the vortices.

![Vortices distribution for CFD configuration (Oval tubes AR=1.80) at the different cross-sections (x=80, 85, 90, 95,100 in mm) at angle of attack of 15° with 7VGs](FLOW)

*Figure -44 Vortices distribution for CFD configuration (Oval tubes AR=1.80) at the different cross-sections (x=80, 85, 90, 95,100 in mm) at angle of attack of 15° with 7VGs*

On the contrary, The CFD configuration of a winglet pair is demonstrated when the transverse distance between the leading edges is less than that of the trailing edges. For CFD, the left winglet forms vortices rotating in a clockwise direction while the right winglet forms vortices rotating in a counterclockwise direction as shown in Figure 44. These vortices create the down wash flow towards the lower channel wall while the up wash flow is away from the wall which is found in the outside region of the vortices. Along the downstream direction, the secondary velocity vectors decreases while the distance between the vortex cores increases. Thinning of the thermal boundary layer thus occurs in between the two vortices.
Figure -45 Simulated vortices for (CFU+CFD) configuration at the different cross-sections (x=80, 85, 90, 95,100 in mm) at angle of attack of 15º with 7VGs

Figure -46 Staggered configuration creates vortices at the different cross-sections (x=80, 85, 90, 95,100 in mm) at angle of attack of 15º with 7VGs

The Effects of CFD and CFU pairs of vortices produced by LVGs with rectangular shape of winglet are studied at various Reynolds number. Results show that effect of LVGs could effectively enhance the heat transfer of the heat exchanger. The common flow up configuration shows a better overall performance than the CFU, CFU+CFD and staggered configuration for the RWP. The longitudinal vortices are generated behind the RWPs due to the pressure difference and the friction. The induced spanwise velocity of longitudinal
vortices is almost 2~3 times as much as that of the frontal velocity. This strong swirling flow will transport the fluid in the tube wake region to the main flow regions. The improved bulk fluid mixing between the tube wake and main flow by the longitudinal vortices is one of the important heat transfer mechanisms. The development of the secondary flow obtained from the simulations is presented at five different locations along the stream wise direction downstream of the LVGs for Re=850. The plots show the generation of the counter rotating vortices by the LVGs, their gradual deformation and decrease in their strength as they move downstream in the channel. The effects of channel walls on vertical structures can be seen when the cross flow at any two sections of the winglets are compared. Two counter rotating main vortices behind the winglets cause the fluid to churn. This churning motion causes the fluid near the wall to flow in the central region. The circular tubes show a better heat transfer coefficient due to high velocity induced flow near the tube wall than that of oval tubes.

**Variation of Heat Transfer Coefficient and Pressure Drop with Reynolds Number**

![Variation of Heat Transfer Coefficient and Pressure Drop with Reynolds Number](image-url)
Increasing Reynolds number increases heat transfer coefficient for all the stated cases. Circular tube with vortex generators in CFU configuration shows a higher heat transfer capacity than that of oval tubes in CFU, but considering the pressure drops from Figure 47(b) the overall performance of oval tube is better than the circular ones.

At the same time, Oval tubes with 7 vortex generators in CFU gives the best output results considering the pressure drop penalty. Increasing Reynolds number decreases the ratio while it shows highest values at low Reynolds number due to lower pressure drop offered by the flowing fluid resistance as shown in Figure 48.
Figure 48—Performance parameter \( (h/p) \) ratio for a range of \( Re \) numbers at 15° angle of attack with 7VG

4.3.2 Effect of Reynolds Number

Velocity Contours for YZ Plane
*Figure 49-* velocity contours for circular tubes at X=98 mm for 7 vortex generator configuration at a) Re=500 b) Re=700 c) Re=850 at angle of attack of 15º

The Reynolds number has a significant impact on heat transfer distribution. Increasing Reynolds number defines a higher velocity of the fluid as seen by the color contours. The deepest brown color indicates the highest velocity under stated condition which tends more fluid to go to the central tube region and thereby intensifies the mixing.

**Temperature Contours at Exit Plane**

*Figure 50-* Temperature contours for oval tubes at Exit plane for 7 VGs at a) Re= 500 b) Re= 700 c) Re= 850 at angle of attack of 15º

It is observed that increasing Reynolds number increases heat transfer coefficient for all the stated cases. Oval tube with 7 vortex generators at Higher Reynolds number shows a higher heat transfer capacity than that of oval tubes at low Reynolds number. Suppose, oval tubes at Re=850 shows a good heat transfer region at the tube portion (the blue color gradually fades away) than that of Re=500 and Re=700.

Figure 51 shows the variation of streamline with Reynolds number. It is seen that increasing Reynolds number increases velocity. Oval tube with Re=850 shows a higher velocity region (indicated by red portion) than that of oval tubes with 700 Reynolds number. Figure 51(c) depicts a lower velocity region. The flow recirculation region appears quite similarly for all the cases.
c) **Streamline Distribution at YZ Plane**

![Streamline distribution for oval tubes at x=89.5mm for 7 vortex generators at a) Re=850 b) Re= 700 c) Re=500 at angle of attack of 15°](image)

**Figure 51** - Streamline distribution for oval tubes at x=89.5mm for 7 vortex generators at a) Re=850 b) Re= 700 c) Re=500 at angle of attack of 15°

**Effect of Reynolds numbers on Variation of Velocity and Temperature Profile**
*(For Oval tube)*

![Effect of Reynolds numbers on Variation of Velocity and Temperature Profile](image)
From Figure 2 it is clearly observed that velocity increases with the increment of Reynolds number and the peak velocity is obtained for Re=850. Nearly zero velocity is investigated at the tube surface while it gradually increases towards the wall of the channel.

From Figure 53 it is seen that temperature variation is clearly evident at the tube surface with the change of Reynolds numbers. The highest possible heat transfer is obtained for Re=850 as suggested by the temperature value of approximately 300K near the tube surface.

Figure 54(a) shows the variation of heat transfer coefficient with Reynolds number. It is seen that increasing Reynolds number increases heat transfer coefficient for all the stated cases. Circular tube with vortex generators shows a higher heat transfer capacity than that of oval tubes but considering the pressure drops from Figure 54(b) the overall performance of oval tube is better than the circular ones. For illustration, oval tubes with 7 vortex generators gives the identical heat transfer to circular tubes of 3 VGs but reduces the pressure drop of approximately 6 Pa at Re=850.
Variation of Heat transfer Coefficient and Pressure drop with Reynolds number

Figure 54- Performance parameter (a) heat transfer coefficient and (b) pressure drop variations with Reynolds number for CFU configuration of VG at 15º angle of attack.

Variation of Heat Transfer Coefficient and Pressure Drop

Figure 55 shows the variation of Pressure drop and heat transfer coefficient with Reynolds number. It is seen that increasing Reynolds number increases pressure drop and heat transfer coefficient for all the stated cases. Circular tube with vortex generators shows a higher heat
transfer capacity than that of oval tubes at higher Aspect ratios but considering the pressure drops, the overall performance of oval tube at AR=1.8,2.34 is better than the circular ones.

*Figure 55- Variation of heat transfer coefficient and pressure drop with Reynolds number for different aspect ratios of oval tubes at 15°angle of attack*
At the same time, it is also observed that, oval tubes with AR=1.8 gives h=43 at a pressure drop of nearly 28 Pa (h/p=1.535) while circular tube shows a h/p=1.136 for the same case.

**Heat Transfer Coefficient and Pressure Drop for Angle of Attack**

As stated earlier, angle of attack has a significant influence on vortices distribution and thereby its impact on heat transfer is so important. It is observed that increasing Reynolds number increases heat transfer coefficient for higher angle of attack for all the stated cases because a large volume of fluid goes through the narrow passage offered by the large angle of attack. Circular tube with angle of attack of 25 degree shows a higher heat transfer capacity than that of oval tubes but considering the pressure drops the overall performance of oval tube is better than the circular ones. Suppose, oval tubes with 15 degree angle of attack gives h=42.5 for pressure drop of 29 Pa (h/p= 1.465) at Re=850. On the other hand, circular tubes with 15 degree angle of attack gives h=50 for pressure drop of 40 Pa (h/p= 1.25) at Re=850.
Figure 56- Effect of Angle of attack on (a) heat transfer coefficient and (b) pressure drop with Reynolds number for 7VGs.

**Variation of Heat Flux per Unit Pressure Drop with Reynolds Number**
Figure 57: Heat flux per unit pressure drop with Reynolds number at 7VGs for 15° angle of attack

Figure 57 shows the variation of performance parameter (heat flux per unit pressure drop) with Reynolds number. It is investigated that increasing Reynolds number decreases the parameter for both circular and oval tubes. Oval tubes show a higher value of 40 than that of circular which shows a value near 34. At the same time, considering pressure drop, the performance factor actually decreases with the increment of Reynolds number.

Friction Factor for Different Number of Vortex Generators

As Reynolds number increases, friction factor gradually decreases for all the stated cases. Higher Reynolds number implies lower viscous force, thereby lower shear stress and hence lower friction factor.

![Friction Factor Graph](image)

Figure 58: Friction factor at various Reynolds number for different number of vortex generators at 15°angle of attack
The overall performance of the fin-and-tube heat exchangers with different tube arrangements are evaluated using the criterion of the area goodness factor $j/f$ as depicted in Figure 59. For the Reynolds number ranging from 500 to 850, for the case of 1-RWP (Rectangular Winglet Pair), the increment of $j/f$ ratio is 27% over 7RWP case. The same causes the increment of the $j/f$ ratio 15% over the 3RWP case. It is interesting to note that the influence of the tube arrangement on the overall performance becomes progressively small with the increasing number of the RWPs. From the viewpoint of “area goodness,” the 1RWP and 3 RWP case will require a smaller frontal area than the other cases and would be more efficient in compact heat exchanger design.
4.3.3 Number of Vortex Generators

Temperature Contours at YZ Plane

Figure 60- Temperature contours for Circular tubes at X=172mm for Re=850 at a) Baseline b) 1VG c) 3VGs d) 7VGs for 15°angle of attack

The temperature contours for different vortex generator are observed in Figure 60 at 850 Reynolds number. The different temperature contour region for baseline case which is quite identical to 1 Vg configuration is depicted in Figure (b). However, the 3 vortex generator shows a good heat transfer causing a small region of incoming fluid (red) and the 7 vg configuration shows the highest possible heat transfer as the red region diminishes totally.

Figure 61 describes the velocity contours for different vortex generator configuration. It shows a plane at the end of the seventh tube and indicates a little differences for (a)(b)(c) but a higher velocity region is seen near the tube for 7 vortex generator configuration. So, this fluid has again a good heat transfer capability.
Figure 61- Velocity contours for circular tubes at Reynolds number = 850 and at a distance x=172mm for a) Baseline b) 1 VG c) 3 VG d) 7 VG at 15° angle of attack

Comparison of Temperature Contours at Exit Plane
The contour plot of temperature for different vortex generators indicates the heat transfer zone. It is seen that increasing vortex generator number increases heat transfer coefficient for all the stated cases. Circular tube with vortex generators shows a higher heat transfer capacity than that of oval tubes. As the tube is circular, the contact area between mixing fluid increases, thereby increases heat transfer coefficient and pressure drop. Suppose, oval tubes with 7 vortex generators gives a small region of blue temperature color while in case of circular tube the region diminishes. At the same time, the red region persists indicating lower heat transfer.

### a) Temperature Contours at a Plane Just Past Two Tubes from the Location of VG

- **a.**
- **b.**
- **c.**
- **d.**

The after effect of location of VGs has a significant influence on thermal characteristics of heat exchanger. The variation of temperature contours for different vortex generators are observed in the above figure. It is seen that increasing vg number increases heat transfer coefficient for all the stated cases. Circular tube with 7 vortex generators shows a higher heat transfer capacity than that baseline, 1vg and 3 vg cases. For baseline and 1 vg the temperature
profile is near similar but 3, 7 vg indicates that the after effect of vortex generator is significant. The 3 and 7 vortex generator indicates a better heat transfer zone.

**Variation of Velocity and Temperature Profile (For Circular tubes) at Re= 850**

![Graph showing velocity and temperature profiles](image)

*Figure 64- Velocity and Temperature distribution along the midline of YZ plane at a distance X=98mm for circular tube at different number of vortex generators.*

The velocity drops linearly to a lower value for 7VGs than others due to large narrow passage provided by the VGs. Again velocity shows a higher peak close to the tube surface. From Figure, it is observed that temperature drops linearly up to Y=.007m from both sides of the wall streams and shows a nearly constant values near the tube surface due to intense mixing.
between the fluids. 3VGs shows a suddenly higher stable value (300K) than 1Vg and the 7VGs gives a good increment over the previous two.

Percentage Variation of Heat Transfer Coefficient and Pressure Drop with Number of Vortex Generators

![Graph showing percentage variation of heat transfer coefficient and pressure drop with number of vortex generators.](graph.png)
Figure 65- Percentage variation of heat transfer coefficient and pressure drop with Number of vortex generators.

Figure 65(a) (b) shows the variation of percentage increment of heat transfer and pressure drop with number of vortex generators for both circular and oval tubes. Around 170% pressure drop is observed for 7Vgs at Re=850 and it gives a 48 percent increment of heat transfer coefficient. On the other hand, oval tubes gives 26% increment of heat transfer for the penalty of nearly 80% pressure drop.

4.3.4 Effect of Tube shape (Circular/Oval)

Comparison of Velocity at YZ plane

Figure 66- Velocity contours for circular tubes at Reynolds number =850 for 3 vortex generator configuration at defined plane for 15°angle of attack.
Form Figure 66, it is observed that, before passing the tubes and vg the velocity contours at the side of the channel shows a small yellow region. In Figure 66(plane-2) the region changes to dark red which shows a higher velocity band than the previous due to the extra area contraction provided by both tube and VG wall. After passing the tube (plane-3), the red region gradually fades away and a higher yellow region is seen and at the end, at a distance from the tube the velocity reduces to some extent. On the other hand, in case of oval tube, the overall view is nearly same but region occupies a smaller velocity than that of circular tubes.

**Comparison of Temperature Contours at YZ Plane**

*Figure 67 - Temperature contours for circular tubes at Reynolds number =850 for 3 vortex generator configuration at defined plane for 15° angle of attack.*

Form Figure 67, it is observed that, before passing the tubes and vg the temperature contours at the middle side of the channel shows that bulk fluid temperature decreases to near 303K. In Figure 67(plane-2) the heat transfer zone is seen where fluid passes the tube increasing the temperature of the fluid inside the tube. After passing the tube (plane-3), the greater green region is observed that determines the average bulk fluid temperature. But in case of oval tube, the overall view is nearly same.
Figure 68: Temperature contours for Oval tubes at Reynolds number =850 for 3 vortex generator configuration at defined plane for 15°angle of attack.

A small heat transfer region due to the shape of the tube is observed in Figure 68. As a result, a small reduced temperature region (blue) is seen at the center of plane-4. But the range of the heat transfer area is smaller in oval tubes than circular one. As a result, circular tube gives higher heat transfer coefficient.

Velocity Vectors at YZ Plane
**Figure 69** - Velocity vectors for oval tubes at Reynolds number = 850 for 7 vortex generator configuration at defined plane for 15° angle of attack.

In Figure 69, velocity vectors are plotted for mentioned cases. At plane-1, a small heat transfer zone is seen at the middle of the channel. Then in plane-2, vortex generator appears and that creates swirling motion of the flow inducing minor loss but a significant flow recirculation region is noted at plane-3 which can again make an effective heat transfer.

![Velocity vectors](image)

**Figure 70** - Temperature contours for oval tubes at Reynolds number = 850 for 7 vortex generator configuration at defined plane for 15° angle of attack.

In Figure 70, Temperature vectors are plotted for mentioned cases. At plane-1, a small heat transfer zone is seen at the middle of the channel. Then in plane-2, vortex generator appears and that creates swirling motion of the flow inducing minor loss but a significant flow recirculation region is noted at plane-3 which can again make an effective heat transfer causing a wider band of heat transfer region at the rear plane of the channel.
4.3.5 Effect of Aspect Ratio of Oval Tubes

Variation of Velocity Profile at Re= 850

As aspect ratios change, the variation of velocity are nearly negligible due to closely equal flow passage is provided by all the oval tube. Line averaged results are taken along the midline of the defined plane.

![Diagram of Oval Tubes and Velocity Profiling](image)

**Figure 71**- Velocity distribution along the midline of YZ plane at a distance X=98mm for circular tube at different aspect of ratios

**Temperature Contours at Exit plane**

a. 

b. 

c.
Figure 72- Temperature contours for a) circular tubes and b) oval tubes at X=177.8mm for Re=850 at AR=1.24, AR=1.54, AR=1.8, AR=2.32.

Figure 72 shows the variation of temperature contours at Re=850 for different aspect ratios of the oval tubes. It is seen that increasing aspect ratios decreases heat transfer coefficient. Circular tube with vortex generators shows a higher heat transfer capacity than that of oval tubes but the red region at the two end of the channel increases as AR of oval tube increases indicating poor heat transfer region.

Streamline Distribution
Figure 73- Streamline for a) circular tubes and oval tubes at b) AR=1.24, c) AR=1.54, d) AR=1.8, e) AR=2.34.

The distributions of streamline at the rear end of the tube for both circular and oval tube are presented in Figure 73. It is observed that a higher velocity region is seen near vg than that of Figure 73(b). At the same time, increasing AR of oval tubes decreases the recirculation zone past the tube. Thereby actually it decreases the minor loss or pressure drop.
CHAPTER FIVE

CONCLUSION

The present study investigates the effect of vortex generators for flow through a compact heat exchanger, using different geometric configurations of winglets and tubes. The Reynolds number, tube shape, aspect ratios of oval tubes, number of winglets, angle of attack of winglet are varied. A numerical investigation is performed to find the best possible heat transfer enhancement technique. The study has indicated that, vortex generators enhances heat transfer rate at a cost of increased pumping power. However, the obtained results are listed here:

1. The VGs enhance the heat transfer on the tube bank. Scanty difference in enhancement of heat transfer obtained by varying the Reynolds number and angle of attacks of VGs, where the study shows that heat transfer (Nu) and pressure drop (ΔP) increases with increase in Reynolds number and angle of attack.

2. Comparing with the baseline case, the heat transfer coefficient of the fin-and-tube heat exchanger is improved by 38.1–43.6%, 75-80.5%, 58.5-84.1% for the inline-3RWP case, the inline-7RWP case, and the inline-7RWP case (oval tube banks), respectively. The corresponding pressure drop penalty is also increased. The enhancement in heat transfer coefficient outweighs the additional pressure drop penalty generated by RWPs.

3. In 2-D, the best results are obtained with oval tube banks at an angle of attack of 45° and Reynolds number of 1775.

4. In 3-D, the heat transfer enhancement for both circular and oval tubes occurs with the increment of Reynolds number from 500 to 850. In case of circular tube (7VG), the enhancement of heat transfer coefficient is nearly 45% in penalty of 127% pressure drop.

5. Circular tubes with 7VG offers nearly 49% heat transfer enhancement with respect to oval tubes (baseline) while oval tubes with 7VG gives 27.50% increment. So, number of vortex generators significantly increases the overall heat transfer coefficient as well as the increment is greatly influenced by tube shape.
6. Circular tube with 25 degree angle of attack offers a 27% heat transfer increment over oval tubes with 15 degree of angle of attack. However, for a particular tube shape (circular or oval) the enhancement of heat transfer is approximately 12.5% due to the changes of angle of attack from 15 to 25, the oval tubes shows only 40% pressure drop while it is 62% for circular tubes.

7. The effect of aspect ratio is very predominant for enhancing heat transfer. At reduced AR of the oval tubes, both heat transfer and pressure drop increases due to large frontal area is supposed by the tube minor diameter. Aspect ratio of 1.24 gives a remarkable heat transfer enhancement of 26.5% at a mere 69% increase in pumping power than AR=2.34.

8. As Reynolds number increases, friction factor gradually decreases for all the stated cases. In case of 7VG, the friction factor decreases around 33% for the increment of Reynolds number from 500 to 850.

9. For the Reynolds number ranging from 500 to 850, for the case of 1-RWP (Rectangular Winglet Pair), the increment of $j/f$ ratio is 27% over 7RWP case. The same causes the increment of the $j/f$ ratio 15% over the 3RWP case. As number of VG increases, the factor reduces to ~0.18 from ~0.23. It is interesting to note that the influence of the tube arrangement on the overall performance becomes progressively small with the increasing number of the RWPs. From the viewpoint of “area goodness,” the 1RWP and 3 RWP case will require a smaller frontal area than the other cases and would be more efficient in compact heat exchanger design.

From the results above it might be concluded that increasing number of winglets, winglets with higher angle of attack, oval tubes with lower aspect ratio, higher Reynolds number of flow is preferable for high heat transfer enhancement with minimum pressure drop.
CHAPTER SIX

FUTURE RECOMMENDATION

The present study uses rectangular winglets in common flow up configurations. However, the following criteria could be considered for the future research:

It is also possible to conduct these simulations with delta winglet vortex generators and in common flow down configurations.

Due to the continued development of moving mesh technique, semi-passive heat transfer enhancement techniques would deserve more attentions in near future. The vortex generator will be subject to forced vibration due to the fluid pressure, which will further increase the disturbance of the flow field.

A configuration of delta and rectangular winglet (composite winglets) may also be a good consideration for future works.
REFERENCES


APPENDIX

Bar charts for overall results

![Bar chart 1](chart1.png)

![Bar chart 2](chart2.png)
Percentage increment of Pressure drop with respect to 15 degree angle

AR=1.24

AR=2.34

Percentage increment of Heat transfer with respect to AR of Oval tubes
Pressure Distribution

Pressure contour on XY plane at a distance Z=1.5mm from the bottom plane for oval tube (AR=1.80) at Re=850 and 7RWPs configuration.
NUMERICAL STUDY OF FORCED CONVECTION HEAT TRANSFER OVER CIRCULAR AND OVAL TUBE BANKS USING RECTANGULAR WINGLET VORTEX GENERATORS

Md. R. Haque¹, Md. A. Rahman¹∗

¹Bangladesh University of Engineering and Technology (BUET), Dhaka-1000, Bangladesh

ABSTRACT

The present work represents a two-dimensional numerical investigation of forced convection heat transfer over circular and oval tube banks of seven rows in an inline arrangement with rectangular longitudinal vortex generators (LVGs) placed in common flow up configuration. The effects of Reynolds number (from 550 to 1775), the number of vortex generators (no generators, 3, and 7) and the angle of attack (30° and 45°) of rectangular VGs on the heat transfer and fluid flow characteristics are examined. The characteristics of average Nusselt number (Nu), associated pressure drop (∆P), streamline distribution and temperature contours are studied numerically by the aid of the computational fluid dynamics (CFD). The maximum heat transfer coefficient (nearly 75 Wm⁻²K⁻¹) is achieved for 7VGs and 45 degree of angle of attack configuration compared to the baseline case (nearly 18 Wm⁻²K⁻¹) for circular tube banks, having a pressure drop penalty of nearly 210 Pa. The streamline plot shows an excellent recirculation region near the VG’s position indicating that the addition of vortex generators intensifies the mixing between the hot and cold fluids and thereby enhances heat transfer significantly. The performance of the oval and circular tube banks with vortex generators under the same operating conditions is compared with a focus on finding the optimized configuration of vortex generators.
KEY WORDS: Computational Fluid dynamics, Compact heat exchanger, Vortex generators, Heat transfer enhancement, Tube banks