Investigation of an Analytical Approach for the Design of the Convection Section of Fired Heaters

The Opportunity to Succeed 20th Century Correlations

J. van Dijk
Investigation of an Analytical Approach for the Design of the Convection Section of Fired Heaters
The Opportunity to Succeed 20th Century Correlations

by

J. van Dijk

For the degree of Master of Science in Mechanical Engineering at Delft University of Technology

To be defended publicly on January 30th, 2020 at 09:30

Student Number 4218450
Report Number P&E 3024
Project Duration January 24th, 2019 - January 30th, 2020
Thesis Committee Prof.dr. D.J.E.M. Roekaerts TU Delft, 3mE, EPT, Chairman
Dr.ir. C.A. Infante Ferreira TU Delft, 3mE, EPT, Associate Professor
Dr.ir. J.W.R. Peeters TU Delft, 3mE, EPT, Assistant Professor
M. Janeschitz-Kriegl FLUOR, Guest

Faculty of Mechanical, Maritime and Materials Engineering (3mE) · Delft University of Technology
The contents of this master thesis report was supported by FLUOR Corporation. Their help and effort is hereby gratefully acknowledged. Special thanks to Michael, who was always willing and able to answer my questions.
Abstract

The convection section of fired heaters consists of finned tubes to enhance heat transfer. However, finned tubes make heat transfer in the convection section of fired heaters considerably more complex. Consequently, engineers depend heavily on correlations to determine heat transfer in the convection section of fired heaters. The problem related to the use of these correlations is twofold. First, these correlations have been developed approximately forty years ago by performing wind tunnel experiments for 1" tubes and 2" tubes, after which the results have been extrapolated to accommodate for 4" tubes. Second, industry developments led to the use of 6" tubes or even 8" tubes in the convection section of fired heaters. As a result, it should be questioned if correlations still provide accurate results for these tube sizes.

This study aims to validate an entirely new way of calculating heat transfer and pressure drop in the convection section of fired heaters proposed by a recently developed analytical method. In contrast to correlations, the analytical method provides insight in the working principle of fins, is in theory applicable for large tube sizes, and ultimately has the potential to design convection banks more accurately. The latter would result in a reduction of fuel consumption, and with the focus on reducing emissions to reach climate goals, this would mean a significant improvement over existing correlation methods. Numerical models based on the realizable k-ε turbulence model have been developed in ANSYS Fluent to be able to validate the analytical method. The values of the heat transfer and pressure drop coefficient relative to the bare tube area have been compared with the values calculated by the analytical method and the ESCOA correlation method for one tube row and for multiple tube rows to see which method resembles the results calculated by the numerical models the closest.
# Table of Contents

Abstract i

1 Introduction 1
  1-1 Motivation .......................................................... 1
  1-2 Research Questions ............................................... 3
  1-3 Research Approach ............................................... 4
  1-4 Research Scope .................................................. 5
  1-5 Report Outline ................................................... 5

2 Theory Related to Finned Tube Heat Transfer 7
  2-1 Introduction ...................................................... 7
  2-2 Working Principle of Fired Heaters ............................... 7
    2-2-1 Introduction .................................................. 7
    2-2-2 Convection Section .......................................... 9
  2-3 ESCOA Correlation Method ........................................ 12
    2-3-1 Introduction .................................................. 12
    2-3-2 ESCOA Correlations ........................................... 13
  2-4 Analytical Method ............................................... 17
    2-4-1 Introduction .................................................. 17
    2-4-2 Derivation of the Analytical Method ........................... 18
  2-5 Numerical Methods Applied to Finned Tubes ................. 25
    2-5-1 Introduction .................................................. 25
    2-5-2 Brief Introduction to Numerical Methods ..................... 25
    2-5-3 Mesh .......................................................... 27
    2-5-4 Turbulence Models .......................................... 28
    2-5-5 Wall Treatment ............................................... 30
    2-5-6 Boundary Conditions ....................................... 33

Thesis Report J. van Dijk
# Table of Contents

3 Research Methodology .................................................. 35
   3-1 Introduction .......................................................... 35
   3-2 Methodology Applied to Numerical Models ...................... 36
      3-2-1 Geometry ....................................................... 36
      3-2-2 Mesh ........................................................... 40
      3-2-3 Setup .......................................................... 48

4 Results and Discussion .................................................. 59
   4-1 Introduction .......................................................... 59
   4-2 Results for the Investigation of One Tube Row ............... 60
      4-2-1 Heat Transfer Coefficient Comparison for 4", 6", and 8" Tubes ..... 60
      4-2-2 Pressure Drop Coefficient Comparison for 4", 6", and 8" Tubes ..... 69
   4-3 Results for the Investigation of Multiple Tube Rows ......... 76
      4-3-1 Heat Transfer Coefficient Comparison for 4" Tubes .......... 76
      4-3-2 Heat Transfer Coefficient Comparison for 6" Tubes .......... 79
   4-4 Results for the Investigation of Two Fin Configurations .... 81
      4-4-1 Heat Transfer Coefficient Comparison for 6" Tubes .......... 81
      4-4-2 Heat Transfer Coefficient Comparison for 8" Tubes .......... 83

5 Conclusion ................................................................. 85

6 Recommendations .......................................................... 87

A Post-Processing Figures Related to the Investigation of One Tube Row 89
   A-1 Velocity Isosurfaces .............................................. 90
   A-2 XY-Plots ............................................................. 93
   A-3 Temperature Isosurfaces ......................................... 95
   A-4 Finned Tube Temperature Contours ............................. 98
   A-5 Surface Heat Transfer Coefficient Contours .................... 104

B Post-Processing Figures Related to the Investigation of Multiple Tube Rows 111
   B-1 Velocity Isosurfaces .............................................. 112
   B-2 Velocity Profiles ................................................ 114
   B-3 Temperature Profiles ............................................ 115

C Post-Processing Figures Related to the Investigation of Two Fin Configurations 117
   C-1 Temperature Isosurfaces ......................................... 118

Bibliography ................................................................. 123

Glossary .......................................................... 127
   List of Acronyms ..................................................... 127
   List of Symbols ....................................................... 128

J. van Dijk
Thesis Report
List of Figures

1-1 Simple representation of a fired heater (image from Jegla et al. (2015)). Left: Cabin heater with horizontal tube coil. Right: Cylindrical fired heater with vertical coil. ................................................................. 2

1-2 Visual representation of the established research approach. .............................................. 4

2-1 Schematic of a cabin-type fired heater (image from API Standard 560 (American Petroleum Institute, 2001))................................................................. 8

2-2 Two types of commonly used fins. (a): Spiral or solid annular fins (image from Frass (2015)). (b): Serrated or segmented fins (image from ESCOA Corporation (1979)). ................................................................. 10

2-3 Tube arrangements for the convection section of fired heaters. (a): Staggered tube arrangement. (b): In-line tube arrangement (images from Frass (2015)). ............ 12

2-4 Schematic of a solid annular finned tube including relevant fin parameters. (image from Frass (2015))................................................................. 14

2-5 Heat transfer coefficient relative to the bare tube area $U_{bare}$ versus the fin density $N_f$ for various bare tube diameters calculated by the ESCOA correlation method. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2 \text{ kg/m}^2\cdot\text{s}$......................... 15

2-6 Pressure drop coefficient relative to the bare tube area $\zeta$ versus the fin density $N_f$ for various bare tube diameters calculated by the ESCOA correlation method. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2 \text{ kg/m}^2\cdot\text{s}$......................... 17

2-7 Schematic of the flow path length $L_{flow,\text{fin}}$ depicted by the curved dark blue arrow. 19

2-8 Heat transfer coefficient relative to the bare tube area $U_{bare}$ versus the fin density $N_f$ for various bare tube diameters calculated by the analytical method. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2 \text{ kg/m}^2\cdot\text{s}$......................... 22

2-9 Pressure drop coefficient relative to the bare tube area $\zeta$ versus the fin density $N_f$ for various bare tube diameters calculated by the analytical method. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2 \text{ kg/m}^2\cdot\text{s}$......................... 23

2-10 Example of a hybrid mesh, using both a structured grid in the vicinity of the cylinder walls, and an unstructured grid elsewhere (image from Versteeg and Malalasekera (2007)).......................... 27
2-11 Complex flow behavior caused by solid annular-finned tubes (image from Sung et al. (1996)). ................................................................. 28
2-12 Law of the wall (image from Versteeg and Malalasekera (2007)). N.B.: log $y^+$ on the x-axis represents the base logarithm, not the natural logarithm. .......... 31
2-13 Near-wall model approach to resolve the boundary layer (image from Murad (2018)). 32
2-14 Wall function approach to resolve the boundary layer (image from Murad (2018)). 32

3-1 Side view of the adopted computational domain (image from Mon and Gross (2004)). 36
3-2 Top view of the adopted computational domain (image from Mon and Gross (2004)). 36
3-3 Side-view of the proposed computational domain related to the investigation of one tube row. ................................................................. 37
3-4 Top-view of the proposed computational domain related to the investigation of one tube row. ................................................................. 37
3-5 Side-view of the proposed computational domain related to the investigation of multiple tube rows. ....................................................... 39
3-6 Top-view of the proposed computational domain related to the investigation of multiple tube rows. ....................................................... 39
3-7 Side-view of the computational domain with the corresponding name selections. . 41
3-8 Top-view of the computational domain with the corresponding name selections. . 41
3-9 Representation of the generated mesh for the fin and tube region. (a): Side-view. (b): Front-view. ................................................................. 43
3-10 Side-view representation of the generated mesh for the fin fluid domain. .......... 45
3-11 Side-view representation of the generated mesh for the outside fin fluid domain. 46
3-12 Side-view representation of the generated mesh for the downstream fluid domain. 47
3-13 Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ versus the fin density $N_f$ for 4" tubes, calculated by the ESCOA correlation method and by the realizable k-ε and Transition SST models on a coarse grid. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$s. ................................................................. 51
3-14 Comparison of the pressure drop coefficient relative to the bare tube area $\zeta$ versus the fin density $N_f$ for 4" tubes, calculated by the ESCOA correlation method and by realizable k-ε and Transition SST models on a coarse grid. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$s. ................................................................. 51
3-15 Cross-section of the finned tube geometry. $A_{flow,tot}$ is depicted by the white area. 53

4-1 Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ calculated by various methods versus the fin density $N_f$ for 4" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$s. ................................................................. 60
4-2 Wake region behind the tube depicted by the diagonally striped region. .......... 61
4-3 Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ calculated by various methods versus the fin density $N_f$ for 6" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$s. ................................................................. 65
4-4 Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ calculated by various methods versus the fin density $N_f$ for 8" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$s. ................................................................. 68
4-5 Comparison of the pressure drop coefficient relative to the bare tube area $\zeta$ calculated by various methods versus the fin density $N_f$ for 4" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 70

4-6 Locations where the pressure and density have been calculated. These locations are depicted by the red dots. .......................................................... 71

4-7 Comparison of the pressure drop coefficient relative to the bare tube area $\zeta$ calculated by various methods versus the fin density $N_f$ for 6" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 73

4-8 Comparison of the pressure drop coefficient relative to the bare tube area $\zeta$ calculated by various methods versus the fin density $N_f$ for 8" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 75

4-9 Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ calculated by various methods versus the number of tube rows $N_R$ for 4" tubes. Fin density $N_f = 4$ inch$^{-1}$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 77

4-10 Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ calculated by various methods versus the number of tube rows $N_R$ for 6" tubes. Fin density $N_f = 4$ inch$^{-1}$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 80

4-11 Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ between two different fin configurations calculated by various methods for 6" tubes. $N_R = 1$. .......................................................... 82

4-12 Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ between two different fin configurations calculated by various methods for 8" tubes. $N_R = 1$. .......................................................... 84

A-1 Velocity isosurfaces for 4" tubes. Left: $N_f = 2$ inch$^{-1}$. Center: $N_f = 4$ inch$^{-1}$, Right: $N_f = 8$ inch$^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 90

A-2 Velocity isosurfaces for 6" tubes. Left: $N_f = 2$ inch$^{-1}$. Center: $N_f = 4$ inch$^{-1}$, Right: $N_f = 8$ inch$^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 91

A-3 Velocity isosurfaces for 8" tubes. Left: $N_f = 2$ inch$^{-1}$. Center: $N_f = 4$ inch$^{-1}$, Right: $N_f = 8$ inch$^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 92

A-4 XY-plots illustrating the velocity $u$ profile between the fins at half the fin height $h$ for various fin density values for 4" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 93

A-5 XY-plots illustrating the velocity $u$ profile between the fins at half the fin height $h$ for various fin density values for 6" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 93

A-6 XY-plots illustrating the velocity $u$ profile between the fins at half the fin height $h$ for various fin density values for 8" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 94

A-7 Temperature isosurfaces for 4" tubes. Left: $N_f = 2$ inch$^{-1}$. Center: $N_f = 4$ inch$^{-1}$. Right: $N_f = 8$ inch$^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. ........................................ 95
A-8 Temperature isosurfaces for 6'' tubes. Left: $N_f = 2$ inch$^{-1}$. Center: $N_f = 4$ inch$^{-1}$. Right: $N_f = 8$ inch$^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. .................................................. 96

A-9 Temperature isosurfaces for 8'' tubes. Left: $N_f = 2$ inch$^{-1}$. Center: $N_f = 4$ inch$^{-1}$. Right: $N_f = 8$ inch$^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. .................................................. 97

A-10 Finned tube temperature contour for 4'' tubes and $N_f = 2$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 98

A-11 Finned tube temperature contour for 4'' tubes and $N_f = 4$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 98

A-12 Finned tube temperature contour for 4'' tubes and $N_f = 8$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 99

A-13 Finned tube temperature contour for 6'' tubes and $N_f = 2$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 100

A-14 Finned tube temperature contour for 6'' tubes and $N_f = 4$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 100

A-15 Finned tube temperature contour for 6'' tubes and $N_f = 8$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 101

A-16 Finned tube temperature contour for 8'' tubes and $N_f = 2$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 102

A-17 Finned tube temperature contour for 8'' tubes and $N_f = 4$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 102

A-18 Finned tube temperature contour for 8'' tubes and $N_f = 8$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 103

A-19 Surface heat transfer coefficient contour for 4'' tubes and $N_f = 2$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 104

A-20 Surface heat transfer coefficient contour for 4'' tubes and $N_f = 4$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 104

A-21 Surface heat transfer coefficient contour for 4'' tubes and $N_f = 8$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 105

A-22 Surface heat transfer coefficient contour for 6'' tubes and $N_f = 2$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right. ............................................................ 106
A-23 Surface heat transfer coefficient contour for 6\(^\text{th}\) tubes and \(N_f = 4\ \text{in}^{-1}\). Number of tube rows \(N_R = 1\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). Flow direction is from left to right. 

A-24 Surface heat transfer coefficient contour for 6\(^\text{th}\) tubes and \(N_f = 8\ \text{in}^{-1}\). Number of tube rows \(N_R = 1\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). Flow direction is from left to right. 

A-25 Surface heat transfer coefficient contour for 8\(^\text{th}\) tubes and \(N_f = 2\ \text{in}^{-1}\). Number of tube rows \(N_R = 1\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). Flow direction is from left to right. 

A-26 Surface heat transfer coefficient contour for 8\(^\text{th}\) tubes and \(N_f = 4\ \text{in}^{-1}\). Number of tube rows \(N_R = 1\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). Flow direction is from left to right. 

A-27 Surface heat transfer coefficient contour for 8\(^\text{th}\) tubes and \(N_f = 8\ \text{in}^{-1}\). Number of tube rows \(N_R = 1\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). Flow direction is from left to right. 

B-1 Velocity isosurfaces for 4\(^\text{th}\) tubes. From left to right: first tube row, second tube row, third tube row, fourth tube row. \(N_f = 4\ \text{in}^{-1}\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). 

B-2 Velocity isosurfaces for 6\(^\text{th}\) tubes. From left to right: first tube row, second tube row, third tube row, fourth tube row. \(N_f = 4\ \text{in}^{-1}\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). 

B-3 Velocity profiles for 4\(^\text{th}\) tubes. Top: Velocity profile over the tube. Bottom: Velocity profile over the fin. \(N_f = 4\ \text{in}^{-1}\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). 

B-4 Velocity profiles for 6\(^\text{th}\) tubes. Top: Velocity profile over the tube. Bottom: Velocity profile over the fin. \(N_f = 4\ \text{in}^{-1}\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). 

B-5 Temperature profiles for 4\(^\text{th}\) tubes. Top: temperature profile over the tube. Bottom: Temperature profile over the fin. \(N_f = 4\ \text{in}^{-1}\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). 

B-6 Temperature profiles for 6\(^\text{th}\) tubes. Top: Temperature profile over the tube. Bottom: Temperature profile over the fin. \(N_f = 4\ \text{in}^{-1}\), average mass flux \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). 

C-1 Temperature isosurfaces for configuration A (\(N_f = 5\ \text{in}^{-1}\), \(h = 1^\circ\)). Left: \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). Center: \(G_{avg} = 4\ \text{kg/m}^2\cdot\text{s}\). Right: \(G_{avg} = 6\ \text{kg/m}^2\cdot\text{s}\). 

C-2 Temperature isosurfaces for configuration B (\(N_f = 2.5\ \text{in}^{-1}\), \(h = 2^\circ\)). Left: \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). Center: \(G_{avg} = 4\ \text{kg/m}^2\cdot\text{s}\). Right: \(G_{avg} = 6\ \text{kg/m}^2\cdot\text{s}\). 

C-3 Temperature isosurfaces for configuration A (\(N_f = 5\ \text{in}^{-1}\), \(h = 1^\circ\)). Left: \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). Center: \(G_{avg} = 4\ \text{kg/m}^2\cdot\text{s}\). Right: \(G_{avg} = 6\ \text{kg/m}^2\cdot\text{s}\). 

C-4 Temperature isosurfaces for configuration B (\(N_f = 2.5\ \text{in}^{-1}\), \(h = 2^\circ\)). Left: \(G_{avg} = 2\ \text{kg/m}^2\cdot\text{s}\). Center: \(G_{avg} = 4\ \text{kg/m}^2\cdot\text{s}\). Right: \(G_{avg} = 6\ \text{kg/m}^2\cdot\text{s}\).
# List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-1</td>
<td>Operating conditions used to obtain heat transfer and pressure drop results.</td>
</tr>
<tr>
<td>2-2</td>
<td>Bare tube diameters with corresponding transverse and longitudinal tube pitch. Left: 4&quot; tubes. Center: 6&quot; tubes. Right: 8&quot; tubes.</td>
</tr>
<tr>
<td>3-1</td>
<td>Dimensions used to define the geometry of interest related to the investigation of one tube row. Left: 4&quot; tubes. Center: 6&quot; tubes. Right: 8&quot; tubes.</td>
</tr>
<tr>
<td>3-2</td>
<td>Thickness of the computational domain ( Z_1 ) as a function of fin density ( N_f ).</td>
</tr>
<tr>
<td>3-3</td>
<td>Meshing method for the fin and tube region for a medium grid, 4&quot; tube and fin density ( N_f = 4 ) inch(^{-1}).</td>
</tr>
<tr>
<td>3-4</td>
<td>Features corresponding to the multizone meshing method which have been changed from the default settings.</td>
</tr>
<tr>
<td>3-5</td>
<td>Meshing method for the fin fluid domain for a medium grid, 4&quot; tube and fin density ( N_f = 4 ) inch(^{-1}).</td>
</tr>
<tr>
<td>3-6</td>
<td>Method for the generation of inflation layers for the fin fluid domain for a medium grid, 4&quot; tube and fin density ( N_f = 4 ) inch(^{-1}). Top: Inflation settings for the tube surface. Middle: Inflation settings for the fin side surface. Bottom: Inflation settings for the fin tip surface.</td>
</tr>
<tr>
<td>3-7</td>
<td>Meshing method for the outside fin fluid domain for a medium grid, 4&quot; tube and fin density ( N_f = 4 ) inch(^{-1}).</td>
</tr>
<tr>
<td>3-8</td>
<td>Meshing method for the downstream fluid domain for a medium grid, 4&quot; tube and fin density ( N_f = 4 ) inch(^{-1}).</td>
</tr>
<tr>
<td>3-9</td>
<td>Mesh quality parameters obtained for the final integrated for a medium grid, 4&quot; tube and fin density ( N_f = 4 ) inch(^{-1}).</td>
</tr>
<tr>
<td>3-10</td>
<td>Additional settings related to transient modeling.</td>
</tr>
<tr>
<td>3-11</td>
<td>Material properties used as input variables in the materials section in ANSYS Fluent.</td>
</tr>
<tr>
<td>3-12</td>
<td>Values of ( T_{tube} ) and ( T_{in} ) imposed on the tube surface and inlet of the computational domain respectively.</td>
</tr>
<tr>
<td>3-13</td>
<td>Spatial discretization schemes used for all numerical models.</td>
</tr>
<tr>
<td>3-14</td>
<td>Absolute convergence criteria.</td>
</tr>
</tbody>
</table>
4-1  Average velocity $u$ between the fins at half the fin height $h$ calculated by the initial analytical method, improved analytical method, and numerical models for 4" tubes.  63

4-2  Average velocity $u$ between the fins at half the fin height $h$ calculated by the initial analytical method, improved analytical method, and numerical models for 6" tubes.  66

4-3  Average velocity $u$ between the fins at half the fin height $h$ calculated by the initial analytical method, improved analytical method, and numerical models for 8" tubes.  69
“Projects we have completed demonstrate what we know - future projects decide what we will learn.”
— Dr. Mohsin Tiwana
Chapter 1

Introduction

1-1 Motivation

Ever since the origination of the refining and petrochemical industries, fired heaters have been used to provide the process heat necessary. Therefore, fired heaters are considered to be essential pieces of equipment in the design of process plants.

The design of fired heaters is robust, and their working principle has not changed much over the years. Fired heaters provide heat to process fluids by the combustion of fuels. These process fluids flow in tubes inside the fired heater. This is shown in Figure 1-1, where the two most common types of fired heaters are depicted. Depending on the application many other different types and configurations are possible, which are documented in API Standard 560 written by the American Petroleum Institute (API) (American Petroleum Institute, 2001). In short, a fired heater can be split into two main parts. The first part is the radiant section, where the process fluid is heated primarily by means of radiation due to the heat of combustion of fuels. The second part is the convection section, where hot flue gases from the radiation section are led through a tube bank to heat the process fluid by means of convection.

Heat transfer in the convection section of a fired heater is increased by extending the heat transfer area through the addition of fins. Fins prove to be the most useful when the convective heat transfer coefficient $h_c$ is low, which is often the case for gases such as air and flue gas (Mills, 1999). These fins, either serrated (toothed) or solid annular fins, are wound and welded as a fin strip around the tube in a helical manner.

A consequence of the application of fins is that it makes heat transfer in the convection section of a fired heater considerably more complex. Despite its relatively simple geometry, studies have shown that crossflow around finned tubes causes extremely complex three-dimensional flow characteristics (Sung et al., 1996), (Watel et al., 1999), (Şahin et al., 2006). Due to this complexity, various correlations have been developed during the 20th century to estimate heat transfer of finned tubes (Frass, 2015). These correlations are based on the Reynolds number based on the outer tube diameter, and have been determined by testing various tube bank arrangements in wind tunnel experiments. Still to this day, engineers depend heavily
on correlations for the design of the convection section of fired heaters. However, the use of correlations could give rise to complications. First, wind tunnel experiments have been conducted using only small tubes ranging from less than 1” to 2” in diameter (occasionally measurements using 4’ tubes have been encountered) (Aoki and Taborek, 1972). The results were fitted and extrapolated to produce correlations applicable for tubes sizes ranging up to 4’ tubes. Although these correlations have been proven to be accurate for these tube sizes, they do not have the ability to provide insight in how fins actually work. Second, industry developments led to the use of larger tubes in the convection section of fired heaters, ranging up to 6’ or even 8’ in diameter. For these tube sizes wind tunnel experiments have not been conducted yet and existing correlations were not designed for tubes larger than 4’ in mind. Therefore, it should be questioned if the results produced by these correlations are still accurate.

A recent study introduced a method which calculates heat transfer of finned tubes analytically (Janeschitz-Kriegl, 2017). In short, this analytical method can be divided into main principles. First the fraction of the total mass flow which bypasses the fins and the fraction of the total mass flow which flows between the fins is determined. Only the latter participates in subsequent heat transfer calculations, which is calculated by application of the ϵ-NTU method. In contrast to the correlation methods mentioned previously, the analytical method provides proper understanding to the engineer regarding the underlying physics of finned tube heat transfer due to its relatively simple derivation. In theory, the analytical model should also be capable of producing more realistic results for tube sizes greater than 4’ tubes, meeting the requirements imposed by industry developments.
Related to the analytical method, only a proof of concept calculation has been made yet. If this method turns out to be valid, the convection section of fired heaters could be designed with increased accuracy, which would decrease fuel burn and therefore reduce emissions. Nowadays, with the focus on reducing emissions to reach climate goals, this would be a significant improvement over the current correlation methods. For this reason, the goal of this master thesis is to develop and evaluate a numerical model which is able to validate the analytical method. Computational Fluid Dynamics (CFD) will be used for the numerical model to simulate the flow over finned tubes and to obtain the corresponding flow and heat transfer characteristics.

1-2 Research Questions

To define research problems and research questions, a method which provides structure and good scaffolding for a research problem has been adopted (Klopper and Lubbe, 2011). In this method the researcher will define a general problem, from which a number of sub-problems are extracted. Next, for each sub-problem a research question is defined in such manner that there is a proper alignment between sub-problems and research questions. If done correctly, the answer to a research question should provide the solution to the corresponding sub-problem, and the combined solution of the previously defined subproblems should give the solution to the general problem (Klopper and Lubbe, 2011).

When this method is applied to this master thesis research, the general problem can be defined as follows:

1. In the convection section of a fired heater, finned tubes are used to improve heat transfer. For over the past 40 years, various correlation methods have been developed and have been widely used throughout the industry to calculate heat transfer in the convection section of fired heaters. However, a recent study introduces a method which analytically calculates heat transfer in the convection section of fired heaters (Janeschitz-Kriegl, 2017). Neither experiment nor correlation is required in the derivation of the new analytical method.

From the general problem, the following sub-problems can be characterized:

1.1 Correlations have been available to calculate heat transfer in the convection section of fired heaters for more than 40 years. Although correlations have proven to be reliable calculation methods, they do not provide insight in the working principle of fins.

1.2 Nowadays, there is a growing trend towards larger tube diameters. Methods to determine heat transfer for these larger tubes are needed, but existing correlations have not been developed for tubes with diameters larger than 4".

1.3 The analytical method uses a completely different angle to calculate heat transfer in the convection section of fired heaters. It has the potential to replace or to be used alongside existing correlations to provide an alternative approach for the development of fired heaters. However, for the proof of concept calculation some significant assumptions have been made.
1.4 A considerable amount of research is still necessary to be able to validate the new analytical method.

After formulation of the sub-problems, the same number of research questions are defined, while proper alignment with the sub-problems is ensured:

1.1 What method has been used to develop popular correlations? Why are these correlations still widely used throughout the industry, even though correlation methods do not provide insight regarding the working principle of fins?

1.2 Why is there a growing trend towards the use of larger tube diameters? Can existing correlations handle cases with large tube diameters?

1.3 How has the analytical method been developed, and what is the physical foundation behind this method? Which assumptions and simplification have been made to calculate heat transfer of finned tubes?

1.4 What methodology ensures the appropriate numerical model? And with the appropriate numerical model, what research has to be done in order to be able to validate the results of the analytical method?

1.3 Research Approach

The research questions defined above will be answered by application of the research approach which can be seen in Figure 1-2. It is observed that a large portion of these research questions has already been answered during the literature survey, where clear insights were obtained on the working principle of fired heaters, existing correlation methods, the analytical method, flow and heat transfer characteristics around finned tubes, and numerical methods which could be used to develop a numerical model to solve this heat transfer problem (van Dijk, 2019). The focus of the master thesis research is on the actual development and on the subsequent interpretation of the results obtained by the numerical model. These results will be compared with the results calculated by correlation methods and the analytical method. After this comparison has been made, it will be concluded if the analytical method is indeed a valid method to calculate heat transfer in the convection section of fired heaters.

![Figure 1-2: Visual representation of the established research approach.](image-url)
1-4 Research Scope

This master thesis research will be focused on the investigation of heat transfer and flow characteristics around finned tubes, which includes 4’ tubes, 6’ tubes, and 8’ tubes. These tube sizes are the most relevant for applications related to fired heaters. For these three tube sizes, both the heat transfer and pressure drop coefficient relative to the bare tube area $U_{bare}$ and $\zeta$ will be investigated versus the fin density $N_f$ for one tube row. The results obtained from the numerical model will be compared against the results calculated by the the most well-known correlation method (the ESCOA correlation method) and the analytical method. In addition, for 4’ tubes and 6’ tubes $U_{bare}$ will be investigated versus the number of tube rows up to $N_R = 10$ and will again be compared against the results calculated by the ESCOA correlation method and the analytical method. To conclude, a comparison between two different fin configurations is made for 6’ tubes and 8’ tubes for one tube row. Similar to the previous two investigations, the results will be compared against the results calculated by the ESCOA correlation method and the analytical method.

Only solid annular-finned tubes in a staggered equilateral tube arrangement will be considered during this master thesis research. This means that no serrated or toothed fins, and no in-line tube arrangement will be investigated.

1-5 Report Outline

Corresponding to any scientific report, the master thesis report approximately has the same formalized structure and follows the approach from Figure 1-2. First, the theory from the literature survey which is the most relevant for the master thesis research will be touched upon again in Chapter 2. This includes a brief explanation of the working principle of fired heaters, an introduction to the ESCOA correlation method and the analytical method, and an introduction to numerical methods related to modeling flow around finned tubes. In Chapter 3, a methodology is proposed for the development of the numerical model which will be used to validate the analytical method. The results obtained by the numerical models will be presented and discussed by means of a comparison with the ESCOA correlation method and the analytical method in Chapter 4. To conclude, in Chapter 5 final conclusions related to the results will be presented and in Chapter 6 recommendations for further research will be suggested.
2-1 Introduction

In order to develop an appropriate numerical model which is able to validate the analytical method, sufficient knowledge about various topics related to finned tube heat transfer is required. These topics have already been discussed elaborately during the literature survey (van Dijk, 2019). However, the most relevant theory will be touched upon in this chapter concisely to ensure that the steps taken in forthcoming chapters are understood properly.

2-2 Working Principle of Fired Heaters

2-2-1 Introduction

A schematic of one of the most common types of fired heaters (cabin-type fired heater) can be observed in Figure 2-1. In essence, the working principle of fired heaters are similar to that of large heat exchangers. However, in the case of fired heaters the heat transfer mechanism is different in different parts of the fired heater, resulting in a division of two main sections. The first section is the radiant section (no. 14 in Figure 2-1), where the majority of the heat is exchanged by means of radiation. The radiative heat input is provided by the combustion of fuel (no. 5 in Figure 2-1), which is usually oil (nowadays hardly used anymore due to stricter emission regulations) or natural gas, and is transferred to the process fluid flowing in tubes in the radiant section (no. 10 in Figure 2-1).

The remaining heat from the hot flue gas is being led through the convection section. The convection section, often called a convection bank (no. 7 in Figure 2-1), is placed on top of the radiation section to improve the overall efficiency of fired heaters. In fact, the amount of heat transferred by convection in the convection section may vary from five to twenty percent
Figure 2-1: Schematic of a cabin-type fired heater (image from API Standard 560 (American Petroleum Institute, 2001)).
of the total radiant duty in the radiant section (Wimpress and Braun, 1963), and according
to FLUOR engineers responsible for the design of fired heaters, this number can increase up to 30% present-day. The convection section is also the location where the process fluid enters the fired heater (no. 13 in Figure 2-1), after which multiple passes are made through the convection bank with tubes ranging from 4" to 6" in diameter (no. 10 and no. 11 in Figure 2-1). To improve heat transfer and achieve the percentages mentioned above, the heat transfer surface is increased by the application of fins (no. 12 in Figure 2-1). After the flue gas has passed the convection section, the stack is used to provide draft and dispose the gas (no. 21 in Figure 2-1).

The main focus of this master thesis lies on the investigation of heat transfer and flow characteristics related to the convection section of fired heaters. This requires more knowledge of the convection bank and corresponding finned tubes rather than of the radiant section. Both sections have been discussed elaborately during the literature survey, but in the master thesis report solely the convection section will be briefly discussed.

### 2-2-2 Convection Section

#### Shield Tubes

Due to the high operating temperatures in the radiant section of the fired heater, the lower part of the convection bank consists of shield tubes, also called shock tubes (no. 15 in Figure 2-1). These tubes protect the subsequent tube rows from receiving direct radiation from the radiant section.

The shield section must have at least three rows of bare tubes, and the shield tubes exposed to direct radiation have to be of the same material and thickness as the radiant tubes in the radiant section of the fired heater (American Petroleum Institute, 2001). Bare tubes are used for the shield section due to the high heat flux coming from direct radiation from the radiant section. Heat transfer for the shield tubes is already substantial, and the addition of fins would increase the heat transfer area and therefore would increase heat transfer even further. As a consequence, the increased heat flux would severely damage the fins and the shield tubes to which the fins are attached to.

#### Finned Tubes

After the hot flue gas has passed the shield section, it arrives at the finned section of the convection bank (no. 11 in Figure 2-1). The finned section of the convection bank is characterized by the use of tubes with a diameter ranging from 4" to 6" in diameter, occasionally an outlier to 8" is encountered. On these tubes fins are attached to increase the heat transfer area of the tubes, and correspondingly increase the overall heat transfer rate. It has been previously mentioned that the finned section of the convection bank is also the location where the process fluid enters the fired heater. Heating of the process fluid is initiated in the convection section before heated further in the radiant section of the fired heater, because of the lower operating temperature of the convection section compared to the radiant section of the fired heater. Despite this lower operating temperature, the heat transfer rate in the
convection section could approach the heat transfer rate of the radiant section as a result of the use of fins (Schweppe and Torrijos, 1964).

There is a great abundance of fin configurations available, but only transversely mounted fins will be considered here. Two main types of this fin configuration can be seen in Figure 2-2: Spiral or solid annular fins, and serrated (toothed) fins. Both types have their advantages and disadvantages. Serrated fins can achieve higher heat transfer rates compared to spiral fins. The first reason for this is because serrated fins have a larger total outside surface compared to spiral fin configurations. The second reason for this is that, due to the short flow length, serrated fins disallow boundary layers to grow to a larger thickness on the fin surface (ESCOA Corporation, 1979). However, the enhanced heat transfer comes at the expense of a larger pressure drop compared to the pressure drop of an solid annular fin configuration.

Fins are essentially metal strips which are wound around and fixed to the bare tube. Various methods can be used to establish this. If the gas temperature is not too high, the fin strip can be pressure mounted to the bare tube, which is done regularly for Heating, Ventilation and Air Conditioning (HVAC) related applications where the fins are often made from aluminum (Frass, 2015). This method cannot be applied for fired heaters, since the high temperatures in the convection section would cause the aluminum to expand and lose the connection with the bare tubes, making them nonfunctional. Instead, steel fins are used (can be various types of steel depending on the fin temperature) which are welded on the bare tube by high frequency welding or laser welding (Frass, 2015), (ESCOA Corporation, 1979). When these steel strips are wound around the tube, the outer edge of the fin will stretch, and the inner edge (or fin base) will compress. Due to this stretching and compressing, the tip of the fin will be slightly thinner while the base of the fin will be slightly thicker. In addition, at the fin base corrugation (wrinkled pattern) could sometimes be observed. This can result in poor weld quality, and will have an influence on the conductivity of the tube to the fin (ESCOA Corporation, 1979). It is hypothesized that these two aspects might also influence heat transfer and flow characteristics. This is not included in the scope of this master thesis research, but it could be a suggestion for further research.
It has been previously mentioned that the convection bank is characterized by the use of 4” tubes and 6” tubes. However, over the last couple of years, FLUOR engineers responsible for the design of fired heaters notice a trend towards the use of tubes which are greater than 6” in diameter. The first and main reason for this trend is that processes are getting larger. A consequence of this development is that fired heaters are getting larger as well, which results in higher process fluid flow rates. This is directly correlated to the second reason why there is a growing trend towards increased tube diameters. An increase in flow rate is not necessarily directly proportional to the amount of finned tubes used in the convection section of the fired heater. Fired heaters are often designed with approximately fixed ratios in mind, and deviating from this ratio could give rise to design problems. Hence, to cope with the increased flow rates it is preferred to use larger tubes instead of an extra number of smaller tubes. Even though this is less favorable for heat transfer, this has some advantages. First, the use of larger tubes reduces costs. Although larger tubes are more expensive, every pass through the convection bank needs instrumentation to measure and control the flow. Larger tubes require less control sets, which overall results in cheaper design. Second, it makes convection bank assembly and maintenance more convenient due to the reduction in weight and due to the increase in tube pitch.

**Tube Pitch**

The process fluid makes multiple passes in the convection bank before flowing to the radiant section. To accomplish this, so-called return bends (no. 12 in Figure 2-1) are used. Multiple return bends are placed next to each other, this way layering the tubes in the convection bank. In essence, this layering of tubes can be done in two arrangements, which is shown in Figure 2-3. The figure on the left represents a staggered arrangement, where the tubes are stacked on top of each other in a zigzag formation. The angle corresponding to this formation is determined by:

$$\tan \alpha = \frac{t_q}{2t_l}$$  \hspace{1cm} (2-1)

Here, \(t_l\) and \(t_q\) represent the longitudinal (along the main flow direction) and transverse (orthogonal of the main flow direction) tube pitch between tube centers respectively. For staggered tube arrangements \(\alpha\) can vary, depending on the design. An often encountered value of \(\alpha\) is 30°. In addition, tube centers are often placed two times the inside tube diameter from each other. Together this represents equilateral triangular pitch where all sides of the triangle have the same length and the corners have the same angle of 60° (Frass, 2015). The second arrangement is called an in-line arrangement, where the tubes are stacked on each other in a zigzag formation. The angle corresponding to this formation is 0°.

Whether there has been chosen for a staggered or an in-line tube arrangement makes a dramatic impact on heat transfer and flow characteristics of finned tubes inside the convection bank. It can be pictured that the subsequent tube rows will be influenced by the flow coming from the upstream tubes. For the in-line arrangement, the subsequent tube rows are in the wake region of the prior tube row. In the wake region at the rear side of the tube even reverse flow could occur, resulting in poor local heat transfer (Mon and Gross, 2004). For a staggered tube arrangement, the first tube row stimulates turbulence for the subsequent tube rows. In addition, the so-called "blockage effect", where the gas is forced to flow around the fins.
causes an increase in velocity for subsequent tube rows (Mon and Gross, 2004). The latter is favorable when high heat transfer is required and as a result, this is often the arrangement which is chosen for the design of the convection section of fired heaters. The same applies to this master thesis research, where only a staggered tube arrangement with equilateral pitch has been considered.

2-3 ESCOA Correlation Method

2-3-1 Introduction

Complex heat transfer and flow characteristics caused by fined tubes paved the way for the development of many correlation methods to calculate heat transfer and pressure drop, which have been derived mostly from wind tunnel experiments. In general, these correlations express the convective heat transfer coefficient $h_c$ as the dimensionless Nusselt number $Nu$:

$$Nu = \frac{h_c l'}{k}$$  \hspace{1cm} (2-2)

Where $l'$ represents the characteristic length and $k$ represents the thermal conductivity of the flowing medium. Various correlations to calculate $Nu$ for external forced convection around a bare tube can be found in the Basic Heat & Mass Transfer book written by Mills (1999) in Chapter 4.3.2. Which of these correlations to use to calculate $Nu$ depends on the value of the Reynolds number $Re$. The Reynolds number determines if a flow is laminar or turbulent and is defined as:

$$Re = \frac{\rho V l'}{\mu} = \frac{Gl'}{\mu}$$  \hspace{1cm} (2-3)

By multiplication of the density with the velocity, $G = \rho V$ denotes the mass flux and represents the rate of mass flow per unit area. In the process industry it is more common to specify the mass flux $G$ rather than velocity $V$. The dynamic viscosity is represented by $\mu$. 

---

J. van Dijk

Thesis Report
Fins make these correlations even more bulky compared to the correlations for bare tubes, since geometrical variables for the fins are added (fin height, fin thickness, fin density, etc.). In addition, no clear characteristic length \( l' \) can be specified for finned tubes. For calculations corresponding to bare tubes it is evident that the outside tube diameter should be used, yet for finned tubes many different proposals for the characteristic length exist (Frass, 2015).

Two correlation methods have been investigated. The correlations developed by Heat Transfer Research, Inc. (HTRI) and Extended Surface Corporation Of America (ESCOA) have proven to be the most complete and accurate correlations available today, and for both methods heat transfer and pressure drop correlations have been listed and explained during the literature survey (van Dijk, 2019). FRNC, which is a software program used by FLUOR engineers responsible for the design of fired heaters, solely adopts the ESCOA correlation method. In contrast to the HTRI correlation method, the correlations developed by ESCOA are able to calculate heat transfer and pressure drop for both staggered and in-line tube arrangements, and both for solid and serrated fins. Due to this versatility, only the ESCOA correlation method has been considered for a comparison with the analytical method during the literature survey, and also for this master thesis research only the ESCOA correlations have been investigated. The investigation of the ESCOA correlation method and the other methods have been done only for solid annular fins. The reason for this is that solid annular fins are more widely used compared to serrated fins. However, the investigation of serrated fins could be a suggestion for further research.

2-3-2 ESCOA Correlations

Heat Transfer Correlations

The ESCOA correlation method to determine heat transfer and pressure drop was first proposed by Weierman (1976). The equations regarding heat transfer have been transformed from U.S. customary units to metric units by Frass (2015), and are given by Equation 2-4 to Equation 2-7. The main correlation equation is given by:

\[
Nu = C_1 Re Pr^{1/3} \left( \frac{T_{in} + 273.2}{T_{rm} + 273.2} \right)^{1/4} \left( \frac{D}{d_A} \right)^{0.5} C_3 C_5 \tag{2-4}
\]

\( T_{in} \) and \( T_{rm} \) represent the inlet flue gas temperature and the mean fin temperature in Celsius respectively. \( D \) denotes the bare tube diameter including the fins: \( D = d_A + 2h \).

Coefficient \( C_1 \) is a function of the Reynolds number and is defined by:

\[
C_1 = 0.25 Re^{-0.35} \tag{2-5}
\]

Since \( C_1 \) is dependent on the Reynolds number, it accounts for variables such as mass flux, bare tube diameter and viscosity.

Coefficient \( C_3 \) specifies the influence of fin height and the fin distance:

\[
C_3 = 0.35 + 0.65 \exp \left( -0.25h/(t_R - s_R) \right) \tag{2-6}
\]

Variables \( h, t_R \) and \( s_R \) represent fin geometries, and can be seen schematically in Figure 2-4.
Coefficient $C_5$ accounts for the influence of the transverse and longitudinal pitch in the tube bundle as well as the number of consecutive tube rows $N_R$:

$$C_5 = 0.7 + (0.7 - 0.8 \exp (-0.15N_R^2)) \exp (-t_l/t_q)$$  \hspace{1cm} (2-7)

To obtain the values of the heat transfer coefficient relative to the bare tube area $U_{bare}$:

$$U_{bare} = \frac{Nuk_{gas}d}{A_{bare}} \cdot \frac{E A_{xfer,fin} + A_{xfer,prim}}{A_{bare}}$$  \hspace{1cm} (2-8)

$A_{bare}$ is the bare tube area, $A_{xfer,fin}$ is the fin area, and $A_{xfer,prim}$ is the primary surface area, which is the bare tube surface area minus the area to which the fins are attached to on the bare tube. $E$ is called the fin efficiency and is calculated according to ESCOA by (Weierman, 1976):

$$E = Y (0.45 \ln \frac{D}{d_A} (Y - 1) + 1)$$  \hspace{1cm} (2-9)

$$Y = X (0.7 + 0.3X)$$  \hspace{1cm} (2-10)

$$X = \frac{\tanh(mb)}{mb}$$  \hspace{1cm} (2-11)

$$m = \sqrt{\frac{2hc}{s_R k_{fin}}}, \hspace{1cm} b = h + \frac{s_R}{2}$$  \hspace{1cm} (2-12)

A plot of $U_{bare}$ versus the fin density $N_f$ for various tube diameters calculated by the ESCOA correlation method is shown in Figure 2-5. The operation conditions which have been used can be seen in Table 2-1 and Table 2-2. Unless stated otherwise, all subsequent plots listed further in this master thesis report will adopt these values.
Table 2-1: Operating conditions used to obtain heat transfer and pressure drop results.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_{tube}$</td>
<td>148.9 °C</td>
</tr>
<tr>
<td>$T_{in}$</td>
<td>426.7 °C</td>
</tr>
<tr>
<td>$T_{rm}$</td>
<td>0.5($T_{in} + T_{tube}$) = 287.8 °C</td>
</tr>
<tr>
<td>$h$</td>
<td>1 [inch] = 0.0254 [m]</td>
</tr>
<tr>
<td>$s_R$</td>
<td>0.00127 [m]</td>
</tr>
<tr>
<td>$N_R$</td>
<td>1 [-]</td>
</tr>
<tr>
<td>$G_{avg}$</td>
<td>2 [kg/m²·s]</td>
</tr>
<tr>
<td>$\mu$</td>
<td>3.16·10⁻⁵ [kg/m·s]</td>
</tr>
<tr>
<td>$k_{fin}$</td>
<td>46.737 [W/m·K]</td>
</tr>
<tr>
<td>$k_{gas}$</td>
<td>0.04916 [W/m·K]</td>
</tr>
<tr>
<td>$\Pr$</td>
<td>0.7 [-]</td>
</tr>
</tbody>
</table>

Table 2-2: Bare tube diameters with corresponding transverse and longitudinal tube pitch. Left: 4” tubes. Center: 6” tubes. Right: 8” tubes.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_A$</td>
<td>0.1143 [m]</td>
</tr>
<tr>
<td>$t_q$</td>
<td>0.2032 [m]</td>
</tr>
<tr>
<td>$t_l$</td>
<td>0.1760 [m]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_A$</td>
<td>0.1683 [m]</td>
</tr>
<tr>
<td>$t_q$</td>
<td>0.3048 [m]</td>
</tr>
<tr>
<td>$t_l$</td>
<td>0.2640 [m]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_A$</td>
<td>0.2191 [m]</td>
</tr>
<tr>
<td>$t_q$</td>
<td>0.4064 [m]</td>
</tr>
<tr>
<td>$t_l$</td>
<td>0.3520 [m]</td>
</tr>
</tbody>
</table>

Figure 2-5: Heat transfer coefficient relative to the bare tube area $U_{bare}$ versus the fin density $N_f$ for various bare tube diameters calculated by the ESCOA correlation method. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m²·s.
Pressure Drop Correlations

Proficiency in obtaining gas-side pressure drop over the finned tube bundle is just as important as being able to calculate heat transfer related to the design of fired heaters. Pressure drop can be expressed according to Equation 2-14:

$$\Delta p = \zeta \frac{\rho}{2} V^2$$  \hspace{1cm} (2-14)

For the velocity $V_{\text{max}}$ is used to calculate the pressure drop over the tube bundle, which represents the velocity at the area with the narrowest cross-section between two tubes stacked on top of each other. $\zeta$ represents the pressure drop coefficient relative to the bare tube area.

In addition to heat transfer correlations, ESCOA also developed correlations regarding pressure drop (Weierman, 1976). Similar to the correlations related to heat transfer, the pressure drop correlations also have been transformed from U.S. customary units to metric units by Frass (2015) and are represented by Equations 2-15 to 2-18:

$$\zeta = 4C_2C_4C_6 \left( \frac{D}{d_A} \right)^{0.5}$$  \hspace{1cm} (2-15)

The correlations the pressure drop also depends on three coefficients: $C_2$, $C_4$, and $C_6$. Similar to coefficient $C_1$ used for the heat transfer calculations, $C_2$ takes the influence of the Reynolds number, and therefore flow speed, bare tube diameter and viscosity into account:

$$C_2 = 0.07 + 8.0Re^{-0.45}$$  \hspace{1cm} (2-16)

Coefficient $C_4$ represents the influence of fin and tube variables, such as the bare tube diameter, transverse tube pitch and fin specifications like the fin thickness and fin height:

$$C_4 = 0.11(0.05 \frac{t_q}{d_A})^{(-0.7)} \frac{h}{r_R} \frac{1}{(r_R - h)}^{0.20}$$  \hspace{1cm} (2-17)

Coefficient $C_6$ accounts for the influence of both the longitudinal and the transverse tube pitch, as well as the number of tube rows in the tube bundle:

$$C_6 = 1.1 + (1.8 - 2.1 \exp(-0.15N_R^2)) \exp\left(-2 \frac{t_q}{t_l} \right) - (0.7 - 0.8 \exp(-0.15N_R^2) \exp\left(-0.6 \frac{t_l}{t_q} \right)$$  \hspace{1cm} (2-18)

A plot of $\zeta$ versus the fin density $N_f$ for various tube diameters calculated by the ESCOA correlation method is shown in Figure 2-6. Compared to the heat transfer calculations, the same operating conditions have been used for the pressure drop calculations, which can be seen in Table 2-1 and Table 2-2.

Although the ESCOA correlations might appear to look bulky, it is still relatively simple to calculate both heat transfer and pressure drop assuming that all necessary process conditions are known. In Figure 2-5 and Figure 2-6 it can be observed that 4" tubes have superior heat transfer but also have an increase in pressure drop over 6" and 8" tubes, whereas 8" tubes have the lowest value for both $U_{\text{bare}}$ and $\zeta$ compared to the other tube sizes. This is evident, because of the increased transverse tube pitch $t_q$ corresponding to 6" tubes and 8" tubes compared to 4" tubes. In addition, the maximum fin height is restricted to $h = 1"$.

J. van Dijk
2-4 Analytical Method

2-4-1 Introduction

An entirely new way of calculating heat transfer (and pressure drop as a by-product) in the convection section of fired heaters has been introduced by the analytical method (Janeschitz-Kriegl, 2017). The analytical method considers heat transfer locally at the fins while using appropriate methods, which will be discussed elaborately in this section. This is in contrast with the ESCOA correlation method, which uses the Reynolds number based on the outer bare tube diameter \( d_A \) to correlate it to fin heat transfer to which it is only remotely related.

Figure 2-6: Pressure drop coefficient relative to the bare tube area \( \zeta \) versus the fin density \( N_f \) for various bare tube diameters calculated by the ESCOA correlation method. Number of tube rows \( N_R = 1 \), average mass flux \( G_{\text{avg}} = 2 \text{ kg/m}^2\cdot\text{s} \).

According to API Standard 560 (American Petroleum Institute, 2001). Together, this makes it easier for the flue gas to flow between the tubes for 6” tubes and 8” tubes compared to 4” tubes. However, it difficult to clarify this by observation of the correlations themselves since the ESCOA correlations have been derived from wind tunnel experiments and data sets (Weierman, 1976). A change in one of the variables (e.g. transverse tube pitch \( t_q \)) leads to a change in \( C_1 \) to \( C_6 \) and subsequently results in a change in \( U_{\text{bare}} \) and \( \zeta \), but both the correlations for heat transfer and pressure drop do not provide the user insight in how finned tube heat transfer actually works. This lack of insight, together with the absence of a validity range in any way are considered to be major deficiencies corresponding to the ESCOA correlation method.

Thesis Report J. van Dijk
Neither wind tunnel experiments nor correlations have been used for the derivation of the analytical method. The derivation is fairly simple and consists of two main principles. The first principle is the determination of the mass flow ratios. This includes the flue gas mass flow which bypasses the fins and the flue gas mass flow which flows between the fins. Only the latter takes part in the subsequent heat transfer calculations corresponding to the second principle. Here, the the heat transfer coefficient relative to the bare tube area $U_{bare}$ is calculated by application of $\epsilon$-NTU method.

The derivation of the analytical method explained in this section still represents a proof of concept calculation. This means that a number of assumptions and simplifications have been made in order to make the proof of concept calculation work, which will be discussed further on in this section.

### 2-4-2 Derivation of the Analytical Method

#### Mass Flow Calculations

The pressure drop over one tube row has already been described by the general expression Equation 2-14. The analytical method assumes a pressure loss of one velocity head:

$$\Delta p_{\text{free}} = \zeta_{\text{free}} \frac{\rho_{\text{free}} V^2_{\text{free}}}{2} = \frac{1}{2 \rho_{\text{free}}} G^2_{\text{free}}$$  \hspace{1cm} (2-19)

Subscript "free" denotes the free flow or the flow which bypasses the fins. $\zeta_{\text{free}}$ is not comparable to $\zeta$ calculated by the ESCOA correlation method listed in Section 2-3 because $\zeta$ from ESCOA is based on the total flow area $A_{\text{flow,total}}$ (the area of the flow which bypasses the fins plus the area of the flow which flows between the fins), while $\zeta_{\text{free}}$ is based on the free flow area $A_{\text{flow,fine}}$ (only the area of the flow which bypasses the fins). However, it is possible to calculate $\zeta$ for the analytical method to compare it with the ESCOA correlation method, which will be demonstrated further in this section. The assumption that $\zeta_{\text{free}} = 1$ made according to Equation 2-19 is an unproven one, and further research needs to be done to further specify $\zeta_{\text{free}}$. Since this assumption does not change the derivation and the working principle of the analytical method, it is presumed that $\zeta_{\text{free}} = 1$ is a valid assumption for this proof of concept calculation.

Corresponding to the fins, a similar Equation like Equation 2-19 takes form:

$$\Delta p_{\text{fin}} = \zeta_{\text{fin}} \frac{1}{2 \rho_{\text{fin}}} G^2_{\text{fin}}$$  \hspace{1cm} (2-20)

Subscript 'fin' denotes the flow between the fins. $\zeta_{\text{fin}}$ represents the fin pressure drop coefficient. According to Mills (1999), $\zeta_{\text{fin}}$ can be calculated by multiplication of the laminar Darcy friction factor $f_{\text{lam}}$ with a ratio which consist of the flow path length between the fins $L_{\text{flow,fin}}$ and the hydraulic diameter $D_{\text{hydr,fin}}$ between the fins:

$$f_{\text{lam}} \frac{L_{\text{flow,fin}}}{D_{\text{hydr,fin}}} \rightarrow \zeta_{\text{fin}} = f_{\text{lam}} \frac{L_{\text{flow,fin}}}{D_{\text{hydr,fin}}}$$  \hspace{1cm} (2-21)
For the proof of concept calculation, the fins are assumed to be infinite parallel plates. Definition of the hydraulic diameter $D_{\text{hydr,fin}}$ for parallel plates is given by:

$$D_{\text{hydr,fin}} = 2S_{\text{fin}} = 2\left(\frac{1}{N_f} - s_R\right)$$  \hspace{1cm} (2-22)

$S_{\text{fin}}$ stands for the fin spacing (not to be confused with fin pitch $t_R$ shown in Figure 2-4). Similar to the assumption made in Equation 2-19, the assumption that the fins can modeled as parallel plates according to Equation 2-22 is an assumption which needs improvement, but does not influence the working principle of the analytical method.

The visualization of the flow path length $L_{\text{flow,fin}}$ is shown in Figure 2-7, and can be defined as half the circumference of the bare tube diameter plus the fin height:

$$L_{\text{flow,fin}} = \frac{\pi}{2}(h + d_A)$$  \hspace{1cm} (2-23)

Adding the assumption that that the flow through the fins is laminar to the assumption that the fins can be modeled as infinite parallel plates leads to (Mills, 1999):

$$f_{\text{lam}} = \frac{96}{Re_{\text{fin}}}$$  \hspace{1cm} (2-24)

$Re_{\text{fin}}$ represents the Reynolds number of the flue gas flow which flows between the fins and is defined as $Re_{\text{fin}} = G_{\text{fin}}D_{\text{hydr,fin}}/\mu$. Equation 2-24 is valid only if the flow through the fins is laminar, which means that the Reynolds number for a flow between two flat plates should be $Re_{\text{fin}} < 2300$ in order for the analytical method to be valid (Mills, 1999). Although this might look like a restriction, a clear validity range is specified in contrast to the ESCOA correlation method.

By substitution of Equation 2-21, Equation 2-23, and Equation 2-24 in Equation 2-20, the following equation of the pressure drop over the fin is obtained:

$$\Delta p_{\text{fin}} = 48\pi\mu \frac{d_A + h}{D_{\text{hydr,fin}}^2} \frac{G_{\text{fin}}}{2\rho_{\text{fin}}}$$  \hspace{1cm} (2-25)

Figure 2-7: Schematic of the flow path length $L_{\text{flow,fin}}$ depicted by the curved dark blue arrow.
By definition, $\Delta p_{\text{row}} = \Delta p_{\text{fin}}$, since the pressure drop over one tube row is also the driving force for the flow through the fins:

$$\frac{G_{\text{free}}^2}{2\rho_{\text{free}}} = 48\pi \mu \frac{d_A + h}{D^2_{\text{hydr,fin}}} \frac{G_{\text{fin}}}{2\rho_{\text{fin}}} \tag{2-26}$$

This can be rewritten to:

$$G_{\text{free}}^2 = RG_{\text{fin}} \tag{2-27}$$

$R$ is a collective variable depending on geometrical parameters from Equation 2-26 with the same units as the mass flux $G$, and is described as:

$$R = 48\pi \mu \rho_{\text{free}} \frac{d_A + h}{D^2_{\text{hydr,fin}}} \tag{2-28}$$

To calculate the value of $R$, densities $\rho_{\text{free}}$ and $\rho_{\text{fin}}$ are calculated according to the alternate form of the ideal gas law:

$$\rho = \frac{pM}{\bar{R}T} \tag{2-29}$$

Where $p$ denotes the pressure (which is assumed to be atmospheric), $\bar{R}$ the universal gas constant, and $M$ the molar weight of the flue gas. For the calculation of $\rho_{\text{free}}$ it applies that $T = T_{\text{in}}$. For $\rho_{\text{fin}}$ it applies that $T = T_{\text{rm}}$, with $T_{\text{rm}} = 0.5(T_{\text{in}} + T_{\text{tube}})$. Temperatures have to be converted from Celsius to Kelvin in order to calculate $\rho_{\text{free}}$ and $\rho_{\text{fin}}$ in Equation 2-29.

With $R$ known, Equation 2-27 can be rewritten to:

$$\left(\frac{\dot{m}_{\text{free}}}{A_{\text{flow,free}}} \right)^2 = R \frac{\dot{m}_{\text{fin}}}{A_{\text{flow,fin}}} \rightarrow \dot{m}_{\text{free}}^2 = S \cdot \dot{m}_{\text{fin}} \tag{2-30}$$

Where $S$ is a collective variable similar to $R$, but with the same units as the mass flow $\dot{m}$. $S$ is given by the following equation:

$$S = \frac{A_{\text{flow,free}}^2}{A_{\text{flow,fin}}} R \tag{2-31}$$

$A_{\text{flow,free}}$ and $A_{\text{flow,fin}}$ denote the projected flow area between the finned tubes and the flow area between the fins per meter of tube row respectively, and are calculated according to $A_{\text{flow,free}} = t_q - (d_A + 2h)$ and $A_{\text{flow,fin}} = 2hS_{\text{fin}}$.

From mass conservation, it follows that:

$$\dot{m}_{\text{fin}} = \dot{m}_{\text{total}} - \dot{m}_{\text{free}} \tag{2-32}$$

Substitution of Equation 2-31 in Equation 2-30 yields:

$$\dot{m}_{\text{free}}^2 = S(\dot{m}_{\text{total}} - \dot{m}_{\text{free}}) \rightarrow \dot{m}_{\text{free}}^2 + S\dot{m}_{\text{free}} - S\dot{m}_{\text{total}} = 0 \tag{2-33}$$

This is a quadratic equation in the form of $Ax^2 + Bx + C = 0$ with $x = \dot{m}_{\text{free}}$, $A = 1$, $B = S$, and $C = -S\dot{m}_{\text{total}}$. Equation 2-33 can be solved to obtain $\dot{m}_{\text{free}}$. Two real solutions exist for Equation 2-33, one which is positive and one which is negative. This is due to the fact that the discriminant of this equation $S(S + \dot{m}_{\text{tot}}) > 0$ for any value of $S$ and $\dot{m}_{\text{tot}}$, since both $S$ and $\dot{m}_{\text{tot}} > 0$. From these two answers, the positive value is the only correct value for $\dot{m}_{\text{free}}$. With $\dot{m}_{\text{free}}$ known, $\dot{m}_{\text{fin}}$ can be determined from Equation 2-32 after which the subsequent heat transfer calculations can be carried out.
Heat Transfer Calculations

By combination of the two assumptions that the fins are considered to be infinite parallel planes and that the flow between the fins is considered to be laminar ($Re_{fin} < 2300$), the Nusselt number for laminar flow between two infinite plat plates is constant and has a value of $Nu = 7.54$ (Mills, 1999). As a result, the fin heat transfer coefficient $U_{fin}$ can be determined according to:

$$U_{fin} = \frac{Nu k_{gas}}{D_{hydr,fin}} = \frac{7.54k_{gas}}{D_{hydr,fin}}$$  \hspace{1cm} (2-34)

The $\epsilon$-NTU method (Kays and London, 1984) is a renowned method to calculate heat transfer in heat exchangers. The Number of Transfer Units (NTU) can be seen as a nondimensional expression of the heat transfer size of the heat exchanger. The $\epsilon$-NTU method is particularly useful when the inlet and outlet temperatures of a heat exchanger are unknown, which are variables required for the Log-Mean Temperature Difference (LMTD) method to calculate heat transfer. The NTU can be calculated as follows:

$$NTU = \frac{U_{fin} A_{xfer,eff}}{C_{min}}$$  \hspace{1cm} (2-35)

$A_{xfer,eff}$ is the effective heat transfer area, and is defined as:

$$A_{xfer,eff} = A_{xfer,prim} + E \cdot A_{xfer,fin}$$  \hspace{1cm} (2-36)

For better comparison, the ESCOA correlation method is used to calculate fin efficiency $E$. Equation 2-9 till Equation 2-13 in Section 2-3 describe how $E$ is calculated.

In order to calculate the NTU described in Equation 2-35, the heat capacity rates need to be calculated (mass flow rate multiplied by the specific heat, see Equation 2-35). The smaller heat capacity rate is called $C_{min}$, and the larger heat capacity rate is called $C_{max}$. Finned tubes in the convection section of a fired heater are made of metal, and the flow inside the tubes is mixed well. In addition, the inside heat transfer coefficient is high (> 1000 W/m²·K). The well mixed flow and high inside heat transfer coefficient imply that the temperature of the tube wall $T_{tube}$ can be assumed to be constant. This has significant consequences for the way $\epsilon$ is calculated. According to Kays and London (1984), $\epsilon$ represents the ratio between the actual heat transfer rate and the maximum possible heat transfer rate and is given by $\epsilon = Q/Q_{max}$. The effectiveness of heat exchangers are given by $\epsilon$-NTU relations and is calculated for a counter-current heat exchanger as follows (Kays and London, 1984):

$$\epsilon = \frac{1 - e^{-NTU(1-C_{min}/C_{max})}}{1 - (C_{min}/C_{max})e^{-NTU(1-C_{min}/C_{max})}}$$  \hspace{1cm} (2-37)

For the limit case where the fluid side of the convection section will remain at constant temperature for one tube row, the temperature at the fin base is constant, and by definition: $C_{min}/C_{max} = 0$. This reduces Equation 2-36 to:

$$\epsilon = 1 - e^{-NTU}$$  \hspace{1cm} (2-38)
To calculate NTU in Equation 2-35, the smaller heat capacity rate $C_{\text{min}}$ can be calculated by $C_{\text{min}} = \dot{m}_{\text{fin}} \cdot c_p$. With $\epsilon$ known, the total heat transferred per meter of tube row can be determined by:

$$Q_{\text{fin}} = \dot{m}_{\text{fin}} c_p \epsilon (T_{\text{in}} - T_{\text{tube}})$$

(2-39)

The outlet flue gas temperature $T_{\text{out}}$ can now be calculated according to:

$$Q_{\text{fin}} = \dot{m}_{\text{tot}} c_p (T_{\text{in}} - T_{\text{out}}) \rightarrow T_{\text{out}} = T_{\text{in}} - \frac{Q_{\text{fin}}}{\dot{m}_{\text{tot}} c_p}$$

(2-40)

Ultimately, the heat transfer coefficient relative to the bare tube can be determined:

$$U_{\text{bare}} = \frac{Q_{\text{fin}}}{A_{\text{bare}}(T_{gm} - T_{\text{tube}})}$$

(2-41)

$T_{gm}$ represents the mean flue gas temperature and is calculated according to $T_{gm} = 0.5(T_{\text{in}} + T_{\text{out}})$. A plot of $U_{\text{bare}}$ versus the fin density $N_f$ calculated by the analytical method is given in Figure 2-8.

It has been previously mentioned in this section that it is also possible to calculate $\zeta$ for the analytical method which is comparable with $\zeta$ calculated by the ESCOA correlation method. The pressure coefficient by the latter is based on the total flow area $A_{\text{flow,total}}$, while for the analytical method $\zeta_{\text{free}}$ and $\zeta_{\text{fin}}$ are based on the free flow area $A_{\text{flow,free}}$ and fin flow area $A_{\text{flow,fin}}$ respectively. This also means that $\zeta$ calculated by the ESCOA correlation method depends on the average mass flux $G_{\text{avg}}$, while $\zeta_{\text{free}}$ and $\zeta_{\text{fin}}$ from the analytical method depend on $G_{\text{free}}$ and $G_{\text{fin}}$. For a fair comparison, the equation for $\zeta$ corresponding to the analytical method is therefore given by:

$$\Delta p_{\text{fin}} = \Delta p_{\text{row}} = \Delta p_{\text{avg}} = \frac{G_{\text{avg}}^2}{2 \rho_{\text{avg}}}$$

(2-42)

Figure 2-8: Heat transfer coefficient relative to the bare tube area $U_{\text{bare}}$ versus the fin density $N_f$ for various bare tube diameters calculated by the analytical method. Number of tube rows $N_R = 1$, average mass flux $G_{\text{avg}} = 2$ kg/m$^2$·s.
The average mass flux $G_{\text{avg}}$ is known, since it is an input variable for the analytical method and $\rho_{\text{avg}}$ is the average density defined as $\rho_{\text{avg}} = (\dot{m}_{\text{free}} + \dot{m}_{\text{fin}})/\dot{V}_{\text{total}}$, where $\dot{V}_{\text{total}}$ stands for the total volumetric flow rate. A plot of $\zeta$ versus the fin density $N_f$ calculated by the analytical method is shown in Figure 2-9.

Both the ESCOA correlation method and the analytical method calculate the maximum and minimum values of $U_{\text{bare}}$ for 4” tubes and 8” tubes respectively. However, certain aspects stand out by observation of Figure 2-8. In contrast to the ESCOA correlation method, which predicts increased heat transfer for an increased fin density, the analytical method displays a distinct fin density optimum for $U_{\text{bare}}$. Related to $\zeta$, the difference in results between the ESCOA correlation method and the analytical method are still significant, though not as dramatic compared to $U_{\text{bare}}$. In Figure 2-9 it can be observed that for an increasing fin density, $\zeta$ seems to attain an asymptotic value for all investigated tube sizes. This is plausible, since high fin density values approximately resemble a tube with a diameter corresponding to $D = d_A + 2h$. In addition, $\zeta$ increases more rapidly for 4” tubes compared to 6” tubes and 8” tubes. This is due to two reasons. First, when the fin density increases less flue gas will flow between the fins and has to be diverted around the fins. For 4” tubes, this effect is more pronounced since (for fin height $h = 1”$) the diameter has essentially become 1.5x as large, while for 6” and 8” tubes this is 1.33x and 1.25x as large respectively. Second, the transverse tube pitch $t_q$ is smaller for 4” tubes compared to 6” and 8” tubes, which is shown in Table 2-2. Combined with the first reason, this means that less space between tubes is available for the diverted flow, which ultimately results in both a higher value of $\zeta$ and in a steeper slope of $\zeta$ corresponding to 4” tubes. Both the plots related to $U_{\text{bare}}$ and $\zeta$ will be discussed much more elaborate in Chapter 4.

Figure 2-9: Pressure drop coefficient relative to the bare tube area $\zeta$ versus the fin density $N_f$ for various bare tube diameters calculated by the analytical method. Number of tube rows $N_R = 1$, average mass flux $G_{\text{avg}} = 2$ kg/m²-s.
The explanation of the plots calculated by the analytical method is much more convenient compared to the plots calculated by the ESCOA correlation method, because the analytical method has a solid physical foundation and provides a clear picture on how fins actually work. This is in contrast with the ESCOA correlation method, where the plots related to heat transfer and pressure drop are explained much more difficult due to the fact that these correlations have been developed from wind tunnel experiments.

**Assumptions Made by the Analytical Method**

The analytical method presented here has been explained in its most simple form and is based on several assumptions. In short, these assumptions are:

1. The pressure drop of the free flow (the flue gas flow which bypasses the fins) is assumed to be one velocity head, which implies that $\zeta_{\text{free}} = 1$. In reality this is not exactly the case, and this effect will work its way through the derivation of the analytical method. However, it does not change the derivation and the working principle of the analytical method.

2. Since the fins are considered to be infinite parallel plates, the hydraulic diameter is assumed to be two times the fin spacing: $D_{\text{hydr,fin}} = 2S_{\text{fin}} = 2((1/N_f) - s_R)$. Although this is a good approximation for this application, this is not according to how the hydraulic diameter is usually calculated.

3. The flow between the fins is assumed to be laminar, and the flow between the tubes is assumed to be fully turbulent. Only the laminar flue gas flow which flows between the fins contributes to heat transfer. The assumption that the flow between the fins is laminar and that the fins are represented by infinite parallel plates makes the Nusselt number constant: $Nu = 7.54$. Due to this assumption, the laminar friction factor can be calculated as: $f_{\text{lam}} = \frac{96}{Re_{\text{fin}}}$. 

4. The flow inside the tubes is assumed to be well mixed, and the inside heat transfer coefficient is high. This makes $T_{\text{tube}}$ a constant variable which allows the heat exchanger effectiveness $\epsilon$ from the $\epsilon$-NTU method to be written as $\epsilon = 1 - e^{-\text{NTU}}$.

5. Heat transfer is evenly spread over the fin and tube surface. It can be realized that this is not really the case, since the upstream side of the finned tube would contribute considerably more in terms of heat transfer compared to the wake region of the finned tube. In addition, no heat transfer at fin tip surface has been considered yet.
2-5 Numerical Methods Applied to Finned Tubes

2-5-1 Introduction

To be able to validate the analytical method which has been presented Section 2-4, further research has to be done to investigate heat transfer and flow characteristics around finned tubes. This can be done either by conducting experiments or by developing numerical models. Both approaches have their benefits and limitations. Experiments are a direct representation of the real-world problem, and particle tracking methods such as Particle Image Velocimetry (PIV) allow researchers to investigate the mean velocity and velocity fluctuations with high accuracy and spacial resolution. On the other hand, the set up of an experiment can be an expensive and time-consuming process and it is nearly impossible to control every variable. Another drawback related to experiments is that measuring tools might interfere with local fluid flow and heat transfer, thus influencing experimental results (Nemati and Moghimi, 2014).

This is in contrast with CFD simulations, which are not affected by these disadvantages. However, the development of a numerical model can be, similar to experiments, an expensive and time-consuming process (depending on the model and geometry). Numerical models are able to study systems which cannot be studied by experiments, such as large systems where controlled experiments are difficult to perform, or systems which involve hazardous conditions. However, the underlying physics related to fluid dynamics and heat transfer is very complex, and, since the model is a resemblance of the real problem, the right choices and assumptions have to be made in order to produce a satisfactory model. It is necessary to reduce the complexity of the problem to a reasonable level for the computer to solve, while at the same time the most notable features of the problem need to be preserved. The creator of the numerical model should at all times be aware of every assumption and simplification that has been made, since it is the quality of these assumptions and simplifications that (partly) determine the quality of the numerical model.

Because various tube sizes and fin densities have to be investigated, experiments are too costly and time-consuming for the scope of this master thesis research. However, experiments related to finned tube heat transfer have been conducted in the past, from which the results can be found in the literature survey (van Dijk, 2019). In addition, the results obtained by numerical experiments related to finned tube heat transfer can also be found in the literature survey. Since multiple numerical models have been developed over the course of this master thesis research, the basics of CFD will briefly be touched upon in this section.

2-5-2 Brief Introduction to Numerical Methods

It has been previously mentioned that numerical methods such as CFD can provide a good alternative for experiments. This is especially true if large tubes or complete convection banks have to be investigated, where experiments simply would have been too expensive. However, the results can only be of any value if the numerical models have been developed according to certain guidelines, and if validation of the results has been taken place. Above all, there should be a proper understanding of fluid mechanics and heat transfer fundamentals, as well as the numerical algorithms used by the CFD software package. In short, a numerical model consists of three main components (Versteeg and Malalasekera, 2007):
• **Pre-processor:** During the pre-processing stage, the geometry of interest or the computational domain is defined. For this computational domain, a grid or mesh is generated, which is a subdivision of the computational domain into smaller domains. Material properties are also defined during the pre-processing stage. To conclude, where the grid cells touch the boundary of the computational domain appropriate boundary conditions need to be specified.

In general, a more accurate solution is obtained for a refined grid compared to a coarse grid. This comes at the expense of an increase in calculation time, because for each extra grid cell governing and turbulence model equations have to be solved. The generated mesh is often non-uniform, which means that the grid is more refined for areas where large gradients (e.g. in velocity, pressure, temperature) take place, and a grid which is more coarse for areas where little gradients occur.

• **Solver:** The work done by the pre-processing stage enables the solver to calculate a solution for the given problem. It is apparent that this is done numerically, and three numerical solution techniques exist: Finite Volume Method (FVM), Finite Element Method (FEM), and Finite Difference Method (FDM). FVM has an important advantage over the other two solution techniques. Since FVM is a conservative method, it ensures conservation of relevant properties for each grid cell which has been generated. For this specific reason FVM is the solution technique used by commercial CFD software packages such as ANSYS Fluent and CFX. FVM uses the following steps to calculate a solution:

  – Integration of the governing equations over all the individual grid cells which have been made by the meshing tool.
  – Descretization of the governing equations which have been integrated over all the control volumes into a system of equations which can be solved. Commercial CFD packages contain proper techniques to discretize governing equations while preserving important transport phenomena.
  – The system of equations is solved by an iterative method. This has to be done iteratively since the equations are complex and highly non-linear. Commercial CFD packages allow users to specify numerous iterative methods.

• **Post-processor:** During the post-processing stage, the solution data calculated by the solver is represented visually. This enables direct observation of the velocity field, temperature gradients, etc. corresponding to the computational domain.

The pre-processing stage requires the most labor to ensure the development of a high quality CFD model. The definition of the computational domain and the generation of the grid can take up to 50% of the labor during the development of a numerical model (Versteeg and Malalasekera, 2007). Therefore, four important features related to the pre-processing stage will be discussed: Grid generation, turbulence models, near-wall treatment, and appropriate selection of boundary conditions.
2-5-3 Mesh

The generation of a grid is essentially the same as the discretization of the computational domain, and has a great influence on the quality and required calculation time related to the numerical model. It has been previously mentioned that a refined mesh generally yields more accurate results, but is also more expensive in terms of calculation time. Therefore, the generation of a mesh is a trade-off between accuracy and calculation speed, where the smaller grid cells should be placed in critical regions in order to capture the most relevant characteristics and to preserve good quality.

A grid can be classified in two categories based on topology: Structured and unstructured grids. Compared to structured grids, unstructured grid cells are connected randomly with each other. As a result, a computer cannot see unstructured grids as arrays (either two- or three-dimensional, depending on the engineering problem) (Design Engineering, 2017). In terms of accuracy and complexity, both types have their advantages and disadvantages. Complex geometries are involved in many engineering problems, and an unstructured mesh has an advantage over a structured mesh in handling these complex geometries, since it can be difficult for a complex geometry to fit in a structured (e.g. Cartesian) grid. In addition, the creation of an unstructured grid is more straightforward and adaptions and refinements can be made much more easy compared to structured grids. Therefore, unstructured grids are considered to be the most popular meshing technique (Versteeg and Malalasekera, 2007). However, a structured mesh has a better accuracy for viscous calculations compared to an unstructured mesh, which is favorable for near-wall flows (Sadrehaghighi, 2017). In addition, a structured mesh provides a higher degree of control by the user, since unstructured meshing techniques are highly automated (Design Engineering, 2017).

Another type of meshing involves the combination of a structured and unstructured mesh, called a hybrid mesh. An example is shown in Figure 2-10, where the structured (quadrilateral) mesh is used in the vicinity of the cylinder walls, and the unstructured (triangular) mesh is used elsewhere. The advantage of a hybrid mesh is that viscous effects at the cylinder surface are properly preserved and modeled, while computer resources are used effectively. Related to finned tube heat transfer, literature does not come to an agreement on which grid type should be used. However, due to an increase in computing power over the last decades, a shift towards the increased use of hybrid grids is observed (van Dijk, 2019).

![Figure 2-10: Example of a hybrid mesh, using both a structured grid in the vicinity of the cylinder walls, and an unstructured grid elsewhere (image from Versteeg and Malalasekera (2007)).](image-url)
2-5-4 Turbulence Models

Reynolds-Averaged Navier-Stokes (RANS) Equations

It has been previously mentioned that flow over solid annular-finned tubes cause complex flow behavior. Despite the simple geometry, flow around annular-finned tubes can be steady or unsteady, laminar or turbulent, and (momentum and thermal) boundary layers develop and are destroyed on the tube and fin surface (Şahin et al., 2006), (Mon and Gross, 2004), (Sung et al., 1996). This complex flow behavior is shown schematically in Figure 2-11.

![Figure 2-11: Complex flow behavior caused by solid annular-finned tubes (image from Sung et al. (1996)).](image)

The presence of turbulent flow is one of the aspects which makes flow around solid annular-finned tubes complicated to solve, since turbulent flows cannot be solved exactly. Therefore, turbulence models exist to solve turbulent flows numerically in CFD. Solving turbulent flows can be done according to three different methods, from which the method based on the Reynolds-Averaged Navier-Stokes (RANS) equations is the most popular. This finding has been supported over the course of the literature survey, where it was observed that all literature sources studied related to finned tube heat transfer used turbulence models based on the RANS equations (van Dijk, 2019).

Because turbulent flows are random, it is impossible to give an easy description of the fluid flow. Instead, flow properties are decomposed into a (steady) mean value, and a value which fluctuates over time. This is called the Reynolds decomposition and can be seen in Equation 2-43 for a general variable $\phi$:

$$\phi(t) = \Phi + \phi'(t) \quad (2-43)$$

The same principle holds for actual flow properties such as $u$, $v$, $w$, $p$, etc. When substituted in the unsteady Navier-Stokes equations for incompressible flow, the resulting equations are the time (or Reynolds)-Averaged Navier-Stokes (RANS) equations. In these equations extra terms appear, which are called Reynolds stresses. These extra terms include three normal stresses: $\tau_{xx} = -\rho u'^2$, $\tau_{yy} = -\rho v'^2$, $\tau_{zz} = -\rho w'^2$, and three shear stresses: $\tau_{xy} = \tau_{yx} = \rho u'v'$, $\tau_{xz} = \tau_{zx} = \rho uw'$ and $\tau_{yz} = \tau_{zy} = \rho vw'$ (Versteeg and Malalasekera, 2007).
The Reynolds stress tensor which includes the Reynolds stresses cannot be "closed" since the RANS equations contain unknown correlations of fluctuations of flow properties. To achieve closure, additional equations are needed to solve the Reynolds stresses. The additional equations result in sufficient equations to solve all unknown variables. The (number of) equations used to achieve closure characterize the type of turbulence model. For instance, the mixing length model uses zero additional equations, the Spalart-Allmaras model (often encountered in the aeronautical industry) one additional equation, k-\(\epsilon\) model and k-\(\omega\) model two additional equations, and the Reynolds Stress Model (RSM) seven additional equations (Versteeg and Malalasekera, 2007). The k-\(\epsilon\) model is considered to be on of the most popular turbulence models, since this type of turbulence model has good numerical stability and convergence. Also, the k-\(\epsilon\) model has a well defined region of capability, meaning that it is relatively easy to investigate for which engineering problems the k-\(\epsilon\) model would or would not converge and give useful results (Versteeg and Malalasekera, 2007). Additions to the standard k-\(\epsilon\) model, such as the realizable k-\(\epsilon\) model and Re-Normalization Group (RNG) k-\(\epsilon\) model make the k-\(\epsilon\) model applicable for a wide variety of engineering problems, including swirling flows, low Reynolds number flows, and flows with strong streamline curvatures, which is the case related to flow around finned tubes. These flows can also be modeled accurately with the RSM, although more computational resources are required compared to k-\(\epsilon\) based models.

Turbulent modeling based on the RANS equations has been the most popular method for over three decades, since good accuracy can be achieved for fluid dynamics and heat transfer problems while computing resources remain moderate (Versteeg and Malalasekera, 2007). However, because of the developments which have been made in the field of computer performance in recent years, two other methods to calculate turbulent flows have been developed which use other principles than the RANS equations. In spite of these developments, no literature related to heat transfer and flow around finned tubes could be found (van Dijk, 2019). Still, since these two methods might be of interest for further research, both methods will be briefly touched upon.

**Large Eddy Simulation (LES)**

The Large Eddy Simulation (LES) method is another method to model turbulent flows. Instead of time-averaging the Navier-Stokes equations, The LES method conducts a filtering process of the unsteady Navier-Stokes equations, cutting off all the eddies (rotational flow structures) with a length scale which is smaller than a user-specified cutoff width. Larger eddies which pass the cutoff criteria are resolved (so the behavior of large eddies are tracked and actual unsteady flow equations have to be solved). However, the smaller unresolved eddies which have been filtered out interact with the larger resolved eddies which results in so called Sub Grid Scale (SGS) stresses. These stresses are not identical but similar to the Reynolds stresses corresponding to the RANS equations. Numerous SGS models exist to solve the unresolved stresses numerically (Versteeg and Malalasekera, 2007).
Direct Numerical Simulation (DNS)

Where the LES method can be seen as an intermediate solution since only the large-scale eddies are resolved, the Direct Numerical Simulation (DNS) method computes the mean flow and all eddies. This is even done for the eddies of the Kolmogorov length scales, which are the smallest length scales present in a turbulent flow. For this length scale, viscous and inertial effects are considered to be of equal strength, which is why dissipation by viscosity takes place (Versteeg and Malalasekera, 2007).

The DNS method resolves the flow by solving the unsteady Navier-Stokes equation. Together with the continuity equation, this results in a system of four equations and four unknown variables $u, v, w,$ and $p$. Then, a solution is developed which makes use of an extremely fine spacial grid and extremely small time steps. These requirements make sure that even the smallest eddies and fastest fluctuations can be resolved (Versteeg and Malalasekera, 2007). It may come as no surprise that the use of the DNS method to solve engineering problems is a very costly process in terms of calculation time and computer resources.

2-5-5 Wall Treatment

In the vicinity of the fin and tube surface, effects such as viscous damping (since the velocity is sufficiently low for viscous stresses to become dominant over inertial stresses) and conditions like the no-slip and no-penetration boundary conditions cause large mean velocity gradients throughout the near-wall region which promotes the creation of turbulent kinetic energy. More importantly, the near-wall region is also the location where momentum and heat is transported the most actively. Therefore, great care should be taken in order to model the near-wall region properly. During the literature survey, it has been previously mentioned that RANS equations based k-ε models are the most applied turbulence models for solving flow and heat transfer problems regarding finned tubes. However, the standard k-ε model is essentially valid for turbulent flows only. This means that boundary layers and adverse pressure gradients cannot be modeled properly (Versteeg and Malalasekera, 2007). Fortunately, developments led to modified versions of the k-ε model, from which one is called the realizable k-ε model.

Compared to the standard k-ε model, the realizable k-ε model can be used for low Reynolds number flows (which is what is expected for the flue gas flow between the fins) and it has a superior ability to grasp flow structures involving boundary layers and boundary layer separation, strong adverse pressure gradients, and recirculation. It is therefore no surprise that most of the literature related to flow around finned tubes utilize the realizable k-ε model (McIlwain, 2010), (Nemati and Moghimi, 2014), (Bacellar et al., 2014), (Chen et al., 2018).

A near-wall region can be divided into three distinct regions, depicted in Figure 2-12. In this Figure, $u^+$ and $y^+$ represent the normalized velocity and normalized wall distance respectively. The region closest to the wall is called the viscous sublayer, which is profoundly thin ($y^+ < 5$). In the viscous sublayer, viscous forces are dominant over inertial forces in terms of momentum and heat transfer, and after some assumptions and algebra it applies that:

$$u^+ = y^+$$  \hspace{1cm} (2-44)
The next layer encountered with increasing wall distance is called the log-law layer ($30 < y^+ < 500$). In the log-law layer, Reynolds stresses dominate the flow and the relationship between $u^+$ and $y^+$ is given by:

$$u^+ = \frac{1}{\kappa} \ln(y^+) + B$$  \hspace{1cm} (2-45)

Variable $\kappa$ is called the von Karman’s constant with a value of $\kappa = 0.4$, and $B$ is a constant which is $B = 5.5$ for smooth walls (Versteeg and Malalasekera, 2007). The region between the viscous sublayer and the log-law layer is called the buffer layer, where viscosity effects and inertial forces are of equal importance and magnitude (Versteeg and Malalasekera, 2007). The last layer encountered is the outer layer, where the flow is sufficiently far from the wall to be influenced by viscous effects.

Although the realizable $k-\epsilon$ model is able to capture boundary layers and boundary layer separation, it still requires near-wall treatment. This can be taken into account in two ways:

- **Near-wall model approach:** All layers influenced by viscous forces are resolved all the way down to the wall. This means that the first cell height has to be $y^+ < 5$ in order for the numerical model to solve the small viscous sublayer, and it is preferred that the first cell height has a value of $y^+ = 1$. It is important that the turbulence model used for the numerical model should still be valid through the near-wall region. Because of this criterion, low Reynolds number turbulence models should be used (Murad, 2018).

  It can be imagined that the near-wall model approach results in a substantial amount of grid cells, which leads to a significant increase in computational resources. A schematic of the near-wall model approach is shown in Figure 2-13.
Wall function approach: It can be realized that when the Reynolds number is increased, the thickness of the viscous sublayer decreases to such an extent that it becomes nearly impossible for the near-wall model approach to resolve. Instead of resolving the inner region, so-called wall functions can be used, which link the otherwise very difficult to model region between the wall corresponding to the geometry of interest and the turbulent region (Murad, 2018).

The main advantage of the wall function approach over the near-wall modeling approach is the reduction in computational resources, since the boundary layer is not resolved anymore but can be modeled with a relatively coarse mesh. The first cell should be placed in the log-law region to ensure sufficient accuracy (Murad, 2018). In addition, high Reynolds number turbulence models like the standard k-ε model are still valid. A schematic of the wall function approach is shown in Figure 2-14.

It has been previously mentioned in the literature survey report that literature related to the numerical investigation of heat transfer and flow characteristics around finned tubes does not come to an agreement on which wall treatment should be used. Both the use of the near-wall modeling approach and the wall function approach have been reported (van Dijk, 2019).
2-5-6 Boundary Conditions

The selection of appropriate initial and boundary conditions is of absolute importance during the development of any numerical model. When these conditions are not chosen correctly, it is almost certain that the numerical model will not be able to obtain physically relevant results. Multiple types of boundary conditions exist. From this selection, the boundary conditions which are considered to be the most relevant for this master thesis research are:

- **Inlet boundary condition:** At the inlet boundary of the computational domain, the values of relevant flow properties (e.g. velocity, pressure, and temperature) need to be specified. Corresponding to the specification of velocity, an uniform flow distribution is often imposed at the inlet of the computational domain.

- **Wall boundary condition:** At the wall of the geometry, wall boundary conditions have to be imposed in order for the numerical model to produce plausible results. Wall boundary conditions are considered to be highly important since almost any engineering problem has to deal with wall boundary conditions. At the wall, the value of both the normal and tangential velocity component should be zero (no penetration boundary condition and no-slip boundary condition).

- **Symmetry boundary condition:** Symmetry boundary conditions are convenient if the geometry and expected flow field can be mirrored along the symmetry plane. This reduces the computational domain and therefore computational resources. It applies that fluxes and the flow across the boundary of the symmetry plane should be zero (Versteeg and Malalasekera, 2007).

  It has been observed during the literature survey that symmetry boundary conditions can be of great use for the master thesis research during the development of the numerical models. For example, symmetry boundary conditions could decrease the number of grid cells dramatically by modeling an entire tube row as a small tube slice with only one fin attached. By application of the symmetry boundary condition one more time along a different axis, this small slice could modeled by a half-circle tube slice with only one fin.

- **Periodic/Cyclic boundary condition:** When a flow around the geometry of interest is of periodically repeating nature, periodic boundary conditions could be used. To apply periodic boundary conditions, the flux of all relevant flow properties which leave the outlet cyclic boundary must be set equal to the flux which enters the inlet cyclic boundary (Versteeg and Malalasekera, 2007). This could be of particular interest to investigate flow around finned tubes for multiple tube rows.
Chapter 3

Research Methodology

3-1 Introduction

In the previous chapter, the most relevant literature related to finned tube heat transfer from the literature survey has been touched upon again. This includes the working principle of fired heaters and the corresponding ESCOA correlation method used to determine heat transfer and pressure drop in the convection section of fired heaters. In addition, the analytical method has been proposed and discussed, including the assumptions which have been made. To conclude, various aspects necessary for the development of a numerical model have been summarized. This was done to provide a better understanding of the subsequent chapters in this master thesis report.

In this chapter, the research methodology for the development of the numerical model will be introduced. The numerical model will be developed in ANSYS Fluent (version 19.2). ANSYS Fluent is a commercial software package which is used to develop and solve numerical problems related to heat transfer and fluid dynamics. It is an acclaimed program and has been widely used for both academic and industrial purposes. This recognition is supported by the literature survey, where it was found that all the literature which has been studied adopted ANSYS Fluent (van Dijk, 2019).

The research methodology introduced in this chapter will follow the same order used by ANSYS Fluent during the development of a numerical model. As a result, the methodology related to the development of the geometry of interest will be introduced first in Section 3-2-1. Second, the methodology used for the generation of the mesh for the corresponding geometry will be given in Section 3-2-2. After completion of the previous two steps, information related to the setup and settings which are adopted by the numerical models will be provided in Section 3-2-3.
3-2 Methodology Applied to Numerical Models

3-2-1 Geometry

The determination of the geometry of interest is the first step which has to be taken during the development of a numerical model. The investigation of flow around solid annular-finned tubes is a three-dimensional problem and cannot be modeled with a two-dimensional model. This has a significant impact on the computational resources required. Therefore, a computational domain similar to Mon and Gross (2004) has been proposed, from which the boundary of the computational domain can be observed by the dashed lines in Figure 3-1 and Figure 3-2. As a result, this allows the tube rows to be modeled by a computational domain with a height of half the transverse tube pitch \( t_q \), and a width which is the sum of half the fin thickness \( s_R \) and fin spacing \( S_{fin} \).

![Figure 3-1: Side view of the adopted computational domain (image from Mon and Gross (2004)).](image)

![Figure 3-2: Top view of the adopted computational domain (image from Mon and Gross (2004)).](image)

Over the course of this master thesis research, three investigations have been conducted in order to draw conclusions about the validity of the analytical method. The corresponding geometries will use the same methodology, but will differ slightly from each other. The first investigation will evaluate the values of the heat transfer and pressure drop coefficient relative to the bare tube area \( U_{bare} \) and \( \zeta \) versus a varying fin density \( N_f \). This has been done for one tube row and for three different tube sizes: 4" tubes, 6" tubes, and 8" tubes. The second investigation examines the values of \( U_{bare} \) for multiple tube rows, and is conducted for 4" tubes and 6" tubes. To conclude, the third investigation includes a study for one tube row for 6" tubes and 8" tubes related to the values of \( U_{bare} \) for two different fin configurations. These two fin configurations which will be discussed more elaborately in Chapter 4.
3-2 Methodology Applied to Numerical Models

Prior to the investigation of multiple tube rows or different fin configurations, the values of $U_{bare}$ and $\zeta$ for one tube row should be examined first. The proposed geometry is shown in Figure 3-3 and Figure 3-4, and the corresponding dimensions for 4” tubes, 6” tubes, and 8” tubes can be found in Table 3-1 and Table 3-2.

The dimensions defined in Figure 3-3 and Figure 3-4 (i.e. $D_1$, $D_2$, $D_3$, $D_4$) represent the inner tube diameter, outer tube diameter $d_A$, outer tube diameter including the fins $D = d_A + h$ and the diameter of the fin fluid domain respectively. $H_1$ and $H_2$ stand for the length of the outside fin fluid domain and the length of the downstream fluid domain. $V_1$ represents the height of the outside fin fluid domain and downstream fluid domain and is equal to the transverse tube pitch $t_q$. To conclude, the thickness of the computational domain is described by $Z_1$.

The geometry of interest has been designed with fixed ratios in mind in order to minimize errors which may arise due to the scale-up of the tubes. Regardless of the tube size investigated, it holds that $H_1 = 2D_4$, $H_2 = 4D_4$, and $V_1 = t_q$, which is shown in Table 3-1. However, two variables remain constant for an increasing tube size: The fin height $h$ and the half-fin thickness $s_R/2$. The values of $h$ and $s_R$ can be seen in Table 2-1 and 2-2 in Chapter 2. In Table 3-2 it is observed that the thickness of the computational domain $Z_1$ in Figure 3-4 decreases for an increased fin density $N_f$, which follows from the fact that an increase in $N_f$ leads to a reduction in fin spacing $S_{fin}$. 

Figure 3-3: Side-view of the proposed computational domain related to the investigation of one tube row.

Figure 3-4: Top-view of the proposed computational domain related to the investigation of one tube row.

Geometry related to the investigation of the heat transfer and pressure drop coefficient versus a varying fin density for one tube row

Thesis Report  J. van Dijk
Table 3-1: Dimensions used to define the geometry of interest related to the investigation of one tube row. Left: 4” tubes. Center: 6” tubes. Right: 8” tubes.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Variable</th>
<th>Value</th>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>D₁</td>
<td>4 [inch]</td>
<td>D₁</td>
<td>6 [inch]</td>
<td>D₁</td>
<td>8 [inch]</td>
</tr>
<tr>
<td>H₂</td>
<td>30 [inch]</td>
<td>H₂</td>
<td>38.5 [inch]</td>
<td>H₂</td>
<td>46.5 [inch]</td>
</tr>
<tr>
<td>V₁</td>
<td>4 [inch]</td>
<td>V₁</td>
<td>6 [inch]</td>
<td>V₁</td>
<td>8 [inch]</td>
</tr>
</tbody>
</table>

Table 3-2: Thickness of the computational domain $Z₁$ as a function of fin density $N_f$.

<table>
<thead>
<tr>
<th>Fin Density $N_f$</th>
<th>Thickness $Z₁$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 [inch$^{-1}$]</td>
<td>0.250 [inch]</td>
</tr>
<tr>
<td>3 [inch$^{-1}$]</td>
<td>0.167 [inch]</td>
</tr>
<tr>
<td>4 [inch$^{-1}$]</td>
<td>0.125 [inch]</td>
</tr>
<tr>
<td>5 [inch$^{-1}$]</td>
<td>0.100 [inch]</td>
</tr>
<tr>
<td>6 [inch$^{-1}$]</td>
<td>0.083 [inch]</td>
</tr>
<tr>
<td>7 [inch$^{-1}$]</td>
<td>0.071 [inch]</td>
</tr>
<tr>
<td>8 [inch$^{-1}$]</td>
<td>0.063 [inch]</td>
</tr>
</tbody>
</table>

The values of the process conditions are the same for all three investigations (unless stated otherwise) and are identical to the process conditions used to discuss the ESCOA correlation method and the analytical method. These values can be found in Table 2-1 and Table 2-2 in Chapter 2.

Geometry related to the investigation of the heat transfer coefficient for multiple tube rows

In addition to the investigation of one tube row, the effect of multiple tube rows on the value of $U_{bare}$ has also been studied. This investigation has been conducted for 4” tubes and 6” tubes, and for these tube sizes the number of tube rows $N_R$ is increased while the value of the fin density is remained constant, which has a value of $N_f = 4$ inch$^{-1}$. It has been previously mentioned that the process conditions used are identical to the process conditions used for the investigation of one tube row.

The geometry for the investigation of multiple tube rows is comparable to the geometry which has been made for the investigation of one tube row and follows the same methodology proposed by Mon and Gross (2004). The result is shown in Figure 3-5 and Figure 3-6, where the number of tube rows is $N_R = 2$ to demonstrate as an example. Regardless of the tube size investigated it still holds that $H₂ = 4D₄$ and $V₁ = t_q$ However, with the addition of an additional tube row, the length of the fluid domain is now expressed by $H₁ = 2D₄ + t_l(N_R−1)$. 

J. van Dijk

Thesis Report
During the literature survey, it has been discussed that tubes in a staggered arrangement are up to one-and-a-half-times more effective in terms of heat transfer compared to an in-line tube arrangement (van Dijk, 2019). Therefore, all numerical experiments related to the investigation of multiple tube rows have been conducted with a staggered tube arrangement shown in Figure 3-5 and Figure 3-6.

**Geometry related to the investigation of the heat transfer coefficient for two different fin configurations**

The third investigation involves numerical experiments where for two different configurations $U_{bare}$ is evaluated against an increasing average mass flux $G_{avg}$. One fin configuration has a fin density of $N_f = 5$ inch$^{-1}$ and fin height of $h = 1''$, while the other fin configuration has a fin density of $N_f = 2.5$ inch$^{-1}$ and fin height of $h = 2''$. According to API Standard 560, the fin height for finned tubes in the convection section of fired heaters is restricted to $h = 1''$ while the large longitudinal and transverse tube pitch $t_l$ and $t_q$ corresponding to 6'' tubes and 8'' tubes would allow for taller fins (American Petroleum Institute, 2001). As a result, this investigation will be conducted for 6'' and 8'' tubes only.

Despite both configurations roughly having the same extension ratio, the ESCOA correlation method and the analytical method calculate completely different results for both configurations. This makes it interesting to observe if the ESCOA correlation method or the analytical method calculates results which show the closest agreement with the results calculated by the numerical model.
The proposed geometries for both fin configurations will closely resemble the geometry proposed for the investigation of one tube row in Figure 3-3 and Figure 3-4. In fact, the geometry developed for the fin configuration with \( N_f = 5 \) inch\(^{-1} \) and \( h = 1'' \) will look identical to the geometry proposed for the investigation of one tube row, while the only difference with the geometry developed for the fin configuration with \( N_f = 2.5 \) inch\(^{-1} \) and \( h = 2'' \) will be an increase in fin height.

### 3-2-2 Mesh

With the geometry of interest defined for all three investigations, the next step during the development of a numerical model is to define a method related to the generation of the mesh. During mesh generation, the computational domain is subdivided into smaller domains. It has been previously mentioned that a more elaborate discretization process yields a finer mesh and therefore more accurate results. However, this comes at the expense of an increase in computational resources. Consequently, the mesh should be a trade-off between accuracy and calculation speed, where smaller grid cells should be placed in critical regions in order to capture relevant heat transfer and flow characteristics, while more coarser grid cells are used for less relevant regions in the computational domain. In addition, the mesh also depends on the type of turbulence model which is used. The turbulence adopted to perform all three investigations will be discussed more elaborately in Section 3-2-3, but is the realizable \( k-\epsilon \) model which is based on the RANS equations. As a result, the main requirement is that the numerical grid must be sufficiently refined in the vicinity of the tube and fin walls to represent and resolve the gradients in local mean quantities properly.

### Named Selections

Simultaneously with the generation of the mesh, appropriate names have to be assigned to all geometries and planes in order for the solver to understand the geometry well. The methodology adopted for the name selection process is the same for all three investigations, and an example from the investigation of one tube row is shown in Figure 3-7 and Figure 3-8. It can be observed in both figures that the fin and tube are the only two solid geometries present in the computational fluid domain, which are depicted by the diagonal stripes. For these two geometries, appropriate boundary conditions have to be selected to ensure that the flue gas exhibits the appropriate flow behavior near the walls.

The fluid domain is divided in three regions, which are all depicted by dotted areas: densely-dotted, medium-dotted, and lightly-dotted. The densely-dotted area represents the fin fluid domain, in other words the flow in the vicinity of the fin and tube. Compared to other areas, the mesh has to be refined in order to properly model the phenomena observed near the walls such as the growth and destruction of boundary layers. The medium-dotted area represents the flue gas flow outside the fin fluid domain, therefore called the outside fin fluid domain. The mesh in this area does not have to be as refined as was the case for the fin fluid domain, but should still be sufficiently small in order for the numerical model to obtain realistic results. The lightly-dotted area represents the downstream fluid domain. In this domain the flue gas has already passed the finned tube, and as a result mixing and dampening of turbulent flow takes place.
Figure 3-7: Side-view of the computational domain with the corresponding name selections.

Figure 3-8: Top-view of the computational domain with the corresponding name selections.

Figure 3-7 and Figure 3-8 also show the names which have been given to the planes to describe the boundaries of the computational domain. The plane on the left represents the inlet plane, where the flue gas enters the computational domain. The plane on the right is called the outlet plane, where the flue gas exits the computational domain. The other planes are all so-called symmetry planes. Symmetry planes essentially function as a mirror, where it applies that both the fluxes and the flow across the boundary are zero.

Meshing method related to all three investigations

A mesh is often non-uniform, which means that the grid is more refined for areas where large gradients in flow properties (e.g. in velocity, pressure, temperature) take place from point to point, and that the grid is more coarse for areas where little gradients in flow properties occur. The same applies to this master thesis research, where it has already been mentioned above that the total computational domain is subdivided in four regions: The fin and tube region, the fin fluid domain, the outside fin fluid domain and the downstream fluid domain.

All three investigations require a substantial amount of labor to complete, from which the investigation of $U_{bare}$ and $\zeta$ for one tube row stands out most. This can be attributed to the fact that for this investigation multiple grid sizes have been used: A coarse, medium and refined grid. This was done to observe changes in $U_{bare}$ and $\zeta$ which might occur due to an increase in the refinement of the mesh. If these values are sufficiently similar for two grid sizes (e.g. between the medium grid and refined grid), it can be concluded that the results calculated by the numerical models do not depend on the size of the grid elements anymore and that grid independence has been reached. For reaching solid results, this is a necessary condition. In addition to this, three tube sizes (4” tubes, 6” tubes and 8” tubes) have been studied during this investigation, and for each tube size the values of $U_{bare}$ and $\zeta$ were calculated versus an increasing fin density ranging from $N_f = 2 \text{ inch}^{-1}$ to $N_f = 8 \text{ inch}^{-1}$. This extensive research for the first investigation yields a total of 63 different numerical models. Adding up the numerical models necessary for the investigation of multiple tube rows and for
the investigation of two different fin configurations, a total amount of 95 different numerical models are attained. For this reason, the methodology adopted for the generation of the mesh cannot be explained for each numerical model in this section. Fortunately, all numerical models from all three investigations approximately use the same meshing methodology (occasionally some exceptions had to be made to prevent errors). Therefore, the methodology proposed for the generation of the mesh will be explained by one particular example: The generation of a medium grid for a 4" tube and a fin density of $N_f = 4$ inch$^{-1}$ related to the investigation of $U_{bare}$ and $\zeta$ for one tube row.

All grids have been developed in the meshing tool provided by ANSYS Fluent. Default settings have been used for all regions, with the addition of a meshing technique and sizing. The meshing technique and sizing for the fin and tube region is shown in Table 3-3. In this region, heat is transported from the tube (where a fixed tube temperature $T_{\text{tube}}$ has been imposed) to the fin surface by means of conduction. It is expected that the highest temperature gradients on the fin surface will be observed at the upstream side of the finned tube, since this is the location where the hot flue gas makes first contact with the relatively colder fin and tube surface. To observe this effect properly, the size of the grid elements have been made sufficiently small.

Table 3-3: Meshing method for the fin and tube region for a medium grid, 4" tube and fin density $N_f = 4$ inch$^{-1}$.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Size</td>
<td>$1.00 \times 10^{-3}$ [m]</td>
</tr>
<tr>
<td>Meshing Technique</td>
<td>Multizone [-]</td>
</tr>
</tbody>
</table>

Utilization of the multizone meshing technique leads to an additional collection of settings. The settings which have been changed from the default settings can be seen in Table 3-4 and are valid for all numerical models where the multizone meshing technique has been used. The multizone meshing technique combined with the settings chosen in Table 3-4 allows the fin and tube region to be decomposed in small blocks. Afterwards, a structured hexagonal mesh is generated where the geometry allows for this, while elsewhere the remaining part of the geometry is generated with an unstructured hexagonal mesh. The result is shown in Figure 3-9. It is worth noting that for 6" tubes the sweep meshing technique has been used instead of the multizone meshing technique to mesh the fin and tube region, which was necessary to prevent the generation of errors. No noticeable differences between both meshing techniques could be observed for the end result.

Table 3-4: Features corresponding to the multizone meshing method which have been changed from the default settings.

<table>
<thead>
<tr>
<th>Subject</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mapped Mesh Type</td>
<td>Hexa [-]</td>
</tr>
<tr>
<td>Surface Mesh Method</td>
<td>Program Controlled [-]</td>
</tr>
</tbody>
</table>
The meshing method of the fin fluid domain is shown in Table 3-5 and is similar to the meshing method proposed for the fin and tube region, with both regions using of the multizone meshing technique. However, to model heat transfer and flow characteristics between the fins properly, it is required that the grid element size of the fin fluid domain should be more refined compared to the element size corresponding to the fin and tube region.

Table 3-5: Meshing method for the fin fluid domain for a medium grid, 4" tube and fin density $N_f = 4$ inch$^{-1}$.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Size</td>
<td>$6.00 \times 10^{-4}$ [m]</td>
</tr>
<tr>
<td>Meshing Technique</td>
<td>Multizone [-]</td>
</tr>
</tbody>
</table>
More important is the need for inflation layers for the fin fluid domain. Inflation layers are flat grid cells with high aspect ratios, stacked on top of each other at the wall of the geometry of interest. These special grid cell layers are necessary in order to properly model the flow in the vicinity of walls. The same applies to this master thesis research, where it has been previously mentioned that the formation and destruction of boundary layers on the fin and tube surface can be observed (van Dijk, 2019).

The methodology on how inflation layers have been implemented in the mesh is identical for all the other numerical models and can be divided in three locations. The first location where inflation layers have to be placed is on the tube surface. The second location where inflation layers should be placed is on the side surface of the fin. To conclude, the third location where inflation layers should be placed is on the fin tip surface. As a result, the complete fin and tube geometry which makes contact with the fin fluid domain is enclosed by inflation layers.

For all numerical models from all investigations, the first layer thickness was chosen in such a way that the value of the normalized wall distance remained sufficiently small enough to use the near-wall model approach, which means that $y^+ < 1$. The value of the first layer thickness and the values other settings can be seen in Table 3-6. According to Versteeg and Malalasekera (2007), the first layer thickness can be estimated by:

$$y^+ = \frac{\Delta y \rho}{\mu \sqrt{\frac{\tau_w}{\rho}}}$$ (3-1)
Where $\Delta y$ and $\tau_w$ denote the first layer thickness and wall shear stress respectively. The latter can be estimated by (Mills, 1999):

$$\tau_w = C_f \cdot \frac{1}{2} \rho V^2$$  \hspace{1cm} (3-2)

Here, $C_f$ stands for the skin friction coefficient which can be estimated by the Blasius solution (assuming that the flow between the fins is laminar) (Mills, 1999):

$$C_f = \frac{0.664}{\sqrt{Re_x}}$$  \hspace{1cm} (3-3)

By setting $y^+ = 1$, a value of the first layer thickness of $\Delta y \approx 0.6 \cdot 10^{-4}$ m was calculated for this particular example. However, the Blasius solution used in Equation 3-3 assumes flow over a flat plate which is not the case for the finned tube geometry. It is certain that this assumption will result in a different value of the boundary layer thickness, and therefore result in a different value of the first layer thickness $\Delta y$. Since all numerical models are unique, the first layer thickness $\Delta y$ also changes for each numerical model, since all numerical models are distinct. To accommodate for this, the value of the first layer thickness which has been selected in Table 3-6 is considerably lower compared to the value calculated according to Equation 3-1 to ensure that $y^+ < 1$.

It has to be mentioned that for some cases a few adjustments had to be made. Related to the investigation of $U_{bare}$ and $\zeta$ for one tube row for example, values of the fin density greater than $N_f > 6$ inch$^{-1}$ required a reduction in the number of inflation layers in order to prevent the fin fluid domain from being completely filled with inflation layers. Related to the investigation of two different fin configurations, the increasing average mass flux $G_{avg}$ led to an adjustment to reduce the height of the first layer thickness to ensure that $y^+ < 1$.

The result of the generated mesh for corresponding to the fin fluid domain is shown in Figure 3-10. The small inflation layers are represented by the thin black lines surrounding the tube and fin surface.
The meshing method related to the outside fin fluid domain is identical to the meshing method proposed for the fin and tube region and can be observed in Table 3-7. The flow in this domain does not require a grid which is as refined as the grid used for the fin fluid domain, which is where the most relevant heat transfer and flow characteristics take place. As a result, the grid element size chosen for the outside fin fluid domain is identical to the element size corresponding to the fin and tube region for this example, but this does not have to be the case for the other numerical models. The result of the mesh for the outside fin fluid domain is shown in Figure 3-11.

Table 3-7: Meshing method for the outside fin fluid domain for a medium grid, 4" tube and fin density $N_f = 4$ inch$^{-1}$.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Size</td>
<td>$1.00 \times 10^{-3}$ [m]</td>
</tr>
<tr>
<td>Meshing Technique</td>
<td>Multizone [-]</td>
</tr>
</tbody>
</table>

Figure 3-11: Side-view representation of the generated mesh for the outside fin fluid domain.

The rectangular shape of the downstream fluid domain allowed for the use of a slightly different meshing technique compared to the other regions, which is shown in Table 3-8 and Figure 3-12. Compared to the other fluid domains, the grid of the downstream fluid domain can generated by a Cartesian grid which divides the domain in rectangular blocks. In addition, the size of the grid elements chosen for the downstream fluid domain are larger compared to the grid elements of the other regions since the downstream flue gas flow is considered to be the least relevant.

It is worth noting that not all numerical models allowed for the generation of a Cartesian grid for the downstream fluid domain. To resolve this issue, a multizone grid was generated for these cases. Between the two different meshing techniques, no differences could be observed for the end result.

Table 3-8: Meshing method for the downstream fluid domain for a medium grid, 4" tube and fin density $N_f = 4$ inch$^{-1}$.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Size</td>
<td>$2.00 \times 10^{-3}$ [m]</td>
</tr>
<tr>
<td>Meshing Technique</td>
<td>Cartesian [-]</td>
</tr>
</tbody>
</table>
After all regions have been integrated in the final grid, the results related to the quality of the mesh obtained for this particular example are shown in Table 3-9. Two parameters which for a significant part determine the quality of the mesh have been evaluated. These are the minimum orthogonal quality and the maximum aspect ratio. The minimum orthogonal quality is a measure to determine how much a grid cell differs from orthogonality when looking at the faces of the corresponding grid cell (ANSYS Inc., 2013a). Values for the orthogonal quality range from 0 to 1, where 0 stands for a poor mesh quality while 1 corresponds to a high mesh quality. The maximum aspect ratio represents the measure of stretching of a cell. No documentation related to the ideal value of the maximum aspect ratio has been found, but values of order $\sim 10$ are common practice in CFD studies and are often encountered near the wall of the geometry (ANSYS Inc., 2013a). It can be observed in Table 3-9 that a relatively low, but acceptable value for the minimum orthogonal quality (due to the round geometry of the finned tube) and a relatively high, but also acceptable value for the maximum aspect ratio (due to the implementation of inflation layers) have been obtained.

The student licence of ANSYS Fluent is limited to generate a maximum of 500,000 grid cells. It has been previously mentioned that the investigation of $U_{bare}$ and $\zeta$ for one tube row involves three different grid sizes: A coarse ($\approx 100,000$ grid cells), medium ($\approx 500,000$ grid cells), and refined grid ($\approx 2,000,000$ grid cells). To be able to also compute the numerical models for which a refined grid has been generated, a PC with the full ANSYS Fluent license was made available. This was not necessary for the investigation of multiple tube rows and for the investigation of two different fin configurations, which have been studied just by application of a medium grid.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum Orthogonal Quality</td>
<td>0.563976 [−]</td>
</tr>
<tr>
<td>Maximum Aspect Ratio</td>
<td>14.6194 [−]</td>
</tr>
<tr>
<td>Number of Grid Cells</td>
<td>465567 [−]</td>
</tr>
</tbody>
</table>

**Contact Zones**

Several contact zones can be observed when all regions are integrated in the final grid. Contact zones are the areas where the boundary of the grid from a region makes contact with the boundary of the grid from another region. Contact zones which can be identified in the integrated grid are:
• **Contact zone 1**: Tube to fin

• **Contact zone 2**: Tube to fin fluid domain

• **Contact zone 3**: Fin to fin fluid domain

• **Contact zone 4**: Fin fluid domain to outside fin fluid domain

• **Contact zone 5**: Outside fin fluid domain to downstream fluid domain

Contact zones consistently include two regions. Contact zone 1 is the only contact zone with a solid-solid interface. Contact zone 2 and contact zone 3 are contact zones with a liquid-solid interface, while contact zone 4 and contact zone 5 are contact zones with a liquid-liquid interface. During the setup, the appropriate mesh interface settings are selected in order to treat these interfaces properly.

### 3-2-3 Setup

Selection of the appropriate settings for the numerical models is the final step prior to executing the calculations. The amount of settings are comprehensive, but the most significant settings include choosing between transient or steady-state modeling, selection of turbulence models with appropriate wall treatment settings, specification of the material properties of the flue gas and finned tube, selection of solution algorithms, specification of convergence criteria, etc. These have to be chosen correctly in order for the numerical models to produce relevant results. A fraction of these settings have already been investigated during the literature survey, while the remaining settings have been investigated during the development of the numerical models with support from the ANSYS Fluent theory and user guide (ANSYS Inc., 2013a), (ANSYS Inc., 2013b).

It has been previously mentioned that a substantial amount of numerical models have been developed to encompass all three investigations. However, in contrast with the methodology proposed for the generation of the mesh, the setup for all numerical models is almost identical, with the only exception being the varying value of the inlet velocity. As a result, the setup does not have to be illustrated by one particular example which eases the explanation process.

### Steady State and Transient Modeling

The first aspect which has to be addressed during the setup is to determine if all numerical models can be simulated by using a steady state or transient model. For certain values of the Reynolds number, a vortex street is formed (Mills, 1999). Vortex streets are non-symmetrical and time-dependent, which can therefore not be simulated by a steady state model. After calculation of the Reynolds number based on the outer tube diameter $d_A$ for some examples, it was found that the values are indeed in the region where vortex streets are formed. As a result, transient simulations have been conducted to observe if this was the case.
The extensive amount of numerical models make it impossible to observe this for every single case. Therefore, two limit cases have been investigated: A case where the value of the Reynolds number is the lowest, and a case where the value of the Reynolds number is the highest. It is hypothesized that if these limit cases are time-independent, the other numerical models should be time-independent as well. For all investigations, the lowest and highest values of the Reynolds number which have been calculated are \( Re_{min} \approx 2400 \) and \( Re_{max} \approx 18000 \) respectively. To determine if these two particular cases are time-independent, transient simulations have been conducted. During these simulations, the lift coefficient \( C_l \) has been kept under close observation. For a numerical model to be steady state, the value of the lift coefficient \( C_l \) should remain constant and should not fluctuate or oscillate over time. In addition to this, animations of the \( Re_{min} \) and \( Re_{max} \) simulations have been made to observe if the simulation is actually time-independent.

The setup used to study these two cases is identical to the setup used to investigate all the other numerical models, which will be explained further in this section. Extra settings which are the result of transient modeling are shown in Table 3-10.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time Step</td>
<td>0.05 [s]</td>
</tr>
<tr>
<td>Number of Time Steps</td>
<td>100 [-]</td>
</tr>
<tr>
<td>Number of Iterations per Time Step</td>
<td>50 [-]</td>
</tr>
</tbody>
</table>

The outcome of the transient \( Re_{min} \) and \( Re_{max} \) simulations displayed no transient behavior. Both simulations calculated a value of the lift coefficient \( C_l = 0 \), which should come as no surprise since the finned tube geometry is symmetrical and should not produce lift, unlike an airfoil. In addition, no transient behavior was observed in the animations made for both simulations during post-processing. Although the values of \( Re_{min}, Re_{max} \) are in the range of where transient behavior is expected, it is believed that the effect of other finned tubes represented by the symmetry planes suppress these oscillations. As a result, it has been concluded that the remaining numerical models should also display no transient behavior, and that all numerical models can be modeled by means of a steady-state simulation.

**Turbulence Models and Near-Wall Treatment**

After completion of the literature survey, two turbulence models were considered: The realizable \( k-\epsilon \) model and the Transition SST model. Both methods should be able to accurately resolve low Reynolds number flows, flows with strong streamline curvatures, and flows with adverse pressure gradients. It is realized that more models are capable of resolving these flows, but the reason why the realizable \( k-\epsilon \) model and the Transition SST model have been chosen in particular is twofold. First, the Transition SST model showed the closest agreement with experimental data related to solid annular-finned heat transfer (Nemati and Moghimi, 2014). Second, it was observed that the realizable \( k-\epsilon \) model was used by the majority of the literature which has been studied. It has already been mentioned during the literature survey and in Section 2-5 that this can be attributed to the good numerical stability, convergence, and well defined region of capability.
To determine which of these two turbulence models should be preferred, the results obtained by both the realizable k-\(\epsilon\) model and Transition SST model (both have been conducted on a coarse grid) have been evaluated by means of one particular example: a 4" tube with \(G_{avg} = 2\) kg/m\(^2\cdot\)s versus a varying fin density \(N_f\). This example has been chosen because it is known through practice that the ESCOA correlation method produces valid results for 4" tubes and for common fin density and average mass flux values (i.e. \(N_f = 4\) inch\(^{-1}\) and \(G_{avg} = 2\) kg/m\(^2\cdot\)s). The results of \(U_{bare}\) and \(\zeta\) calculated by the ESCOA correlation method and the numerical models are shown in Figure 3-13 and Figure 3-14 respectively. Again, the setup used to investigate both turbulence models is similar to the setup used to investigate all subsequent numerical models.

Both figures already show a glimpse of the results obtained, which will be discussed elaborately in Chapter 4. It can be observed in Figure 3-13 that for the common fin density and average mass flux values mentioned above the values of \(U_{bare}\) calculated by both turbulence models are in good agreement with the values calculated by the ESCOA correlation method. This agreement is less for \(\zeta\) shown in Figure 3-14 and the reason for this will be discussed in Chapter 4. In addition, it can be observed that \(U_{bare}\) and \(\zeta\) calculated by the realizable k-\(\epsilon\) model and Transition SST model do not disagree much. This is in agreement with Nemati and Moghimi (2014), where it was concluded that it is primarily the length of the wake behind the last tube in the finned tube bank which is different, and not the flow field between the tubes in the finned tube bank which was found to be similar during the investigation of various turbulence models (including the realizable k-\(\epsilon\) model and Transition SST model).

After observation of both Figure 3-13 and Figure 3-14, it can be concluded that deciding between using the realizable k-\(\epsilon\) model or the Transition SST model will not have a dramatic impact on the results which will be obtained from the subsequent simulations. As a result, the selection of the turbulence model will rather depend on the literature studied to have more comparison data available for the results calculated by the numerical models. By means of this criterion, it is evident that the realizable k-\(\epsilon\) model has a clear advantage over the Transition SST model, which ultimately leads to the justification of the use of the realizable k-\(\epsilon\) model for this master thesis research.

The selection of the realizable k-\(\epsilon\) turbulence model results in an additional set of options, which are related to near-wall treatment. It has been previously mentioned that near-wall treatment can be divided in two approaches. The first approach is the near-wall model approach, where the flow affected by viscosity is resolved all the way down to the wall. This is not the case for the second approach, where wall functions are used to link the fully turbulent flow and the flow which is affected by viscosity. It is recommended that for k-\(\epsilon\) based turbulence models Enhanced Wall Treatment (EWT) should be used (ANSYS Inc., 2013a). EWT is a near-wall modeling approach which uses a combination a two-layer model together with Enhanced Wall Functions (EWF). The two-layer model divides the total domain into a region which is viscosity affected and into a region which is fully turbulent. To determine which region is viscosity affected and which one is not, a variable called the turbulent Reynolds number \(Re_y\) is introduced. The turbulent Reynolds number \(Re_y\) is based on the distance from a field point in the domain to the wall, and for a certain threshold value of \(Re_y\) a region is assigned to be viscosity affected or fully turbulent. As a result, turbulence models (e.g. two-equation realizable k-\(\epsilon\) model) are applied for the fully turbulent region while a one-equation model is used for the viscosity affected region. In addition, a dampening function is used to provide smooth transition between the two regions (ANSYS Inc., 2013a).
Figure 3-13: Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ versus the fin density $N_f$ for 4" tubes, calculated by the ESCOA correlation method and by the realizable $k$-$\epsilon$ and Transition SST models on a coarse grid. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s.

Figure 3-14: Comparison of the pressure drop coefficient relative to the bare tube area $\zeta$ versus the fin density $N_f$ for 4" tubes, calculated by the ESCOA correlation method and by realizable $k$-$\epsilon$ and Transition SST models on a coarse grid. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s.
During the definition of the methodology related to the generation of the mesh, it was already mentioned that the grid required to achieve a certain accuracy will also depend on the type of turbulence model which is used. The same applies to the realizable $k$-$\varepsilon$ turbulence model in combination with EWT. It has already been mentioned that EWT also makes use of EWF, which are functions to formulate the law-of-the-wall (viscous sublayer, buffer layer, and log-law region) as one blended, law-of-the-wall (ANSYS Inc., 2013a). The fact that EWT uses both the two-layer model and EWF makes it applicable for a wide range of $y^+$ values, which is one of main reasons why EWT is recommended in combination with $k$-$\varepsilon$ based turbulence models. However, to accurately predict heat transfer in the presence of boundary layers, values of $y^+ \approx 1$ are still recommended (ANSYS Inc., 2013b). For these small values of $y^+$, the mesh is sufficiently refined to resolve the viscous sublayer and the EWT is identical to the two-layer approach and no EWF are used.

Material Properties

The material properties of the flue gas and steel remains constant for all numerical models and are identical to the material properties used for both the ESCOA correlation method and the analytical method. The values corresponding to these material properties are shown in Table 2-1 in Section 2-3. The values which have been entered in the materials tab in ANSYS Fluent are shown in Table 3-11. The density of the flue gas is not an input parameter but is calculated by ANSYS Fluent according to the ideal gas law (Equation 2-29).

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_p$</td>
<td>1063.5 [J/kg·K]</td>
</tr>
<tr>
<td>$\mu$</td>
<td>$3.16 \times 10^{-5}$ [kg/m·s]</td>
</tr>
<tr>
<td>$k_{gas}$</td>
<td>0.04916 [W/m·K]</td>
</tr>
<tr>
<td>$k_{steel}$</td>
<td>46.737 [W/m·K]</td>
</tr>
<tr>
<td>$M_{gas}$</td>
<td>28.0 [kg/kmol]</td>
</tr>
</tbody>
</table>

Initial and Boundary Conditions

Similar to the specification of the material properties, the values of the initial and boundary conditions are identical to the initial and boundary conditions adopted for both the ESCOA correlation method and the analytical method. The value of the tube temperature $T_{tube}$, which is imposed on the tube surface, and the value of the inlet flue gas temperature $T_{in}$, which is imposed at the inlet of the computational domain are shown in Table 3-12. These temperatures, also defined in Table 2-1 in Section 2-3, have been converted from degree Celsius to degree Kelvin.
Table 3.12: Values of $T_{tube}$ and $T_{in}$ imposed on the tube surface and inlet of the computational domain respectively.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_{in}$</td>
<td>699.7 [K]</td>
</tr>
<tr>
<td>$T_{tube}$</td>
<td>421.9 [K]</td>
</tr>
</tbody>
</table>

All numerical models, except those related to the investigation of two different fin configurations use a constant value for the average mass flux, which is $G_{avg} = 2$ kg/m$^2$·s. This does not imply that the value of the inlet velocity $V_{in}$ should also be constant. The value of $G_{avg}$ is imposed at the cross-section of the finned tube where the total flow area $A_{flow,tot}$ is the smallest, which is calculated as the domain inlet area $A_{inlet}$ minus the projected area of the tube minus the projected area of the fin. The area $A_{flow,tot}$ is represented by the white area in Figure 3.15. $G_{avg}$ has to be rewritten to specify an inlet velocity $V_{in}$ at the inlet of the computational domain in ANSYS Fluent, which is done according to:

$$V_{in} = G_{avg} \rho \frac{A_{flow,tot}}{A_{inlet}}$$

(3-4)

The inlet area $A_{inlet}$ is calculated by $A_{inlet} = V_1 Z_1$. It is evident that the ratio between $A_{flow,tot}$ and $A_{inlet}$ will decrease for an increased fin density $N_f$, which ultimately results in a lower value of the inlet velocity $V_{in}$ while $G_{avg}$ is kept constant. The value of $\rho$ represents the density of the flue gas at the inlet of the computational domain which is at $T_{in} = 699.7$ K, and has a value of $\rho = 0.5041$ kg/m$^3$ (Mills, 1999).

Figure 3.15: Cross-section of the finned tube geometry. $A_{flow,tot}$ is depicted by the white area.
Mesh Interfaces

All numerical models which have been developed during this master thesis research consist of so-called non-conformal interfaces. This means that the location of the nodes of cells at one side of the interface do not match the location of the nodes of cells at the other side of the interface (ANSYS Inc., 2013b). The mesh interfaces tab in ANSYS Fluent allows users to choose from several options on how these interfaces are treated.

Related to this master thesis research, the non-conformal interfaces which have been identified by ANSYS Fluent are identical to the five contact zones identified earlier during the specification of the meshing method. Although ANSYS Fluent treats these interfaces by default, the treatment of the interfaces have been revised to ensure that the correct option has been chosen for the interfaces:

- **Contact zone 1**: Coupled Wall
- **Contact zone 2**: Coupled Wall
- **Contact zone 3**: Coupled Wall
- **Contact zone 4**: Matching
- **Contact zone 5**: Matching

It can be observed that two different options have been selected to treat these interface zones. The first option is called the coupled wall option, which has been used for solid-solid and liquid-solid interfaces. When the coupled wall option is selected, no mass is allowed to flow across the interface and only heat transfer across the interface is tolerated. The second option is called the matching option, which has been used for liquid-liquid interfaces. By selection of the Matching method, both mass flow and heat transfer across the interface is allowed (ANSYS Inc., 2013b).

Solution Methods

The solution methods tab in ANSYS Fluent consists of two main parts. In the first part a scheme related to pressure-velocity coupling is adopted. In the second part methods related to various aspects of spatial discretization are selected.

Pressure-velocity coupling can be achieved by density-based algorithms and pressure-based algorithms. From these options, the latter has been selected. Although the flue gas flow is considered to be compressible, it is compressible to such an extent that it should still be possible to use pressure-based algorithms, which are used to calculate flows which are incompressible or moderately compressible (although both density-based and pressure-based algorithms are now valid for a wide range of flows) (ANSYS Inc., 2013b). The motivation for the selection of pressure-based algorithms is supported by the calculation of the Mach number Ma. The Mach number is calculated by \( Ma = \frac{V}{c} \), where \( c \) represents the speed of sound in the medium. For this master thesis research, a value of \( c \) corresponding to air at \( T = 400 \) °C was used. From all numerical models investigated, a maximum Mach number of
$Ma \approx 0.03$ has been obtained. Related to annular-finned tube heat transfer, this maximum Mach number is in agreement with the maximum Mach number calculated by Bacellar et al. (2014), where because of this value of the Mach number compressibility effects were neglected, a pressure-based algorithm was used, and the density was calculated according to the ideal gas law.

A variety of pressure-based algorithms are available. From these options, the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm has been selected. This is because the SIMPLE algorithm is the default pressure-based algorithm in ANSYS Fluent for steady-state simulations. It is realized that other solution algorithms such as the Semi-Implicit Method for Pressure Linked Equations-Consistent (SIMPLEC) algorithm and the Coupled algorithm are also applicable to the numerical models which have been developed during this master thesis research, but it was believed to be fundamental that the SIMPLE algorithm operated properly before more advanced solution algorithms are taken into consideration.

The SIMPLE algorithm is a pressure-based segregated algorithm. Pressure-based means that a pressure correction equation is derived from the continuity (mass conservation) equation and from the momentum equations, which is then used to iteratively calculate the velocity field while satisfaction of the continuity equation is maintained. Segregated implies that all governing equations (e.g.: velocity components, pressure, temperature, etc.) are solved one by one while being decoupled from the other governing equations (ANSYS Inc., 2013a). The iterative process of solving the flow field using the SIMPLE algorithm is described by the following steps (Versteeg and Malalasekera, 2007):

- **Step 1:** Solve discretized momentum equations by means of an initial guess for the pressure field. Since the pressure is only a guess, the momentum equations will not satisfy the continuity equation.
- **Step 2:** Derive the pressure correction equation.
- **Step 3:** Correct the pressure and velocities. These updated values satisfy the continuity equation, but do probably not satisfy the momentum equation.
- **Step 4:** Solve all other discretized transport equations (flow rate, temperature, etc.)
- **Step 5:** Check for convergence. If convergence has been reached, exit. If this is not the case, go back to step 1 but use the updated values calculated during the previous steps for the initial guess.

It has been previously mentioned in Section 2-5 that ANSYS Fluent adopts the Finite Volume Method (FVM). This means that for each control volume, represented by individual grid cells in the computational domain, discretized transport equations for scalar quantities are obtained while the conservation law is still valid. The values of these scalar quantities are stored in the center of the grid cells. However, to be able to calculate the convection term in the discretized transport equation, values of scalar quantities are required at the boundaries of the grid cell. Therefore, upwind schemes are introduced to interpolate the values of scalar quantities at the cell center to the grid cell boundary (ANSYS Inc., 2013a).
Various spatial discretization schemes are available in ANSYS Fluent. The discretization schemes used for this master thesis research are identical for all numerical models which have been developed, and are shown in Table 3-13. The settings in Table 3-13 are the default settings provided by ANSYS Fluent, with the exception of the turbulent kinetic energy and the turbulent dissipation rate which have been changed from first-order upwind to second-order upwind. The motivation to prefer second-order discretization schemes over first-order discretization schemes is that in general second-order discretization schemes yield more accurate results compared to first-order discretization schemes, because second-order discretization schemes show less numerical diffusion. Numerical diffusion can be seen as dampening behavior relative to the exact solution calculated by the numerical solution. The existence of numerical diffusion applies especially for triangular and tetrahedral meshes, but also for quadrilateral and hexagonal meshes involving complex flows, where the flow is hardly ever aligned with the mesh and crosses the boundaries of grid cells sideways. As a result, the use of second-order discretization schemes is recommended (ANSYS Inc., 2013b).

### Convergence Criteria

The monitor tab in ANSYS Fluent provides the option to change the values of scaled residuals. Scaled residuals are defined as the inequalities in the conservation of mass, momentum, and energy in a control volume (ANSYS Inc., 2013b). Scaled residuals are useful to indicate if convergence (a solution for the iterative calculation process) has been reached.

**Table 3-14:** Absolute convergence criteria.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Absolute Criteria</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>$1 \cdot 10^{-4}$</td>
</tr>
<tr>
<td>x-velocity</td>
<td>$1 \cdot 10^{-4}$</td>
</tr>
<tr>
<td>y-velocity</td>
<td>$1 \cdot 10^{-4}$</td>
</tr>
<tr>
<td>z-velocity</td>
<td>$1 \cdot 10^{-4}$</td>
</tr>
<tr>
<td>Energy</td>
<td>$1 \cdot 10^{-8}$</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>$1 \cdot 10^{-4}$</td>
</tr>
<tr>
<td>Turbulent Dissipation Rate</td>
<td>$1 \cdot 10^{-4}$</td>
</tr>
</tbody>
</table>
In general, lower values of scaled residuals yield a more accurate solution. However, this does not imply that the final solution is correct since a mistake might have been made elsewhere during the setup of the numerical model, which would compromise the final solution. By default, the values of the scaled residuals set by ANSYS Fluent are $1 \cdot 10^{-6}$ for energy and $1 \cdot 10^{-3}$ and for the other variables. For most problems, these values of the scaled residuals are acceptable ANSYS Inc. (2013b). Preliminary tests have been conducted to determine if this recommendation is also true for problems related to heat transfer and flow around solid annular-finned tubes. Default values of the scaled residuals were compared against reduced values of the scaled residuals, and it was found that the wake region behind the finned tube was unrealistically short for the default values of the scaled residuals, but not for the reduced values of the scaled residuals. As a result, the reduced values of the scaled residuals have been adopted for the setup of the numerical models shown in Table 3-14. In addition to the observation of the scaled residuals, the value of the outlet temperature $T_{out}$ has also been monitored to determine if the solution converges.

Calculations

Except for the numerical models which exceed the maximum number of cells imposed by the ANSYS Fluent student license (surpassing 500,000 cells), all numerical models have been simulated on a laptop with an Intel® Core™ i7-3610QM CPU, 8GB of RAM and a NVIDIA Quadro K1000M graphics card. The remainder of numerical models which required the full ANSYS Fluent license have been simulated on a PC with an Intel® Xeon™ E5-2665 CPU, 32GB of RAM and a NVIDIA GT218 graphics card.

The number of iterations used varied between 1500 and 4000 iterations, depending on the type of investigation and grid size. The calculation time is also dependent on the type of investigation and grid size, which took between half an hour and five hours to complete.
Chapter 4

Results and Discussion

4-1 Introduction

In the previous chapter, a methodology has been introduced for the development of the numerical models necessary to perform all three investigations. Furthermore, the ESCOA correlation method and the derivation of the analytical method have been explained elaborately in preceding chapters. All this information is considered to be essential to be able to comprehend the subject matter in this chapter properly.

In this chapter, the results will be presented and compared with the ESCOA correlation method and the analytical method. After all the results have been presented and all the subsequent comparisons have been made, the aspiration is to prove that the analytical method is indeed a valid method to predict heat transfer and pressure drop in the convection section of fired heaters.

The outline of this chapter is divided into three parts, according to the three investigations which have been conducted. First, the results related to the investigation of one tube row will be presented, which includes the values of the heat transfer and pressure drop coefficient relative to the bare tube area versus a varying fin density for 4” tubes, 6” tubes, and 8” tubes. Second, the values of the heat transfer coefficient relative to the bare tube area for an increasing number of tube rows will be presented and reviewed for 4” tubes and 6” tubes. To conclude, the third part will present and discuss the results for one tube row again, where the effect on the values of the heat transfer coefficient relative to the bare tube area for two different fin configurations has been studied for 6” tubes and 8” tubes.
4-2 Results for the Investigation of One Tube Row

4-2-1 Heat Transfer Coefficient Comparison for 4”, 6”, and 8” Tubes

Heat Transfer Coefficient Comparison for 4” Tubes

The comparison of the heat transfer coefficient relative to the bare tube area \( U_{bare} \) versus the fin density \( N_f \) calculated by various methods for 4” tubes is shown in Figure 4-1. The green line represents the ESCOA correlation method. The red line and the yellow line represent the initial analytical method and the improved analytical method respectively. The main difference between the initial analytical method and the improved analytical method is that the latter excludes the fin area of the wake region behind the tube, which does not participate in heat transfer. This excluded area is depicted by the shaded area in Figure 4-2. To conclude, the orange, blue, and purple lines represent the results obtained from the numerical models on a coarse grid, medium grid, and refined grid respectively.

It has previously been mentioned during the derivation of the analytical method in Section 2-4 that the most conspicuous observation is the presence of a distinct fin optimum density calculated by the analytical method. This is in contrast with the ESCOA correlation method which calculates increased values of \( U_{bare} \) for an increased fin density. This fin density optimum calculated by the analytical method is plausible, since it can be realized that an increased fin density will result in an increased fin resistance. This makes it more difficult for the upstream flow to reach the fluid domain between the fins to exchange heat. The (improved) analytical method takes this increased flow resistance corresponding to an increased fin density into account by calculation of two mass fractions: The flue gas mass flow which flows between the fins \( \dot{m}_{\text{fin}} \) and the flue gas mass flow which bypasses the fins \( \dot{m}_{\text{free}} \), from

![Figure 4-1: Comparison of the heat transfer coefficient relative to the bare tube area \( U_{bare} \) calculated by various methods versus the fin density \( N_f \) for 4” tubes. Number of tube rows \( N_R = 1 \), average mass flux \( G_{\text{avg}} = 2 \) kg/m²·s.](image)
which only $\dot{m}_{fin}$ participates in the subsequent heat transfer calculations. Increased values of the fin density will increase the fin resistance and therefore decrease $\dot{m}_{fin}$. It can be realized that this will result in a decreased value of $U_{bare}$ according to Equation 2-39. This is in contrast with the ESCOA correlation method, which only uses the average mass flux $G_{avg}$ to correlate it to finned tube heat transfer. To a certain extent, coefficient $C_3$ from Equation 2-6 listed in Section 2-3 takes the effect of an increased flow resistance into consideration, by calculating a decreased value of $C_3$ for an increased fin density. However, the decreased value of coefficient $C_3$ does not provide sufficient compensation to account for the increased fin area $A_{xfer,fin}$, which ultimately results in higher values of $U_{bare}$ according to Equation 2-8.

Differences between the ESCOA correlation method and the (improved) analytical method are also present for low fin density values. It is shown in Figure 4-1 that the values of $U_{bare}$ calculated by the (improved) analytical method drop significantly compared to the values of $U_{bare}$ calculated by the ESCOA correlation method. This can be explained by observation of Equation 2-34 in Section 2-4. The fin heat transfer coefficient $U_{fin}$ decreases inversely proportional with the hydraulic diameter $D_{hydr,fin}$, which is represented by two times the fin spacing: $D_{hydr,fin} = 2S_{fin}$ according to Equation 2-22. Lower values of $U_{fin}$, calculated due to a lower fin density result in decreased values of the Number of Transfer Units (NTU). In subsequent heat transfer calculations, this means that lower values of the heat exchanger effectiveness $\epsilon$ and ultimately $U_{bare}$ are obtained. The explanation of $U_{bare}$ calculated by the ESCOA correlation method for low fin densities is similar to the case for high fin densities, only this time the other way around. Higher values of coefficient $C_3$ are calculated for a decreased fin density, but the decrease of $A_{xfer,fin}$ ultimately results in lower values of $U_{bare}$ calculated by the ESCOA correlation method. However, it is shown in Figure 4-1 that this decrease is not as significant compared to the (improved) analytical method.
It is also shown in Figure 4-1 that the agreement between the improved analytical method and the ESCOA correlation method is considerably better compared to the agreement between the initial analytical method and the ESCOA correlation method for common fin density and average mass flux values (i.e. $N_f = 4$ inch$^{-1}$ and $G_{avg} = 2$ kg/m$^2\cdot$s). The exclusion of the wake region behind the tube decreases the heat transfer surface of the fin $A_{xfer, fin}$ and therefore the effective heat transfer surface $A_{xfer, eff}$, which can be seen schematically in Figure 4-2. Corresponding to the improved analytical method, this reduces the magnitude of the $U_{bare}$ values considerably. In addition, the flow path length between the fins $L_{flow, fin}$ is reduced by the same amount due to the exclusion of the wake region behind the tube. This decreases the fin resistance and therefore increases the mass flow between the fins $\dot{m}_{fin}$. As a result, the improved analytical method transfers more heat compared to the initial analytical method for higher fin densities, which explains the shift of the fin density optimum towards higher values.

Related to the comparison between the ESCOA correlation method and the analytical method, it is particularly interesting to observe the values of $U_{bare}$ which have been obtained by the numerical models. By measuring the heat flow through the inlet and outlet of the computational domain, the heat which has been transferred is obtained. However, this is for a quarter of the fin and only for the thickness of the computational domain $Z_1$. For a correct comparison with the ESCOA correlation method and the (improved) analytical method, the measured heat transfer is multiplied by four and by the fin density to obtain the heat transfer per meter of finned tube row. Similar to the analytical method, $U_{bare}$ is calculated according to:

$$U_{bare} = \frac{Q_{fin}}{A_{bare}(T_{gm} - T_{tube})}$$

(4-1)

Where the mean gas temperature $T_{gm}$ is the average between the inlet and outlet temperature corresponding to the computational domain. Figure 4-1 depicts two features which are the most conspicuous. First, although the number of grid cells used by the refined mesh is approximately tenfold the number of cells used by the coarse grid, all three grid sizes approximately calculate the same values of $U_{bare}$. The maximum deviation between all three grid sizes which has been found is 1.47% between the coarse grid and refined grid for $N_f = 2$ inch$^{-1}$. For all fin density values, a maximum deviation of approximately 1% has been observed between the medium grid and the refined grid. Second, the plots of $U_{bare}$ which have been calculated by the numerical models also display a distinct fin optimum density, similar to the fin optimum density calculated by the (improved) analytical method. It has been mentioned before that the (improved) analytical method calculates a decreasing mass flow fraction between the fins $\dot{m}_{fin}$ and therefore also a decreased flow velocity $u$ (velocity component in x-direction) between the fins for an increased fin density. By inspection of the velocity $u$ isosurfaces in Figure A-1 in Appendix A it can be concluded that the same is true for the numerical models, where a significant decrease in velocity $u$ is observed for an increasing fin density. This increase in fin resistance can be attributed to the increased interaction between boundary layers for higher fin densities. If the space between the fins is sufficiently small, boundary layers cannot develop independently from each other anymore and instead touch each other. This makes it more difficult for the upstream flue gas to reach the flow area between the fins. In addition, a XY-plot has been made at half the fin height $h$ to obtain a velocity profile for $u$, which can be seen in Figure A-4. Using the midpoint rule, the area below the curves of the XY-plot can be approximated to determine the average velocity $u$ between the fins. The result is shown...
Table 4-1: Average velocity \( u \) between the fins at half the fin height \( h \) calculated by the initial analytical method, improved analytical method, and numerical models for 4\(^\text{in} \) tubes.

<table>
<thead>
<tr>
<th>Fin Density ( N_f )</th>
<th>Initial Analytical Method</th>
<th>Improved Analytical Method</th>
<th>Numerical Models (Medium Mesh)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 [inch(^{-1})]</td>
<td>3.95 [m/s]</td>
<td>4.25 [m/s]</td>
<td>3.62 [m/s]</td>
</tr>
<tr>
<td>4 [inch(^{-1})]</td>
<td>2.29 [m/s]</td>
<td>2.70 [m/s]</td>
<td>2.79 [m/s]</td>
</tr>
<tr>
<td>8 [inch(^{-1})]</td>
<td>0.51 [m/s]</td>
<td>0.70 [m/s]</td>
<td>1.10 [m/s]</td>
</tr>
</tbody>
</table>

in Table 4-1, together with the values of the average velocity \( u \) calculated by the (improved) analytical method. It can be observed that the average velocity \( u \) calculated by the numerical models are remarkably similar to the average velocity \( u \) calculated by the improved analytical method, especially by taking into consideration that the (improved) analytical method is still a proof of concept calculation and a lot of improvements have yet to be made. Higher values of the average velocity \( u \) are calculated by the numerical models for \( N_f = 4 \) inch\(^{-1} \) and \( N_f = 8 \) inch\(^{-1} \) compared to the (improved) analytical method. This might imply that the fin resistance is not as large for high fin density values compared to the predictions which have been made by the (improved) analytical method. However, for \( N_f = 2 \) inch\(^{-1} \) the numerical models calculate a lower value of the average velocity \( u \) compared to the (improved) analytical method. Additional investigation related to the derivation related to the analytical method learns that for \( N_f = 2 \) inch\(^{-1} \), the (improved) analytical method calculates a higher average flow velocity \( u \) between the fins compared to the average flow velocity \( u \) which bypasses the fins. This is a questionable result, because it is expected that the average flow velocity \( u \) which bypasses the fins should be higher compared to the average flow velocity \( u \) between the fins, even for low fin density values. This is supported by the velocity \( u \) isosurfaces in Figure A-1, which do not show a higher flow velocity \( u \) between the fins compared to the flow velocity which bypasses the fins for \( N_f = 2 \) inch\(^{-1} \). A reason for this result is that the current (improved) analytical method does not take the dynamic pressure (increase in pressure due to the motion of the fluid) into account. Although this is an acceptable assumption for common fin density and average mass flux values values, this results in a velocity \( u \) which is too high for low fin densities.

The decreased velocity \( u \) between the fins for an increased fin density has as an additional consequence. For the hot flue gas which actually manages to flow between the fins, it takes more time to exit the fluid domain between the fins again. As a result, it can be observed from the temperature isosurfaces in Figure A-7 that the temperature of the fin and the temperature of the flow between the fins approach the temperature of the tube \( T_{\text{tube}} \) for an increasing fin density. The flow between the fins starts to behave similar to insulation, which ultimately decreases the value of \( U_{\text{bare}} \) and therefore explains the fin density optimum. This is also supported by the temperature contours of the finned tube in Figure A-10, Figure A-11, and Figure A-12, where it can be seen that because of the decreased flow velocity \( u \) between the fins the temperature of the fin approaches the tube temperature \( T_{\text{tube}} \) for increased fin density values and therefore exchanges less heat.

It is an interesting result that the results obtained by the numerical models also show a distinct fin density optimum. However, significant differences can be observed between the numerical models and the (improved) analytical method. For high fin densities, it is shown in Figure 4-1 that the numerical models calculate higher values of \( U_{\text{bare}} \) compared to the values.
calculated by the (improved) analytical method. The first explanation for this could be that
the latter calculates higher values of the average flow velocity \( u \) between the fins shown in
Table 4-1, because of the decreased fin resistance or different driving force. This difference
in velocity \( u \) is especially noticeable for high fin density values and should therefore result in
increased heat transfer compared to the (improved) analytical method. Second, the analytical
method does not consider heat transfer on the fin tip surface, in contrast to the numerical
models. At the fin tip surface, Figure A-19, Figure A-20, and Figure A-21, which represent
the surface heat transfer coefficient of the finned tube surface, reveal that the surface heat
transfer coefficient can be up to an order of magnitude higher compared to any other location
on the finned tube surface, despite its small area. Especially for high fin density values, the
effect of this increased surface heat transfer coefficient at the fin tip surface becomes more
significant, which would ultimately result in the calculation of higher \( U_{bare} \) values compared
to the (improved) analytical method. It can also be observed from these figures that the fin
area of the wake region behind the tube barely participates in heat transfer, and that this area
appears to look similar to the shaded region in Figure 4-2. This implies that the exclusion
of this area introduced by the improved analytical method has been a critical improvement
over the initial analytical method.

Further examination between the improved analytical method and the numerical models
reveals that the latter also calculate higher \( U_{bare} \) values for low fin densities. The reason
for this could be twofold. First, the value of the Nusselt number used by the (improved)
analytical method \( (Nu = 7.54) \) is for a fully developed laminar flow between two infinite
plates. Especially for a low fin density where the fin spacing \( S_{fin} \) is large, it is shown that the
flow has not been fully developed yet. This is supported by the XY-plot in Figure A-4 where
it is shown that for \( N_f = 2 \) inch\(^{-1} \) the velocity profile represents an undeveloped laminar
flow, whereas for \( N_f = 4 \) inch\(^{-1} \) and \( N_f = 8 \) inch\(^{-1} \) the velocity profiles represent a fully
developed laminar flow. For undeveloped flows, an increased value of the Nusselt number
should be used (Mills, 1999). As a consequence, higher values for \( U_{bare} \) should be calculated
which are more in line with the results obtained by the numerical models. Second, the flow
between the fins for low fin densities could have turbulent flow characteristics. This would
increase heat transfer and would become apparent especially for low fin densities. This is in
contrast with the (improved) analytical method, which assumes the flow to be fully laminar
for all fin density values.

The differences in \( U_{bare} \) discussed previously arise for low and high fin density values. In
practice however, fin density values are in the range of \( 3 \leq N_f \leq 5 \) inch\(^{-1} \). Fin density values
less than \( N_f = 3 \) inch\(^{-1} \) are seldom encountered, and the maximum allowed fin density is
restricted to \( N_f = 5 \) inch\(^{-1} \) according to API Standard 560 (American Petroleum Institute,
2001). It is therefore an important observation in Figure 4-1 that the results obtained by
the ESCOA correlation method, the improved analytical method and the numerical models
are in close agreement with each other for these fin densities. This is valuable information,
since for these common values the ESCOA correlation method has proven to be a valid
method to predict heat transfer in the convection section of fired heaters for 4’ tubes. The
close agreement for \( U_{bare} \) between the ESCOA correlation method, the improved analytical
method and the numerical models implies that valid numerical models have been developed,
and that the improved analytical method calculates valid results for common fin density and
average mass flux values.
Heat Transfer Coefficient Comparison for 6" Tubes

The comparison of $U_{bare}$ versus the fin density calculated by various methods for 6" tubes is shown in Figure 4-3, where it can be observed that similar results have been obtained compared to the $U_{bare}$ comparison for 4" tubes in Figure 4-1. In other words, the (improved) analytical method and the numerical models calculate a distinct fin optimum density in contrast to the ESCOA correlation method, which calculates increased $U_{bare}$ values for an increased fin density. Furthermore, the results obtained by the numerical models are again situated between the results calculated by the ESCOA correlation method and the (improved) analytical method. The values of the average velocity $u$ calculated from the XY-plots in Figure A-5 are again much alike the values calculated by the improved analytical method. The average velocity $u$ calculated by the (improved) analytical method for $N_f = 2$ inch$^{-1}$ is also a questionable result for the same reason explained during the 4" tube comparison. To clarify the differences between the results calculated by the ESCOA correlation method, the (improved) analytical method and the numerical models in Figure 4-3, the same arguments used for the 4" tube comparison are adopted. Supporting figures obtained by ANSYS fluent for 6" tubes can be found in Appendix A.

Aside from the similarities mentioned above, there are also significant differences present between the $U_{bare}$ comparison for 4" tubes and 6" tubes. It can be observed that the plot of the ESCOA correlation method in Figure 4-3 does not agree as well with the other methods compared to the 4" tube comparison in Figure 4-1 for common fin density and average mass flux values (i.e. $N_f = 4$ inch$^{-1}$ and $G_{avg} = 2$ kg/m$^2$·s). However, this increased difference between the ESCOA correlation method and the other methods is not considered to be an unfavorable result. It has been mentioned before that the ESCOA correlation method was derived from wind tunnel experiments, which have been conducted using only small tubes.

![Figure 4-3: Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ calculated by various methods versus the fin density $N_f$ for 6" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s.](image_url)
Table 4-2: Average velocity $u$ between the fins at half the fin height $h$ calculated by the initial analytical method, improved analytical method, and numerical models for 6" tubes.

<table>
<thead>
<tr>
<th>Fin Density $N_r$</th>
<th>Initial Analytical Method</th>
<th>Improved Analytical Method</th>
<th>Numerical Models (Medium Mesh)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 inch$^{-1}$</td>
<td>3.89 [m/s]</td>
<td>4.58 [m/s]</td>
<td>3.49 [m/s]</td>
</tr>
<tr>
<td>4 inch$^{-1}$</td>
<td>1.48 [m/s]</td>
<td>2.00 [m/s]</td>
<td>2.30 [m/s]</td>
</tr>
<tr>
<td>8 inch$^{-1}$</td>
<td>0.23 [m/s]</td>
<td>0.35 [m/s]</td>
<td>0.50 [m/s]</td>
</tr>
</tbody>
</table>

ranging from less than 1" to 2" in diameter (occasionally measurements using 4" tubes have been performed). The results were extrapolated to produce correlations which are applicable for tubes sizes ranging up to 4" in diameter. Although the ESCOA correlation method has proven to be a valid method for these tube sizes, it is debatable to extend the correlations to make them also applicable for larger tube sizes such as 6’ and 8’ tubes, for which they were initially not designed. It can be realized that further extrapolation for larger tube sizes results in correlations which get more diverged from reality. Therefore, it should be questioned if the results produced by these correlations are still accurate, and it is not expected that the ESCOA correlation method and the numerical models should be in agreement with each other for common fin density and average mass flux values in order to be able to validate the (improved) analytical method since the results obtained by the numerical models are considered to be the most accurate of the three methods.

Compared 4’ tubes, increased differences in $U_{bare}$ calculated by the improved analytical method and the numerical models can also be observed in Figure 4-3 for common fin density and mass flux values. The reason for this is twofold. First, because of the larger tube size, $L_{flow,fin}$ increases and the flue gas flow between the fins decreases. As a result, fin tip heat transfer plays a more significant role for 6’ tubes compared to 4’ tubes. The (improved) analytical method has not implemented heat transfer at the fin tip surface yet. Second, it can be seen in Table 4-2 that the difference in velocity $u$ of the flue gas flow between the fins for $N_f = 4$ inch$^{-1}$ calculated by the improved analytical method and numerical models has increased for 6” tubes compared to 4” tubes. It has been mentioned during the 4” tube $U_{bare}$ comparison that the fin resistance calculated by the numerical models is not as large compared to the fin resistance calculated by the (improved) analytical method, and this effect is more noticeable for 6” tubes compared to 4” tubes. Both the increased fin tip area and the reduced velocity $u$ result in a decrease in $U_{bare}$ for the improved analytical method.

It is also shown in Figure 4-3 that the overall values of $U_{bare}$ are generally lower for the 6” tube comparison compared to the $U_{bare}$ values related to the 4” tube comparison. The first reason for this is that 6’ tubes have an increased transverse tube pitch $t_q$ compared to 4’ tubes. Since the fin height is restricted to $h \leq 1’$ according to API Standard 560, the flue gas is forced to a much lesser extent to flow between the fins to exchange heat. Together with the increased fin resistance due to the longer flow path length $L_{flow,fin}$ corresponding to 6’ tubes, this also causes the fin density optimum to shift towards a lower value compared to the fin density optimum for 4’ tubes. The second reason is that the bare tube surface area $A_{bare}$ has also increased for 6’ tubes compared to 4’ tubes, which ultimately results in lower $U_{bare}$ values according to Equation 2-41.
Related to the values of \( U_{\text{bare}} \) calculated by the numerical models, the maximum deviation between all three grid sizes which has been found is 4.31\% between the coarse grid and refined grid for a fin density value of \( N_f = 7 \text{ inch}^{-1} \). This might seem like a significant increase from the 1.47\% calculated for \( N_f = 2 \text{ inch}^{-1} \) corresponding to the 4\" tube comparison. However, further investigation shows that the deviation between the medium grid and refined grid for \( N_f = 7 \text{ inch}^{-1} \) is 0.10\%, which puts the relatively large deviation of 4.32\% between the coarse grid and refined grid into perspective. A reason for this deviation could be that large values of the fin density cause the computational domain to be relatively thin since \( Z_1 \) decreases according to Table 3-2, and the coarse mesh does not place a sufficient amount of cells in the fin fluid domain to properly model the flow in this region. Similar to the 4\" tube comparison, a maximum \( U_{\text{bare}} \) deviation of approximately 1\% has been observed between the medium grid and the refined grid for all fin density values. Furthermore, in Figure A-2 and Figure A-8 it is shown that for 6\" tubes and \( N_f = 2 \text{ inch}^{-1} \), the velocity and temperature isosurfaces have a noticeably different shape in the vicinity of the fin and tube wall compared to the velocity and temperature isosurfaces with the same fin density value for 4\" tubes in Figure A-1 and Figure A-7, where a flat shape at the vicinity of the fin and tube wall can be observed. It is believed that the decreased average velocity \( u \) of the flow between the fins and the longer flow path length \( L_{\text{flow,fin}} \) corresponding to 6\" tubes could give rise to secondary effects. These secondary effects could include the presence of horseshoe vortices or secondary vortex systems which have been observed in literature related to finned tube heat transfer by the conduction of wind tunnel experiments and numerical investigations (Sung et al., 1996) (Mon and Gross, 2004). These effects are not investigated more in-depth over the course of this master thesis research, but could be an interesting subject for further research.

It can be observed in Figure 4-3 that for the fin density range \( 3 \leq N_f \leq 5 \text{ inch}^{-1} \), deviations between \( U_{\text{bare}} \) calculated by the ESCOA correlation method and the numerical models are the largest for \( N_f = 5 \text{ inch}^{-1} \), 25.5\% to be exact. However, large margins are taken into account during the design of fired heaters. According to FLUOR engineers responsible for the design of fired heaters, margins up to 25\% are added to what the fired heater is supposed to achieve on paper. This makes the ESCOA correlation method for 6\" tubes still applicable for the fin density range specified above, which is supported in practice by various projects related to fired heaters which have been designed by FLUOR in the past.

**Heat Transfer Coefficient Comparison for 8\" Tubes**

The comparison of \( U_{\text{bare}} \) versus the fin density calculated by various methods for 8\" tubes is shown in Figure 4-4. Similar to the 4\" tube and 6\" tube \( U_{\text{bare}} \) comparison, the (improved) analytical method and the numerical models calculate an optimum \( U_{\text{bare}} \) value, whereas the ESCOA correlation method calculates increased values of \( U_{\text{bare}} \) for an increased fin density. Also for the 8\" tube comparison the results obtained by the numerical models are situated between the results calculated by the ESCOA correlation method and the (improved) analytical method, and the values of the average velocity \( u \) in Table 4-3 calculated from the XY-plots in Figure A-6 are in close agreement with the values calculated by the improved analytical method. Similar to the 4\" tube and 6\" tube comparison, the average velocity \( u \) calculated by the (improved) analytical method for \( N_f = 2 \text{ inch}^{-1} \) for 8\" tubes is higher corresponding to the flow between the fins compared to the flow which bypasses the fins, which was considered to be a questionable result since regardless of the fin density, the average velocity \( u \) of the flue
Results and Discussion

Figure 4-4: Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ calculated by various methods versus the fin density $N_f$ for 8" tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2 \text{ kg/m}^2\cdot\text{s}$.

gas flow which bypasses the fins should be higher compared to the average velocity $u$ of the flue gas flow which bypasses the fins. It has been mentioned previously that this is attributed to the fact that the (improved) analytical method has not implemented the dynamic pressure yet for the flow between the fins.

Apart from these similarities, it can be observed in Figure 4-4 that the differences between $U_{bare}$ calculated by the ESCOA correlation method, the (improved) analytical method and the numerical models introduced by the 6" tube comparison are increased further. During a comparison between $U_{bare}$ calculated by the ESCOA correlation method and by the numerical models for the fin density range $3 \leq N_f \leq 5 \text{ inch}^{-1}$, deviations of 48.6% for $N_f = 5 \text{ inch}^{-1}$ and 25.3% for $N_f = 4 \text{ inch}^{-1}$ were calculated, which cannot be compensated anymore by the large margins applied to the design of fired heaters. The use of the ESCOA correlations for 6" tubes was already debatable, but it can be concluded that the ESCOA correlation method should be avoided for 8" tubes.

Following the trend which was already noticeable by progressing from 4" tubes to 6" tubes, the overall values of $U_{bare}$ calculated for 8" tubes are lower compared to the values of $U_{bare}$ calculated corresponding to 6" tubes. Again, the reason for this is twofold. First, the transverse tube pitch $t_q$ is increased even further for 8" tubes compared to 6" tubes. Since the fin height is still restricted to $h \leq 1'$, the flue gas is forced to a much less extent to flow between the fins to exchange heat. Together with the increased fin resistance due to the longer flow path length $L_{flow,fin}$ corresponding to 8" tubes, this causes the fin density optimum to shift again towards an even lower value compared to the fin density optimum observed for 6" tubes. Second, the increased bare tube surface area $A_{bare}$ for 8" tubes compared to $A_{bare}$ for 6" tubes ultimately results in lower values of $U_{bare}$ according to Equation 2-41.
4-2 Results for the Investigation of One Tube Row

Table 4-3: Average velocity \( u \) between the fins at half the fin height \( h \) calculated by the initial analytical method, improved analytical method, and numerical models for 8" tubes.

<table>
<thead>
<tr>
<th>Fin Density ( N_f )</th>
<th>Initial Analytical Method</th>
<th>Improved Analytical Method</th>
<th>Numerical Models (Medium Mesh)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 [inch(^{-1})]</td>
<td>3.56 [m/s]</td>
<td>4.51 [m/s]</td>
<td>3.37 [m/s]</td>
</tr>
<tr>
<td>4 [inch(^{-1})]</td>
<td>1.07 [m/s]</td>
<td>1.54 [m/s]</td>
<td>1.93 [m/s]</td>
</tr>
<tr>
<td>8 [inch(^{-1})]</td>
<td>0.15 [m/s]</td>
<td>0.24 [m/s]</td>
<td>0.16 [m/s]</td>
</tr>
</tbody>
</table>

Related to the values of \( U_{bare} \) calculated by the numerical models, the maximum deviation between all three grid sizes which has been found is 4.09% between the coarse grid and medium grid for \( N_f = 7 \) inch\(^{-1}\), much alike the 4.31% which has been found for the 6" tube comparison. For this fin density value, the difference between the medium grid and the refined grid is 0.36%, which is similar to the 0.10% calculated between the medium grid and refined grid for the 6" tube comparison. The difference between 4.09% and 0.36% could again be attributed to the coarse mesh which does not place a sufficient amount of cells in the fin fluid domain to properly model the flow in this region. Similar to the 4" tube and 6" tube comparison, a maximum deviation of approximately 1% has been observed between the medium grid and the refined grid for all fin density values. Furthermore, it can be observed in the velocity and temperature isosurfaces in Figure A-3 and Figure A-9 that the deviating profiles introduced by the 6" tube comparison are more pronounced for 8" tubes. Similar to the increased flow path length between the fins \( L_{flow,fin} \) for 6" tubes compared to 4" tubes, \( L_{flow,fin} \) is also increased for 8" tubes compared to 6" tubes. It is believed that this increase makes 8" tubes even more sensitive to the secondary effects discussed previously during the \( U_{bare} \) comparison for 6" tubes.

4-2-2 Pressure Drop Coefficient Comparison for 4", 6", and 8" Tubes

**Pressure Drop Coefficient Comparison for 4" Tubes**

During the process of calculating \( U_{bare} \) by means of the analytical method, the pressure drop is obtained according to Bernoulli’s principle, since the velocity at the stagnation point is zero and the average velocity \( u \) of the flue gas flow which bypasses the fins has already been calculated. It has previously been mentioned in Section 2-3 that the ESCOA correlation method also provides correlations to determine the pressure drop by calculation of the pressure drop coefficient relative to the bare tube area \( \zeta \). Adding the values of the pressure drop obtained from the numerical models, it would be interesting to compare \( \zeta \) calculated by these methods. This comparison of \( \zeta \) versus the fin density \( N_f \) for 4" tubes is shown in Figure 4-5. Similar to the \( U_{bare} \) comparison, the green line represents the ESCOA correlation method, the red line and the yellow line represent the initial analytical method and the improved analytical method, and the results calculated by the numerical models on a coarse grid, medium grid, and refined grid are depicted by the orange, blue, and purple lines respectively.
Figure 4-5: Comparison of the pressure drop coefficient relative to the bare tube area $\zeta$ calculated by various methods versus the fin density $N_f$ for 4" tubes. Number of tube rows $N_r = 1$, average mass flux $G_{avg} = 2 \text{ kg/m}^2 \cdot \text{s}$.

It is shown in Figure 4-5 that all methods calculate an increased value of $\zeta$ for an increased fin density. It has been mentioned during the derivation of the analytical method that the pressure drop coefficient $\zeta_{fin}$ (related to the the fin flow area $A_{flow,fin}$) and the pressure drop coefficient $\zeta$ (related to the total flow area $A_{flow,tot}$) are connected by:

$$\Delta p_{fin} = \Delta p_{avg} \rightarrow \zeta_{fin} \frac{G_{fin}^2}{2\rho_{fin}} = \zeta \frac{G_{avg}^2}{2\rho_{avg}} \quad (4-2)$$

According to Equation 2-25, this can be rewritten to:

$$48\pi \mu \frac{d_A + h}{D_{hydr,fin}^2} \frac{G_{fin}}{2\rho_{fin}} = \zeta \frac{G_{avg}^2}{2\rho_{avg}} \quad (4-3)$$

It was found during the $U_{bare}$ comparison that the velocity $u$ and therefore the mass flux between the fins $G_{fin}$ decreases for increasing fin densities. According to Equation 4-3, this results in lower values of the pressure drop coefficient relative to the bare tube $\zeta$. However, the increased fin density values also cause the hydraulic diameter between the fins $D_{hydr,fin}$ to decrease. Since $D_{hydr,fin}$ is a squared variable according to Equation 4-3, this ultimately results in higher overall values of $\zeta$ for increased fin densities calculated by the (improved) analytical method. Furthermore, it can be observed that for fin densities greater than $N_f = 8 \text{ inch}^{-1}$, the (improved) analytical method seems to attain an asymptotic value for $\zeta$. For these fin densities, the fin spacing is reduced to such an extent that the flow between the fins is minimal and the pressure drop coefficient approaches the value corresponding to a bare tube with diameter $D = d_A + 2h$.

For a fair comparison, it is important that the pressure drop is calculated over the same length. Related to the numerical models, it is tempting to measure the pressure difference between the
inlet and outlet of the computational domain. Atmospheric pressure is imposed at the outlet, and subsequently a pressure is calculated iteratively at the inlet of the computational domain. This pressure difference is, together with the average density of the flue gas, substituted in Equation 4-2 to obtain $\zeta$. However, the length of the computational domain is not the length which has been used to calculate $\zeta$ for the (improved) analytical method. It can be realized that the long downstream fluid domain would allow the pressure to recover from low pressures in the wake region of the finned tube to atmospheric pressure at the outlet of the computational domain, which would ultimately result in lower values of the pressure drop calculated by the numerical models compared to the (improved) analytical method. For a fair comparison, the pressure is calculated at two locations, depicted by the red dots in Figure 4-6. In this figure, it is shown that each of the red dots are placed at half the fin height $h$ and in the middle between two neighboring fins. The density is determined in each of these points as well, which is then averaged by taking the arithmetic mean to obtain the density of the flue gas flow between the fins $\rho_{\text{fin}}$. The density of the flue gas flow which bypasses the fins $\rho_{\text{free}}$ is obtained from the flow outside the fin fluid domain and corresponds to $T = T_{\text{in}}$. Similar to the (improved) analytical method, both $\rho_{\text{fin}}$ and $\rho_{\text{free}}$ are then used to calculate the average density $\rho_{\text{avg}}$ according to Equation 4-4:

$$\rho_{\text{avg}} = \frac{\dot{m}_{\text{free}} + \dot{m}_{\text{fin}}}{\frac{\dot{m}_{\text{free}}}{\rho_{\text{free}}} + \frac{\dot{m}_{\text{fin}}}{\rho_{\text{fin}}}}$$ (4-4)
Subsequently, the average density $\rho_{\text{avg}}$ is substituted in Equation 4-2 together with the value of the pressure drop over the fin $\Delta p_{\text{fin}}$ to obtain $\zeta$.

Similar to the explanation used to describe the $\zeta$ plot for the (improved) analytical method, the increase in $\zeta$ calculated by the numerical models can be explained in the same fashion. The flow area between the fins $A_{\text{flow,fin}}$ decreases, and as a result the majority of the flue gas has to flow through the area between the tubes $A_{\text{flow,free}}$. Not only does this reduce heat transfer, but it also causes the pressure drop and therefore $\zeta$ shown in Figure 4-5 to increase. For common fin density and mass flux values (i.e. $N_f = 4$ inch$^{-1}$ and $G_{\text{avg}} = 2$ kg/m$^2$s), it can be observed that $\zeta$ calculated by the improved analytical method and by the numerical models are similar. However, it can also be observed in Figure 4-5 that $\zeta$ calculated by the numerical models does not attain an asymptotic value for high fin densities, which is in contrast with the (improved) analytical method. The reason for this is that the latter assumes $\rho_{\text{free}}$ and $\rho_{\text{fin}}$ to be constant. Both $\rho_{\text{free}}$ and $\rho_{\text{fin}}$ have been calculated according to the ideal gas law in Equation 2-29 and are based on the inlet flue gas temperature $T_{\text{in}}$ and the mean fin temperature $T_{\text{rm}}$ respectively. The values of both temperatures are shown in Table 2-1 in Section 2-3. Although this is an acceptable estimate for the flue gas density for common fin density values, this is not the case for high fin densities such as $N_f = 8$ inch$^{-1}$, where the temperature of the flue gas flow between the fins is considerably lower compared to $N_f = 2$ inch$^{-1}$ and $N_f = 4$ inch$^{-1}$, which is supported by the temperature isosurfaces for 4" tubes in Figure A-7. This results in higher values of $\rho_{\text{avg}}$ compared to $\rho_{\text{avg}}$ calculated by the (improved) analytical method, and this explains the ongoing increase of $\zeta$ for increased fin densities calculated by the numerical models. Differences in $\zeta$ between the numerical models and the (improved) analytical method for low fin density values can be attributed to the fact that the latter does not take the dynamic pressure into account, which is especially significant for low fin density values where the velocity $u$ between the fins is relatively high.

Similar to the $U_{\text{bare}}$ comparison for all three tube sizes, it is observed that the numerical models calculate similar $\zeta$ values for three different grid sizes, despite the fact that the number of grid cells used by the refined mesh is approximately tenfold the number of cells used by the coarse grid. The maximum deviation between all three grid sizes which has been found is 1.21% between the coarse grid and medium grid for $N_f = 4$ inch$^{-1}$. For all fin density values, maximum deviations less than 1% have been observed between the medium grid and the refined grid.

Just as the (improved) analytical method and the numerical models, the ESCOA correlation method also calculates an increased value of $\zeta$ for an increased fin density. Responsible for this increase is coefficient $C_4$ calculated by Equation 2-17 listed in Section 2-3, which calculates higher values of $C_4$ for increased fin density values. Similar to the $U_{\text{bare}}$ comparison, the ESCOA correlation method does not explain the actual reason why the value of $C_4$ and hence $\zeta$ increase for an increased fin density. This is in contrast with the (improved) analytical method and the numerical models, from which the plots in Figure 4-5 are explained by means of a solid physical foundation, either by observation of the derivation corresponding to the analytical method or by observation of the post-processing plots. In addition, the ESCOA correlation method does not provide any information related to over which length the pressure drop has been calculated. Since the correlations for the pressure drop have been obtained from wind tunnel experiments, it can be assumed that this length is considerably larger compared to the length used by the (improved) analytical method and the numerical models. Consequently, the values of the pressure drop and therefore $\zeta$ calculated by the
ESCOA correlation method are considerably lower compared to $\zeta$ calculated by the other two methods. Because of this realization, it is concluded that the comparison between $\zeta$ calculated by the ESCOA correlation method and $\zeta$ calculated by the other two methods would not be a fair one.

**Pressure Drop Coefficient Comparison for 6'' Tubes**

The comparison of $\zeta$ versus the fin density calculated by various methods for 6’’ tubes is shown in Figure 4-7. Various similarities can be observed between the $\zeta$ comparison for 4’’ tubes and 6’’ tubes. All methods calculate increased values of $\zeta$ for increased fin densities, similar to the $\zeta$ comparison for 4’’ tubes. The arguments for this increase have already been discussed elaborately for all methods during the comparison for 4’’ tubes, and the same arguments are adopted to explain the increased values of $\zeta$ for increased fin densities for 6’’ tubes. In addition, it can be observed that the improved analytical method and the numerical models calculate similar results for $\zeta$ for common fin density and mass flux values. Also the asymptotic value (and even a slight decrease) calculated by the (improved) analytical method can be observed again in Figure 4-7, which repeatedly can be attributed to the fact that for high fin densities the fin spacing is reduced to such an extent that the flow between the fins is minimal and the pressure drop coefficient approaches the value corresponding to a bare tube with diameter $D = d_A + 2h$. To conclude, another similarity between the comparison for 4’’ tubes and 6’’ tubes is that the flow length used by the ESCOA correlation method to calculate the pressure drop is different compared to the length used by the (improved) analytical method and the numerical models. This makes the examination of $\zeta$ calculated by the ESCOA correlation method and the other two methods still an unfair comparison, because the ESCOA correlation method includes pressure recovery and the other two methods do not.

![Figure 4-7: Comparison of the pressure drop coefficient relative to the bare tube area $\zeta$ calculated by various methods versus the fin density $N_f$ for 6” tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s.](image-url)
Differences are also present between the $\zeta$ comparison for 4" tubes and 6" tubes. It is shown in Figure 4-7 that the overall values of $\zeta$ calculated by the (improved) analytical method and the ESCOA correlation method are considerably lower compared to the 4" tube comparison. This can primarily be attributed to the increased transverse tube pitch $t_q$ corresponding to 6" tubes. According to the derivation of the (improved) analytical method, the increased tube pitch $t_q$ corresponds to an increase in the flow area between the tubes $A_{flow,free}$. It has been touched upon during the $U_{bare}$ comparison for 6" tubes that the longer flow path length $L_{flow,fin}$ increases the fin resistance and that the increased transverse tube pitch $t_q$ makes it easier for the flue gas flow to bypass the fins, therefore reducing the mass flux between the fins $G_{fin}$. Although this causes an increase in the mass fraction which bypasses the fins, $G_{free}$ is still reduced because of the increased $A_{flow,free}$. Consequently, the velocity $u$ of the flow between the fins and the velocity $u$ of the flow between the tubes is also reduced, and as a result of this the increased transverse tube pitch $t_q$ does not only causes heat transfer to decrease but also causes the pressure drop and therefore $\zeta$ to decrease for 6" tubes compared to 4" tubes. The same can be concluded from the results calculated by the numerical models. It has already been mentioned that the average velocity $u$ between the fins is decreased for increased tube sizes according to Table 4-1 and Table 4-2. In addition, observation of the velocity isosurfaces for 4" and 6" tubes in Figure A-1 and Figure A-2 reveal that also the velocity $u$ of the flow which bypasses the fins is higher for 4" tubes compared to 6" tubes. Similar to the (improved) analytical method, this decrease in velocity $u$ should result in lower $\zeta$ values for 6" tubes compared to 4" tubes. The increase from 4" tubes to 6" tubes causes the ESCOA correlation method to calculate lower values of coefficients $C_2$ and $C_4$ according to Equation 2-16 and Equation 2-17. As a result, lower values of $\zeta$ are obtained for 6" tubes compared to 4" tubes. Compared to the (improved) analytical method and the numerical models however, this explanation lacks a physical foundation which is considered to be a major drawback of the ESCOA correlation method.

Related to $\zeta$ calculated by the numerical models, the maximum deviation between all three grid sizes which has been found is 1.93% between the coarse grid and medium grid for $N_f = 4$ inch$^{-1}$. For all fin density values, maximum deviations less than 1% have been observed between the medium grid and the refined grid.

**Pressure Drop Coefficient Comparison for 8" Tubes**

The comparison of $\zeta$ versus the fin density calculated by various methods for 8" tubes is shown in Figure 4-8. Much alike the similarities which have been found for the $\zeta$ comparison between 4" tubes and 6" tubes, the same similarities can be observed for the $\zeta$ comparison corresponding to 8" tubes. This includes the increased values of $\zeta$ for increased fin densities, the similar results for $\zeta$ calculated by the (improved) analytical method and the numerical models for common fin density and mass flux values, the seemingly asymptotic value calculated by the (improved) analytical method, and the unfair comparison related to the ESCOA correlation method. These similarities and their corresponding arguments have already been discussed elaborately for all methods during the $\zeta$ comparison for 4" tubes and 6" tubes, and the same arguments are adopted to explain the $\zeta$ plots for 8" tubes.
It is shown in Figure 4-8 that the values of $\zeta$ calculated by all methods are lowered further for 8" tubes compared to 6" tubes. Again, this can be attributed to the increased transverse tube pitch $t_q$ corresponding to 8" tubes compared to 6" tubes. The reason why the increased transverse tube pitch $t_q$ can be seen as the cause for this decrease has already been touched upon elaborately during the $\zeta$ comparison for 6" tubes. In short, it leads to a reduction in velocity $u$ of the flue gas, both of the flow between the fins and the flow which bypasses the fins. This is also supported by the velocity isosurfaces for 8" tubes in Figure A-3, where it is observed that the velocity $u$ is reduced compared to the velocity $u$ shown in the velocity isosurface figures for 6" tubes in Figure A-2. The reduction in velocity $u$ ultimately leads to lower values of $\zeta$. The ESCOA correlation method calculates even lower values for coefficients $C_2$ and $C_4$ for 8" tubes compared to 6" tubes, and as a result lower values of $\zeta$ are obtained for 8" tubes compared to 6" tubes. Similar to the 4" tube and 6" tube comparison however, the ESCOA correlations lack a physical foundation and do not provide insight in the working principle of fins.

The plots calculated by the numerical models in Figure 4-8 show a clear stagnation of the increase in $\zeta$, which is in contrast with the $\zeta$ plots calculated for 4" tubes and 6" tubes. Although the stagnation of the increase in $\zeta$ was already observable for 6" tubes, the effect is more pronounced for 8" tubes. It has already been discussed that the (improved) analytical method also calculates asymptotic values for $\zeta$, including for 4" tubes and 6" tubes. The reason for this was that the (improved) analytical method assumes $\rho_{\text{free}}$ and $\rho_{\text{fin}}$ to be constant. For high fin densities, it applies that $\rho_{\text{avg}} \approx \rho_{\text{free}}$ according to Equation 4-4 because the fin spacing is reduced to such an extent that the flow between the fins is minimal. Therefore, $\zeta$ approaches the value corresponding to a bare tube with diameter $D = d_A + 2h$. The asymptotic value for $\zeta$ is reached more rapidly for 8" tubes compared to 4" tubes and 6" tubes due to the increased fin resistance and increased transverse tube pitch corresponding.
to larger tube sizes. It was concluded by observation of the average velocity \( u \) between the fins in Table 4-1, Table 4-2, and Table 4-3 that the fin resistance is not as high compared to the (improved) analytical method. In addition, \( \rho_{\text{fin}} \) is not constant. Especially for high fin densities where the flue gas mass flow between the fins is low, higher values of \( \rho_{\text{fin}} \) are obtained. Both the lower fin resistance and the higher density of the flue gas flow between the fins \( \rho_{\text{fin}} \) corresponding to the numerical models result in a higher average density \( \rho_{\text{avg}} \) for increased fin densities compared to the (improved) analytical method according to Equation 4-4, which explains the ongoing increase of \( \zeta \) calculated by the numerical models. However, for 6" tubes and 8" tubes accompanied by high fin densities, \( \rho_{\text{fin}} \) stagnates because the flue gas flow between the fins cannot be cooled down any further (due to the longer flow path length \( L_{\text{flow,fin}} \) and lower velocity \( u \) between the fins), and the fin resistance has become sufficiently large to deduce that \( \rho_{\text{avg}} \approx \rho_{\text{free}} \). As a result, asymptotic values of \( \zeta \) which approach the value corresponding to a bare tube with diameter \( D = d_A + 2h \) are also obtained for the numerical models.

Related to \( \zeta \) calculated by the numerical models, the maximum deviation between all three grid sizes which has been found is 2.36\% between the coarse grid and medium grid for \( N_f = 2 \) inch\(^{-1} \). For all fin density values, maximum deviations less than 1\% have been observed between the medium grid and the refined grid which is similar to the deviations found for the \( \zeta \) comparison for 4" tubes and 6" tubes.

### 4-3 Results for the Investigation of Multiple Tube Rows

#### 4-3-1 Heat Transfer Coefficient Comparison for 4" Tubes

The heat transfer and pressure drop coefficient relative to the bare tube area have been studied thoroughly for one tube row. In addition to this, the effect of multiple tube rows on the values of \( U_{\text{bare}} \) has been investigated also. This was done for 4" tubes and for 6" tubes. Related to the investigation of multiple tube rows, 8" tubes have not been considered. Similar to the investigation of one tube row, the results and differences in \( U_{\text{bare}} \) calculated by all three methods corresponding to 8" tubes would probably only magnify the results and differences which have already been observed for 6" tubes. However, it would be a logical suggestion for further research to also investigate the effect of multiple tube rows on the values of \( U_{\text{bare}} \) for 8" tubes.

The comparison of \( U_{\text{bare}} \) versus the number of tube rows \( N_R \) calculated by various methods for 4" tubes is shown in Figure 4-9. A fin density value of \( N_f = 4 \) inch\(^{-1} \) and an average mass flux of \( G_{\text{avg}} = 2 \) kg/m\(^2\)·s were used to perform the investigation for multiple tube rows. Similar to the investigation related to one tube row, the green line represents the ESCOA correlation method, the red and the yellow line represent the initial and the improved analytical method respectively, and the blue line represents the results calculated by the numerical models performed on a medium grid. No numerical calculations on a coarse grid or on a refined grid have been performed. The reason for this is that the medium grid provides a good balance between speed and accuracy. It was observed during the investigation of one tube row that the values of \( U_{\text{bare}} \) calculated by a coarse grid are still grid sensitive and are therefore insufficiently accurate, whereas the results obtained by a medium grid deviated less than 1\% from \( U_{\text{bare}} \) calculated by a refined grid. Up to \( N_R = 4 \) have been investigated for all methods,
which are depicted by the solid lines in Figure 4-9. Only for the ESCOA correlation method $U_{bare}$ has been calculated for a larger number of tube rows up to $N_R = 10$. The reason for this is threefold. First, the ESCOA correlation method was designed for multiple tube rows. Therefore, it can be concluded that the ESCOA correlation method calculates accurate results for a large number of tube rows for 4" tubes. Second, calculations performed by the numerical models would become increasingly more time-consuming and are therefore limited to a maximum number of tube rows of $N_R = 4$. Third, it can be observed from Figure 4-9 (and Figure 4-10 corresponding to the investigation of multiple tube rows for 6" tubes) that $U_{bare}$ calculated by the (improved) analytical method and the numerical models is already stagnating for $N_R = 4$. As a result, it is not significant to calculate $U_{bare}$ for an increased number of tube rows.

This stagnation of $U_{bare}$ shown in Figure 4-9 is not only observable for the (improved) analytical method and the numerical models, but is also present for the ESCOA correlation method. Each subsequent tube row transfers less heat due to a decrease in temperature difference between the incoming flue gas temperature and the tube temperature $T_{tube}$. At a certain point, the flow regime becomes quasi-stationary. This results in the stagnation of $U_{bare}$ according to Equation 4-5, which has been used to calculate $U_{bare}$ for the (improved) analytical method and the numerical models:

$$U_{bare} = \frac{Q_{fin}}{A_{bare}\Delta T_{lm}} \quad (4-5)$$

It can be observed from this equation that the logarithmic mean temperature difference $\Delta T_{lm}$ is used for the multiple tube row investigation instead of $T_{gm} - T_{tube}$, which was used for the investigation of one tube row according to Equation 4-1. Corresponding to the design of fired heaters and heat exchangers in general, temperature differences between the hot flue gas and tube wall are not constant, even for ideal situations. Although $T_{gm} - T_{tube}$ (with
\[ T_{gm} = 0.5(T_{in} - T_{out}) \] is a good approximation to calculate \( U_{bare} \) for one tube row, this is not the case for multiple tube rows. It has been discussed during the derivation of the analytical method in Section 2-4 that the temperature of the tube wall \( T_{tube} \) is assumed to be constant for one tube row. For the investigation of multiple tube rows, it is assumed that the subsequent tube rows also have a constant temperature with the same \( T_{tube} \) as the first tube row. As a result, the logarithmic mean temperature difference for multiple tube rows is calculated as follows:

\[
\Delta T_{lm} = \frac{(T_{in} - T_{tube}) - (T_{out} - T_{tube})}{\ln(T_{in} - T_{tube}) - \ln(T_{out} - T_{tube})}
\] (4-6)

The ESCOA correlation method takes the effect of multiple tube rows into account by calculation of coefficient \( C_5 \) in Equation 2-7, which approaches a value of \( C_5 = 1 \) for an increasing number of tube rows. Since only the number of tube rows are adjusted, this results in stagnation of \( U_{bare} \). Similar to the investigation for one tube row, this explanation lacks a physical foundation due to the fact the ESCOA correlation method has been derived from wind tunnel experiments. In addition, only the mean gas temperature \( T_{gm} \) (with \( T_{gm} = T_{in} \)) and mean fin temperature \( T_{rm} = 0.5(T_{gm} + T_{tube}) \) are used to calculate \( U_{bare} \), independent of the number of tube rows which have been used. This means that similar to the (improved) analytical method and the numerical models, the ESCOA correlation method also specifies one \( T_{tube} \) for all tube rows.

As expected, it is shown in Figure 4-9 that for \( N_R = 1 \) the values of \( U_{bare} \) calculated by all methods are identical to the values calculated in Figure 4-1 for \( N_f = 4 \) inch\(^{-1}\). Furthermore, for an increased number of tube rows, \( U_{bare} \) calculated by both the ESCOA correlation method and the numerical models approximately attains the same the asymptotic value. Thus, not only for one tube row, but also for multiple tube rows the results obtained by the ESCOA correlation method and the numerical models are in close agreement with each other. This supports the conclusion already drawn during the investigation of one tube row for 4" tubes that the numerical models which have been developed are valid, since the ESCOA correlation method has proven to be a valid method to predict heat transfer in the convection section of fired heaters for common fin density and average mass flux values for 4" tubes. For \( 2 \leq N_R \leq 4 \) however, the values of \( U_{bare} \) calculated by the numerical models are considerably higher compared to the values calculated by the ESCOA correlation method. A closer look at the velocity isosurfaces for multiple tube rows and velocity profiles in Figure B-1 and Figure B-3 shows that the upstream flue gas flow which bypasses the fins corresponding to the first tube row gets accelerated. This accelerated flue gas is then forced to flow between the fins corresponding to the second tube row. As a result, the velocity of the flue gas flow between the fins is increased for the second tube row compared to the first tube row, and subsequently relatively more heat is transferred for the second tube row compared to the first tube row. This effect is already moderated for the third and fourth tube row which explains the stagnation of \( U_{bare} \) calculated by the numerical models. Since the plot calculated by the ESCOA correlation method represents a regression analysis, the effects corresponding to the second tube row which results in increased heat transfer may not be captured properly.

The \( U_{bare} \) values calculated by the initial analytical method are surprisingly similar to the values calculated by the ESCOA correlation method and the numerical models for an increased number of tube rows. However, it has been concluded during the investigation for one tube row that \( U_{bare} \) calculated by the improved analytical method are considerably more plausible compared to the values obtained by the initial analytical method. As a result, the
close agreement between the initial analytical method and the other two methods can be disregarded. However, it is also observed in Figure 4-9 that the improved analytical method calculates considerably lower $U_{\text{bare}}$ values for an increased number of tube rows compared to the other two methods. In addition, both the initial and improved analytical method do not properly model the effect of the second tube row. The (improved) analytical method uses an improvement factor for subsequent tube rows which is applied to $R$ (variable which describes the proportions of $G_{\text{free}}$ and $G_{\text{fin}}$) in Equation 2-27. Application of this improvement factor results in a lower value of $R$ and therefore results in a larger mass flux between the fins $G_{\text{fin}}$ for subsequent tube rows. Consequently, the larger mass flux $G_{\text{fin}}$ increases heat transfer which explains the slight increase in $U_{\text{bare}}$ for subsequent tube rows calculated by the (improved) analytical method. Still, the implementation of this improvement factor is not sufficient. However, the low $U_{\text{bare}}$ values compared to the other two methods and the incapability to properly model the effect of the second tube row can be attributed to another reason. The (improved) analytical method assumes the flue gas flow downstream of each tube to be fully mixed. This means that the (improved) analytical method assumes the flue gas flow downstream of each tube to have an uniform temperature. It is shown in Figure B-5 which represents the temperature profiles that this is not the case. Especially the flue gas flow which bypasses the fins corresponding to the first row, but hits the second tube row has a considerably higher temperature (close to $T_{\text{in}}$) compared to the uniform temperature calculated by the (improved) analytical method. Together with the increased $G_{\text{fin}}$ discussed previously, this high temperature difference between the incoming flue gas flow and tube temperature $T_{\text{tube}}$ leads to a significant increase in heat transfer for the second tube row. Since this is not captured by the (improved) analytical method, the steep increase in $U_{\text{bare}}$ corresponding to the second tube row does not take place and lower $U_{\text{bare}}$ values are obtained for subsequent tube rows in Figure 4-9. Although it has been confirmed that the improved analytical method provides accurate results for common fin density and mass flux values for one tube row, it can be concluded that this is not the case for multiple tube rows. Therefore, additional refinements should be made in order for the (improved) analytical method to be applicable for multiple tube rows also.

### 4-3-2 Heat Transfer Coefficient Comparison for 6" Tubes

The comparison of $U_{\text{bare}}$ versus the number of tube rows $N_R$ calculated by various methods for 6" tubes is shown in Figure 4-10. Identical to the comparison for 4" tubes, a fin density value of $N_f = 4 \text{ inch}^{-1}$ and an average mass flux of $G_{\text{avg}} = 2 \text{ kg/m}^2\cdot\text{s}$ have been used to perform the investigation for multiple tube rows.

Similar to the 4" tube comparison for multiple tube rows, the 6" tube comparison also displays the stagnation of $U_{\text{bare}}$ for all methods in Figure 4-10. The arguments for this stagnation have already been discussed elaborately during the 4" tube comparison for multiple tube rows, and are identical to the stagnation corresponding to 6" tubes. However, for a large number of tube rows it is observed that the ESCOA correlation method overpredicts $U_{\text{bare}}$ compared to the values calculated by the numerical models. The reason for this definition instead of stating that the numerical models underpredict $U_{\text{bare}}$ compared to the values calculated by the ESCOA correlation method can be attributed to the fact that the results calculated by the numerical models are the most accurate, which was already concluded from the investigation of one tube row for 6" tubes in Figure 4-3. During this investigation it was mentioned that
extrapolation of the ESCOA correlations to cope with 6" tubes and even 8" tubes leads to an increased divergence from the results obtained by wind tunnel experiments, which have been performed mostly for 1" tubes and 2" tubes. As a result, $U_{bare}$ calculated by the ESCOA correlation method should be questioned for 6" tubes and 8" tubes, and it is not expected that the results calculated by ESCOA correlation method and the numerical models should be in close agreement with each other for these tube sizes.

It is also shown in Figure 4-10 that $U_{bare}$ values calculated by the (improved) analytical method are still lower compared to the values calculated by the numerical models. It has been discussed before that the (improved) analytical method does not sufficiently take the effect of the second tube row into account. Consequently, considerably lower $U_{bare}$ values are calculated. The reason for this has already been explained during the investigation of multiple tube rows for 4" tubes, and the same arguments are adopted for the investigation of multiple tube rows related to 6" tubes. Corresponding to the numerical models, a closer look at the velocity profiles in Figure B-4 show a reduction in velocity $u$ of the flue gas flow which bypasses the first tube row compared to the 4" tube investigation. As a result of this reduction in velocity $u$, the flue gas flow is to a lesser extent forced to flow between the fins of the second tube row. Consequently, the velocity $u$ between the fins of the second tube row is also reduced which is shown in the velocity isosurfaces for multiple tube rows in Figure B-2. This reduction in velocity $u$, both for the first and second tube row can be attributed to the increased transverse tube pitch $t_q$ corresponding to 6" tubes. The increased transverse tube pitch $t_q$ increases the flow area which bypasses the fins $A_{flow,free}$, and as a result the velocity $u$ is reduced. However, it can be observed in figure 4-10 that the effect of the second tube row is still significant. Observation of the temperature profiles in Figure B-6 reveals that the hot flue gas flow which bypasses the first tube row is increased for 6" tubes compared to 4" tubes, which is an additional consequence of the increased transverse tube pitch $t_q$ corresponding to

Figure 4-10: Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ calculated by various methods versus the number of tube rows $N_R$ for 6'" tubes. Fin density $N_f = 4 \text{ inch}^{-1}$, average mass flux $G_{avg} = 2 \text{ kg/m}^2\cdot\text{s}$. 
6” tubes. In addition, this hot flue gas flow hits a greater subsequent tube row area for 6” tubes compared to 4” tubes. As a result, more heat is transferred and the effect of the second tube row is still substantial.

In contrast with the investigation of multiple tube rows for 4” tubes, it is shown in Figure 4-10 that the improved analytical method calculates higher values of $U_{bare}$ compared to the values calculated by the initial analytical method. The reason for this can be explained by recollection of the $U_{bare}$ comparison for one tube row. For 4” tubes and $N_f = 4$ inch$^{-1}$, it is shown in Figure 4-1 that the initial analytical method calculates higher $U_{bare}$ values compared to the improved analytical method, while for 6” tubes and $N_f = 4$ inch$^{-1}$ the opposite is true and the improved analytical method calculates higher $U_{bare}$ values compared to the initial analytical method according to Figure 4-3. Apparently, this observation for one tube row persists for multiple tube rows.

4-4 Results for the Investigation of Two Fin Configurations

4-4-1 Heat Transfer Coefficient Comparison for 6” Tubes

Prior research related to the investigation of one tube row and multiple tube rows have proven that the numerical models which have been developed are valid. This allows for an additional investigation related to finned tube heat transfer. It has been discussed elaborately during the investigation for one tube row and for multiple tube rows that a considerable part of the decrease in heat transfer can be attributed to the increased transverse tube pitch $t_q$ corresponding to 6” tubes and 8” tubes. Because of this increase, an approach to provide a solution for this problem would be to increase the fin height. This increase in fin height would result in dimensional ratios which are more in line with 4” tubes. According to API Standard 560 however, the maximum allowed fin height is restricted to $h = 1”$ (American Petroleum Institute, 2001). Neglecting this guideline would allow for an interesting comparison between the fin height and fin density. To perform this investigation, two different fin configurations have been proposed:

- **Fin configuration A**: Fin height $h = 1”$, fin density $N_f = 5$ inch$^{-1}$
- **Fin configuration B**: Fin height $h = 2”$, fin density $N_f = 2.5$ inch$^{-1}$

These two fin configurations approximately have the same extension ratio. However, it is expected that two different $U_{bare}$ plots will be obtained, because the flow and heat transfer characteristics of these fin configurations are substantially different.

The comparison of $U_{bare}$ between two different fin configurations calculated by various methods for 6” tubes is shown in Figure 4-11. Only one tube row has been considered. Similar to the prior investigations, the green lines represent the results obtained by the ESCOA correlation method, the red and yellow lines the results obtained by the initial analytical method and the improved analytical method respectively, and the blue lines represent the results obtained by the numerical models performed on a medium grid. The solid lines represent fin configuration A, while fin configuration B is represented by the dashed lines. For high average mass flux $G_{avg}$ values, the flow between the fins is not strictly laminar anymore according to
Results and Discussion

Figure 4-11: Comparison of the heat transfer coefficient relative to the bare tube area $U_{bare}$ between two different fin configurations calculated by various methods for 6” tubes. $N_R = 1$.

With the exception of the ESCOA correlation method, it is observed in Figure 4-11 that fin configuration B is preferred over fin configuration A in terms of heat transfer for $G_{avg} \leq 3$ kg/m²·s. The reason for this is rather simple and intuitive. Because of the increased fin height $h$ and lower fin density, the flow area between the fins $A_{flow, fin}$ is increased and the flow area bypassing the fins $A_{flow, free}$ is decreased for fin configuration B compared to fin configuration A. This greatly reduces the fin resistance and results in more flue gas mass flow between the fins $\dot{m}_{fin}$. This is in contrast with fin configuration A. The small $A_{flow, fin}$ and large $A_{flow, free}$, which are the result of the high fin density and short fin height $h$, increase the fin resistance. For low $G_{avg}$, the flue gas flow is diverted around the fins which decreases the flue gas mass flow between the fins $\dot{m}_{fin}$. This flue gas mass flow deficit between the fins $\dot{m}_{fin}$ leads to the lower $U_{bare}$ values for fin configuration A compared to fin configuration B.

For $3 \leq G_{avg} \leq 4$ kg/m²·s, a transition takes place where fin configuration A catches up with fin configuration B in terms of heat transfer. Observation of the derivation corresponding to the (improved) analytical method shows that the heat exchanger effectiveness $\epsilon$ is increased significantly for fin configuration A compared to fin configuration B. According to Equation 2-39, this should lead to more heat transferred per meter of tube $Q_{fin}$, which should therefore also result in increased $U_{bare}$ values. Analogous to fin configuration A however is the reduced flue gas mass flow between the fins $\dot{m}_{fin}$, and according to the same equation this decreases $Q_{fin}$ and therefore decreases $U_{bare}$ again. It has been discussed before that this results in lower values of $U_{bare}$ for fin configuration A compared to fin configuration B for low $G_{avg}$, but for $3 \leq G_{avg} \leq 4$ kg/m²·s, the average mass flux is eventually sufficiently high to overcome the high fin resistance which results in a drastic increase in $U_{bare}$ for fin configuration A.
The results obtained by the numerical models also display the behavior where fin configuration A catches up with fin configuration B in terms of heat transfer for \(3 \leq \frac{G_{\text{avg}}}{\text{kg/m}^2\cdot\text{s}} \leq 4\). The reason for this is similar to the reason explained for the (improved) analytical method. For low \(G_{\text{avg}}\), the velocity \(u\) between the fins is reduced to such an extent that the temperature of the flue gas flow approaches the tube temperature \(T_{\text{tube}}\), which is observed from the temperature isosurfaces for configuration A shown in Figure C-1. The flow between the fins acts like insulation, which ultimately results in the low \(U_{\text{bare}}\) for configuration A. The opposite applies to configuration B, which can be observed from the temperature isosurfaces for configuration B shown in Figure C-2. The fin spacing and therefore the velocity \(u\) between the fins and is sufficiently large for the hot flue gas to reach the flow area between the fins to exchange heat, which explains the difference between both fin configurations for low \(G_{\text{avg}}\). For an increased average mass flux, the flue gas velocity \(u\) is sufficiently high to overcome the high fin resistance corresponding to fin configuration A. This can be seen in Figure C-1, showing that the hot flue gas is able to reach the flow area between the fins to exchange heat. For fin configuration B however, it is shown in Figure C-2 that an increased average mass flux does not necessarily result in a large increase in \(U_{\text{bare}}\). The hot flue gas flow was already able to reach the flow area between the fins for low \(G_{\text{avg}}\), and as a result, the increase in \(U_{\text{bare}}\) is not as significant compared to fin configuration A.

The ESCOA correlation method calculates a larger fin area \(A_{\text{xfer, fin}}\) for fin configuration B compared to fin configuration A. For fin configuration B however, a substantially lower fin efficiency \(E\) is calculated, and overall this results in lower \(U_{\text{bare}}\) values for fin configuration B compared to fin configuration A over the whole \(G_{\text{avg}}\) domain investigated. It can be concluded that the ESCOA correlation method fails to distinguish the two fin configurations properly, and similar to the investigation for one tube row and for multiple tube rows the ESCOA correlation method is not able provide a clear explanation for the plots which have been calculated. This is in contrast with the results obtained by both the (improved) analytical method and the numerical models, from which the explanation is notably more comprehensive.

Despite the lack of a proper explanation for the plots calculated by the ESCOA correlation method, it is a conspicuous observation in Figure 4-11 that the results calculated by the ESCOA correlation method and the numerical models related to fin configuration B are almost identical. Beside the possibility of coincidence, the reason for this resemblance could not be explained.

4-4-2 Heat Transfer Coefficient Comparison for 8" Tubes

The comparison of \(U_{\text{bare}}\) between two different fin configurations calculated by various methods for 8" tubes is shown in Figure 4-12. Again, only one tube row has been considered. The methods are represented by their corresponding colors. Similar to the fin configuration comparison for 6" tubes, the solid lines represent fin configuration A, while fin configuration B is represented by the dashed lines. The grey dashed lines depict the region again where the (improved) analytical method is invalid, since the flow between the fins is not strictly laminar anymore (\(Re_{\text{fin}} > 2300\)).
Similar results which have been found and discussed for the 6" tube comparison were observed for the 8" tube comparison. With the exception of the ESCOA correlation method, fin configuration B is preferred over fin configuration A in terms of heat transfer for low $G_{avg}$. However, the preference for fin configuration B has shifted to the right towards $G_{avg} \leq 4$ kg/m$^2$·s, making fin configuration B the preferred configuration for a wider range of the average mass flux. The explanation for this shift can be attributed to the increased fin resistance corresponding to 8" tubes. The reason for this increase in fin resistance has already been explained elaborately during the investigation of one tube row. First, an increased fraction of the flue gas flow bypasses the fins due to the increased transverse tube pitch $t_q$ compared to 6" tubes. Second, the fin resistance increases due to the longer flow path length of the flow between the fins $L_{flow,fin}$ corresponding to 8" tubes. Fin configuration B solves this problem by decreasing the fin density and by increasing the fin height. Not until $4 \leq G_{avg} \leq 5$ kg/m$^2$·s a shift takes place where fin configuration A catches up with fin configuration B in terms of heat transfer. However, these values of $G_{avg}$ are already considered to be significant and are not often encountered in practice.

Similar to the fin configuration investigation for 6" tubes, the ESCOA correlation method fails to distinguish the two fin configurations properly and prefers fin configuration A over fin configuration B in terms of heat transfer over the whole $G_{avg}$ range without providing a clear physical explanation for this preference. Again, the ESCOA correlation method calculates a larger fin heat transfer surface area $A_{xfer,fin}$ for fin configuration B compared to fin configuration A. For fin configuration B however, a substantially lower fin efficiency $E$ is calculated. Ultimately, this leads to lower $U_{bare}$ values for fin configuration B compared to fin configuration A.

It can be observed in Figure 4-12 that the results obtained by the ESCOA correlation method and the numerical models for fin configuration B are again almost identical. Similar to the fin configuration comparison for 6" tubes, the reason for this resemblance could not be explained. However, since both tube sizes feature this result, it could be a suggestion for further research.
This master thesis research aimed to develop valid numerical models capable of validating an analytical method for calculation of heat transfer for solid annular-finned tubes. Based on the strong agreement between the ESCOA correlation method and the numerical models corresponding to 4” tubes for both one tube row and multiple tube rows, it can be concluded that this is goal has been achieved. For 6” tubes and 8” tubes, increased differences between the results calculated by the ESCOA correlation method and the numerical models have been observed because the ESCOA correlations were never designed for these tube sizes. Because of the large margins applied to the design of fired heaters, this shortcoming goes unnoticed and the ESCOA correlation method is still applied to 6” tubes and 8” tubes, while for the latter this method should be avoided.

In contrast to the ESCOA correlation method, the analytical method and the numerical models calculate a fin density optimum. It is concluded by observation of the post-processing plots that this can be attributed to an increased fin resistance corresponding to an increased fin density. For these fin density values, boundary layers developed on the fin surface cannot develop independently from each other anymore and touch instead. This result in a decreased flue gas mass flow between the fins and less heat transfer.

Based on a quantitative and qualitative analysis of the ESCOA correlation method, the analytical method, and the numerical models, the results indicate that the improved analytical method calculates valid results for 4” tubes for common fin density and mass flux values. The assumptions made by the analytical method are appropriate for these values, but not fin density and mass flux values which are outside this region. Because of this this shortcoming, increased differences arise between the results calculated by the analytical method and the numerical models for other fin density values. In addition, these differences are more noticeable for 6” tubes and 8” tubes. The improved analytical method addressed some of these assumptions, by excluding the wake region behind the tube made, which turned out to be a critical improvement over the initial analytical method.
It was found that the (improved) analytical method is not suitable yet to be applied for multiple tube rows. This can be attributed to the lower mass flux between the fins calculated by the (improved) analytical method compared to the mass flux calculated by numerical models, and to the assumption that the flue gas flow downstream of each tube is fully mixed. As a result, the effect of the second tube row is not modeled properly, which was found to have a large impact on heat transfer. Similar to the investigation of one tube row the ESCOA correlation method overpredicts heat transfer compared to the numerical models for 6" tubes.

The investigation of two different fin configurations has shown that fin configuration B ($h = 2\text{"}, N_f = 2.5 \text{ inch}^{-1}$) is preferred over fin configuration A ($h = 1\text{"}, N_f = 5 \text{ inch}^{-1}$) for low average mass flux values. However, these values are already substantial for fired heater applications. As a result, it was concluded that for design related to the convection section of fired heaters fin configuration B should always be preferred over fin configuration A, both for 6" and 8" tubes. The application of fin configuration B leads to dimensional ratios between bare tube diameter, fin height and tube pitch which are more in line with the ratio used for 4" tubes. Although this is favorable for heat transfer, this fin configuration cannot be used since the fin height is restricted to $h \leq 1\text{"}$ according to API Standard 560. The ESCOA correlation method is not able to distinguish the two fin configuration properly and recommends fin configuration A over fin configuration B for all average mass flux values investigated. In addition, the close resemblance between the results calculated by the ESCOA correlation method and the numerical models for configuration B could not be explained.

Although the analytical method produces valid results for common fin density and average mass flux values, it cannot be concluded that the analytical method has been validated. For that to happen, the assumptions made by the analytical method should be addressed and more improvements have to be made. However, the analytical method shows huge potential. Because of its solid physical foundation, results can be more accurately explained and discussed compared to the ESCOA correlation method, which does not provide insight in the working principle of fins. Once refined, the convection section of fired heaters could be designed more accurately in the near future, which reduces fuel burn and therefore reduces emissions.
Chapter 6

Recommendations

A significant first step has been taken over the course of this master thesis research related to the validation of the analytical method. However, further research is needed to determine if the analytical method is indeed a valid method to calculate heat transfer in the convection section of fired heaters. Future studies could further validate the numerical models made during this master thesis research and could address the application of other turbulence models which are suitable for this master thesis research as well. Based on the RANS equations, these include the Transition SST turbulence model and the RSM, but also turbulence modeling based on LES should be examined. Compared to the SIMPLE algorithm, more advanced solution algorithms such as the SIMPLEC and Coupled algorithms could be adopted.

Parallel to the further development of the numerical models, assumptions made by the analytical method have to be addressed and further improvements should made to the analytical method as well. Three suggestions for improvements would be the integration of fin tip heat transfer, dynamic pressure, and to make the analytical method applicable for multiple tube rows. It is expected that when the improvements for both the numerical models and the analytical method have been incorporated, the results calculated by both methods will be more similar for an increased fin density and average mass flux range. In addition, besides applications related to fired heaters, a wider operational envelope and smaller tube sizes could be investigated.

With regard to fin and tube arrangements, future studies could investigate serrated fins and in-line tube arrangements. Fins could be modeled as a spiral instead of a circle, and the effect of imperfect fins which arises because of stretching and compressing during the manufacturing process could be investigated. Related to the investigation of multiple tube rows, 8' tubes could be investigated as well in addition to 4' tubes and 6' tubes. Also, the investigation of $U_{bare}$ and $\zeta$ versus the average mass flux instead of the fin density could be a suggestion for future research. Other subjects for future studies include the analysis of secondary effects induced by horseshoe vortices and the examination of similarities between the results calculated by the ESCOA correlation method and the numerical models for fin configuration B. To conclude, actual (scaled down) wind tunnel experiments could be performed for comparison with the methods which have been investigated.
Appendix A

Post-Processing Figures Related to the Investigation of One Tube Row
A-1 Velocity Isosurfaces

The velocity isosurfaces have been imposed at the cross-section of the finned tube where the total flow area $A_{\text{flow,tot}}$ is the smallest, which is calculated as the domain inlet area $A_{\text{inlet}}$ minus the projected area of the tube minus the projected area of the fin. The area $A_{\text{flow,tot}}$ is represented by the white area in Figure 3-15 in Chapter 3.

Velocity Isosurfaces for 4" Tubes (Medium Grid)

![Velocity Isosurfaces for 4" Tubes](image)

Figure A-1: Velocity isosurfaces for 4" tubes. Left: $N_f = 2$ inch$^{-1}$. Center: $N_f = 4$ inch$^{-1}$. Right: $N_f = 8$ inch$^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{\text{avg}} = 2$ kg/m$^2$.s.
Velocity Isosurfaces for 6" Tubes (Medium Grid)

Figure A-2: Velocity isosurfaces for 6" tubes. Left: $N_f = 2$ inch$^{-1}$. Center: $N_f = 4$ inch$^{-1}$. Right: $N_f = 8$ inch$^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s.
Velocity Isosurfaces for 8" Tubes (Medium Grid)

Figure A-3: Velocity isosurfaces for 8\" tubes. Left: \( N_f = 2 \text{ inch}^{-1} \). Center: \( N_f = 4 \text{ inch}^{-1} \). Right: \( N_f = 8 \text{ inch}^{-1} \). Number of tube rows \( N_R = 1 \), average mass flux \( G_{avg} = 2 \text{ kg/m}^2\cdot\text{s} \).
The velocity $u$ profiles have been imposed at half the fin height $h$ and enclose the entire computational domain thickness $Z_1$ to obtain the XY-plots.

**XY-Plots for 4'' Tubes (Medium Grid)**

![XY-Plots for 4'' Tubes](image1)

**Figure A-4:** XY-plots illustrating the velocity $u$ profile between the fins at half the fin height $h$ for various fin density values for 4'' tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2 \text{ kg/m}^2 \cdot \text{s}$.

**XY-Plots for 6'' Tubes (Medium Grid)**

![XY-Plots for 6'' Tubes](image2)

**Figure A-5:** XY-plots illustrating the velocity $u$ profile between the fins at half the fin height $h$ for various fin density values for 6'' tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2 \text{ kg/m}^2 \cdot \text{s}$.
XY-Plots for 8\textquotedbl{} Tubes (Medium Grid)

Figure A-6: XY-plots illustrating the velocity $u$ profile between the fins at half the fin height $h$ for various fin density values for 8\textquotedbl{} tubes. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s.
A-3 Temperature Isosurfaces

The temperature isosurfaces have been imposed at the cross-section of the finned tube where the total flow area $A_{\text{flow,tot}}$ is the smallest, which is calculated as the domain inlet area $A_{\text{inlet}}$ minus the projected area of the tube minus the projected area of the fin. The area $A_{\text{flow,tot}}$ is represented by the white area in Figure 3-15 in Chapter 3.

Temperature Isosurfaces for 4" Tubes (Medium Grid)

![Temperature isosurfaces for 4" tubes](image)

Figure A-7: Temperature isosurfaces for 4" tubes. Left: $N_f = 2 \text{ inch}^{-1}$. Center: $N_f = 4 \text{ inch}^{-1}$. Right: $N_f = 8 \text{ inch}^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2 \text{ kg/m}^2\cdot\text{s}.\]
Temperature Isosurfaces for 6" Tubes (Medium Grid)

Figure A-8: Temperature isosurfaces for 6" tubes. Left: $N_f = 2$ inch$^{-1}$. Center: $N_f = 4$ inch$^{-1}$. Right: $N_f = 8$ inch$^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s.
Temperature Isosurfaces for 8" Tubes (Medium Grid)

Figure A-9: Temperature isosurfaces for 8" tubes. Left: $N_f = 2 \text{ inch}^{-1}$. Center: $N_f = 4 \text{ inch}^{-1}$. Right: $N_f = 8 \text{ inch}^{-1}$ Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2 \text{ kg/m}^2\cdot\text{s}$. 
A-4 Finned Tube Temperature Contours

Finned Tube Temperature Contours for 4" Tubes (Medium Grid)

Figure A-10: Finned tube temperature contour for 4" tubes and \( N_f = 2 \) inch\(^{-1} \). Number of tube rows \( N_R = 1 \), average mass flux \( G_{avg} = 2 \) kg/m\(^2\)-s. Flow direction is from left to right.

Figure A-11: Finned tube temperature contour for 4" tubes and \( N_f = 4 \) inch\(^{-1} \). Number of tube rows \( N_R = 1 \), average mass flux \( G_{avg} = 2 \) kg/m\(^2\)-s. Flow direction is from left to right.
Figure A-12: Finned tube temperature contour for 4" tubes and $N_f = 8$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right.
Finned Tube Temperature Contours for 6" Tubes (Medium Grid)

Figure A-13: Finned tube temperature contour for 6" tubes and \( N_f = 2 \) inch\(^{-1} \). Number of tube rows \( N_R = 1 \), average mass flux \( G_{\text{avg}} = 2 \) kg/m\(^2\)-s. Flow direction is from left to right.

Figure A-14: Finned tube temperature contour for 6" tubes and \( N_f = 4 \) inch\(^{-1} \). Number of tube rows \( N_R = 1 \), average mass flux \( G_{\text{avg}} = 2 \) kg/m\(^2\)-s. Flow direction is from left to right.
Figure A-15: Finned tube temperature contour for 6º tubes and \( N_f = 8 \) inch\(^{-1} \). Number of tube rows \( N_R = 1 \), average mass flux \( G_{avg} = 2 \) kg/m\(^2\)-s. Flow direction is from left to right.
Finned Tube Temperature Contours for 8" Tubes (Medium Grid)

**Figure A-16:** Finned tube temperature contour for 8" tubes and $N_f = 2$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. Flow direction is from left to right.

**Figure A-17:** Finned tube temperature contour for 8" tubes and $N_f = 4$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. Flow direction is from left to right.
Figure A-18: Finned tube temperature contour for 8" tubes and $N_f = 8$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$s. Flow direction is from left to right.
A-5 Surface Heat Transfer Coefficient Contours

Surface Heat Transfer Coefficient Contours for 4" Tubes (Medium Grid)

Figure A-19: Surface heat transfer coefficient contour for 4" tubes and $N_f = 2$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. Flow direction is from left to right.

Figure A-20: Surface heat transfer coefficient contour for 4" tubes and $N_f = 4$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$.s. Flow direction is from left to right.
Figure A-21: Surface heat transfer coefficient contour for 4" tubes and $N_f = 8$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right.
Surface Heat Transfer Coefficient Contours for 6" Tubes (Medium Grid)

**Figure A-22:** Surface heat transfer coefficient contour for 6" tubes and $N_f = 2$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right.

**Figure A-23:** Surface heat transfer coefficient contour for 6" tubes and $N_f = 4$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right.
Figure A-24: Surface heat transfer coefficient contour for 6" tubes and $N_f = 8$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right.
Surface Heat Transfer Coefficient Contours for 8" Tubes (Medium Grid)

Figure A-25: Surface heat transfer coefficient contour for 8" tubes and $N_f = 2$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right.

Figure A-26: Surface heat transfer coefficient contour for 8" tubes and $N_f = 4$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right.
Figure A-27: Surface heat transfer coefficient contour for 8" tubes and $N_f = 8$ inch$^{-1}$. Number of tube rows $N_R = 1$, average mass flux $G_{avg} = 2$ kg/m$^2$·s. Flow direction is from left to right.
Appendix B

Post-Processing Figures Related to the Investigation of Multiple Tube Rows
B-1 Velocity Isosurfaces

Similar to the investigation of one tube row, the velocity isosurfaces have been imposed at the cross-section of the finned tube where the total flow area $A_{\text{flow,tot}}$ is the smallest, which is calculated as the domain inlet area $A_{\text{inlet}}$ minus the projected area of the tube minus the projected area of the fin. The area $A_{\text{flow,tot}}$ is represented by the white area in Figure 3-15 in Chapter 3.

**Velocity Isosurfaces for 4" Tubes (Medium Grid)**

![Velocity isosurfaces for 4" tubes. From left to right: first tube row, second tube row, third tube row, fourth tube row. $N_f = 4$ inch$^{-1}$, average mass flux $G_{\text{avg}} = 2$ kg/m$^2$·s.](image)

**Figure B-1:** Velocity isosurfaces for 4" tubes. From left to right: first tube row, second tube row, third tube row, fourth tube row. $N_f = 4$ inch$^{-1}$, average mass flux $G_{\text{avg}} = 2$ kg/m$^2$·s.
Velocity Isosurfaces for 6" Tubes (Medium Grid)

Figure B-2: Velocity isosurfaces for 6" tubes. From left to right: first tube row, second tube row, third tube row, fourth tube row. \( N_f = 4 \text{ inch}^{-1} \), average mass flux \( G_{\text{avg}} = 2 \text{ kg/m}^2\text{s} \).
B-2 Velocity Profiles

The velocity profiles have been imposed at two locations: For each tube size, the top figure represents the velocity profile over the tube and the bottom figure represents the velocity profile over the fin. The black lines surrounding each finned tube represents the fin fluid domain (and not the fin).

Velocity Profile for 4" Tubes (Medium Grid)

Figure B-3: Velocity profiles for 4" tubes. Top: Velocity profile over the tube. Bottom: Velocity profile over the fin. \( \text{\(N_f\)} = \text{4 inch}^{-1} \), average mass flux \( G_{\text{avg}} = 2 \text{ kg/m}^2\cdot\text{s} \).

Velocity Profile for 6" Tubes (Medium Grid)

Figure B-4: Velocity profiles for 6" tubes. Top: Velocity profile over the tube. Bottom: Velocity profile over the fin. \( \text{\(N_f\)} = \text{4 inch}^{-1} \), average mass flux \( G_{\text{avg}} = 2 \text{ kg/m}^2\cdot\text{s} \).
B-3 Temperature Profiles

Similar to the velocity profiles, the temperature profiles have been imposed at two locations: For each tube size, the top figure represents the velocity profile over the tube and the bottom figure represents the velocity profile over the fin. The black lines surrounding each finned tube represents the fin fluid domain (and not the fin).

Temperature Profile for 4" Tubes (Medium Grid)

Figure B-5: Temperature profiles for 4" tubes. Top: temperature profile over the tube. Bottom: Temperature profile over the fin. \( N_f = 4 \text{ inch}^{-1} \), average mass flux \( G_{avg} = 2 \text{ kg/m}^2\text{s} \).

Temperature Profile for 6" Tubes (Medium Grid)

Figure B-6: Temperature profiles for 6" tubes. Top: Temperature profile over the tube. Bottom: Temperature profile over the fin. \( N_f = 4 \text{ inch}^{-1} \), average mass flux \( G_{avg} = 2 \text{ kg/m}^2\text{s} \).
C-1 Temperature Isosurfaces

Similar to the investigation of one tube row and multiple tube rows, the temperature isosurfaces have been imposed at the cross-section of the finned tube where the total flow area $A_{\text{flow,tot}}$ is the smallest, which is calculated as the domain inlet area $A_{\text{inlet}}$ minus the projected area of the tube minus the projected area of the fin. The area $A_{\text{flow,tot}}$ is represented by the white area in Figure 3-15 in Chapter 3.

**Temperature Isosurfaces for 6'' Tubes: Configuration A (Medium Grid)**

![Temperature Isosurfaces](image)

*Figure C-1:* Temperature isosurfaces for configuration A ($N_f = 5$ inch$^{-1}$, $h = 1^\circ$). Left: $G_{\text{avg}} = 2$ kg/m$^2 \cdot $s. Center: $G_{\text{avg}} = 4$ kg/m$^2 \cdot $s. Right: $G_{\text{avg}} = 6$ kg/m$^2 \cdot $s.
Temperature Isosurfaces for 6'' Tubes: Configuration B (Medium Grid)

Figure C-2: Temperature isosurfaces for configuration B \( (N_f = 2.5 \text{ inch}^{-1}, h = 2") \). Left: \( G_{avg} = 2 \text{ kg/m}^2\cdot\text{s} \). Center: \( G_{avg} = 4 \text{ kg/m}^2\cdot\text{s} \). Right: \( G_{avg} = 6 \text{ kg/m}^2\cdot\text{s} \).
Temperature Isosurfaces for 8" Tubes: Configuration A (Medium Grid)

**Figure C-3:** Temperature isosurfaces for configuration A ($N_f = 5$ inch$^{-1}$, $h = 1$*). Left: $G_{avg} = 2$ kg/m$^2$·s. Center: $G_{avg} = 4$ kg/m$^2$·s. Right: $G_{avg} = 6$ kg/m$^2$·s.
Temperature Isosurfaces for 8" Tubes: Configuration B (Medium Grid)

Figure C-4: Temperature isosurfaces for configuration B ($N_f = 2.5 \text{ inch}^{-1}$, $h = 2^\circ$). Left: $G_{avg} = 2 \text{ kg/m}^2 \text{s}$. Center: $G_{avg} = 4 \text{ kg/m}^2 \text{s}$. Right: $G_{avg} = 6 \text{ kg/m}^2 \text{s}$.


J. van Dijk

Thesis Report

List of Acronyms

3mE Mechanical, Maritime and Materials Engineering
API American Petroleum Institute
HVAC Heating, Ventilation and Air Conditioning
ESCOA Extended Surface Corporation Of America
HTRI Heat Transfer Research, Inc.
LMTD Log-Mean Temperature Difference
NTU Number of Transfer Units
PIV Particle Image Velocimetry
CFD Computational Fluid Dynamics
FVM Finite Volume Method
FEM Finite Element Method
FDM Finite Difference Method
RANS Reynolds-Averaged Navier-Stokes
RSM Reynolds Stress Model
RNG Re-Normalization Group
LES Large Eddy Simulation
DNS Direct Numerical Simulation
SGS Sub Grid Scale
EWT Enhanced Wall Treatment
**EWF** Enhanced Wall Functions

**SIMPLE** Semi-Implicit Method for Pressure Linked Equations

**SIMPLEC** Semi-Implicit Method for Pressure Linked Equations-Consistent

### List of Symbols

#### Abbreviations

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>α</td>
<td>Tube pitch angle [$^\circ$]</td>
</tr>
<tr>
<td>$\bar{R}$</td>
<td>Universal gas constant: $\bar{R} = 8.314$ [J/K·mol]</td>
</tr>
<tr>
<td>$\Delta p$</td>
<td>Pressure drop [Pa]</td>
</tr>
<tr>
<td>$\Delta p_{\text{avg}}$</td>
<td>Average pressure drop [Pa]</td>
</tr>
<tr>
<td>$\Delta p_{\text{fin}}$</td>
<td>Fin flow pressure drop [Pa]</td>
</tr>
<tr>
<td>$\Delta p_{\text{free}}$</td>
<td>Free flow pressure drop [Pa]</td>
</tr>
<tr>
<td>$\Delta T_{\text{lm}}$</td>
<td>Logarithmic Mean Temperature Difference [$^\circ$C]</td>
</tr>
<tr>
<td>$\Delta y$</td>
<td>First layer thickness [m]</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Mass flow [kg/s]</td>
</tr>
<tr>
<td>$\dot{m}_{\text{fin}}$</td>
<td>Mass flow between fins [kg/s]</td>
</tr>
<tr>
<td>$\dot{m}_{\text{free}}$</td>
<td>Mass flow between tubes [kg/s]</td>
</tr>
<tr>
<td>$\dot{m}_{\text{tot}}$</td>
<td>Total mass flow [kg/s]</td>
</tr>
<tr>
<td>$\dot{V}_{\text{total}}$</td>
<td>Total volumetric flow rate [m$^3$/s]</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>NTU heat exchanger effectiveness [-]</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>Von Karman’s constant [-]</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity [kg/m·s]</td>
</tr>
<tr>
<td>$\Phi$</td>
<td>Mean value of general variable [-]</td>
</tr>
<tr>
<td>$\phi$</td>
<td>General variable [-]</td>
</tr>
<tr>
<td>$\phi'(t)$</td>
<td>Time-dependent value of general variable [-]</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density [kg/m$^3$]</td>
</tr>
<tr>
<td>$\rho_{\text{avg}}$</td>
<td>Average density [kg/m$^3$]</td>
</tr>
<tr>
<td>$\rho_{\text{fin}}$</td>
<td>Fin flow density [kg/m$^3$]</td>
</tr>
<tr>
<td>$\rho_{\text{free}}$</td>
<td>Free flow density [kg/m$^3$]</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>Wall shear stress [Pa]</td>
</tr>
<tr>
<td>$\tau_{xx}$, $\tau_{yy}$, $\tau_{zz}$</td>
<td>Reynolds normal stresses [Pa]</td>
</tr>
<tr>
<td>$\tau_{xy}$, $\tau_{xz}$, $\tau_{yz}$</td>
<td>Reynolds shear stresses [Pa]</td>
</tr>
<tr>
<td>$\zeta$</td>
<td>Pressure drop coefficient [-]</td>
</tr>
<tr>
<td>$\zeta_{\text{fin}}$</td>
<td>Fin flow pressure drop coefficient [-]</td>
</tr>
<tr>
<td>$\zeta_{\text{free}}$</td>
<td>Free flow pressure drop coefficient [-]</td>
</tr>
<tr>
<td>$A_{\text{bare}}$</td>
<td>Bare tube area [m$^2$]</td>
</tr>
</tbody>
</table>
\( A_{\text{flow, fin}} \) Flow area between fins \([\text{m}^2/\text{m}]\)
\( A_{\text{flow, free}} \) Flow area between tubes \([\text{m}^2/\text{m}]\)
\( A_{\text{flow, tot}} \) Total flow area \([\text{m}^2/\text{m}]\)
\( A_{\text{inlet}} \) Computational domain inlet area \([\text{m}^2]\)
\( A_{\text{xfer, eff}} \) Effective heat transfer area \([\text{m}^2]\)
\( A_{\text{xfer, fin}} \) Fin area \([\text{m}^2]\)
\( A_{\text{xfer, prim}} \) Primary area \([\text{m}^2]\)
\( B \) Constant related to the calculation of \( u^+ \) in the log-law region \([-]\)
\( b \) Variable related to the calculation of the fin efficiency \([-]\)
\( c \) Speed of sound in flowing medium \([\text{m/s}]\)
\( C_1 \ldots C_6 \) ESCOA correlation coefficients \([-]\)
\( C_f \) Skin friction coefficient \([-]\)
\( C_l \) Lift coefficient \([-]\)
\( c_p \) Specific heat capacity \([\text{J/kg} \cdot \text{K}]\)
\( C_{\text{max}} \) Maximum heat capacity \([\text{J/K}]\)
\( C_{\text{min}} \) Minimum heat capacity \([\text{J/K}]\)
\( D \) Tube diameter including the fins \([\text{m}]\)
\( d_A \) Bare tube diameter \([\text{m}]\)
\( D_{\text{hydr, fin}} \) Hydraulic diameter between fins \([\text{m}]\)
\( E \) Fin efficiency \([-]\)
\( G \) Mass flux \([\text{kg/m}^2 \cdot \text{s}]\)
\( G_{\text{avg}} \) Average mass flux \([\text{kg/m}^2 \cdot \text{s}]\)
\( G_{\text{fin}} \) Mass flux of the flow between fins \([\text{kg/m}^2 \cdot \text{s}]\)
\( G_{\text{free}} \) Mass flux of the flow bypassing the fins \([\text{kg/m}^2 \cdot \text{s}]\)
\( h \) Fin height \([\text{m, inch}]\)
\( h_c \) Convective heat transfer coefficient \([\text{W/m}^2 \cdot \text{K}]\)
\( k \) Thermal conductivity \([\text{W/m} \cdot \text{K}]\)
\( k_{\text{fin}} \) Thermal conductivity fin \([\text{W/m} \cdot \text{K}]\)
\( k_{\text{gas}} \) Thermal conductivity flue gas \([\text{W/m} \cdot \text{K}]\)
\( l' \) Characteristic length \([\text{m}]\)
\( L_{\text{flow,fin}} \) Flow path length between fins \([\text{m}]\)
\( M \) Molecular weight \([\text{kg/kmol}]\)
\( m \) Variable related to the calculation of the fin efficiency \([-]\)
\( M_{\text{gas}} \) Molecular weight flue gas \([\text{kg/kmol}]\)
\( Ma \) Mach number \([-]\)
\( N_R \) Number of tube rows \([-]\)
\( N_f \) Fin density \([\text{m}^{-1}, \text{inch}^{-1}]\)
\( Nu \) Nusselt number \([-]\)
\( p \) Pressure \([\text{Pa}]\)
\( Pr \) Prandtl number \([-]\)
Glossary

$Q_{fin}$
Total heat transferred per meter of finned tube [W]

$R$
Collective variable related to the derivation of the analytical method [kg/m²-s]

$Re$
Reynolds number [-]

$Re_y$
Turbulent Reynolds number [-]

$Re_{fin}$
Reynolds number related to the flow between fins [-]

$S$
Collective variable related to the derivation of the analytical method [kg/s]

$s_R$
Fin thickness [m]

$S_{fin}$
Fin spacing [m]

$T$
Temperature [°C]

$t_l$
Longitudinal tube pitch [m]

$t_q$
Transverse tube pitch [m]

$t_R$
Fin pitch [m]

$T_{gm}$
Mean flue gas temperature [°C]

$T_{in}$
Inlet flue gas temperature [°C]

$T_{out}$
Outlet flue gas temperature [°C]

$T_{rm}$
Mean fin temperature [°C]

$T_{tube}$
Tube temperature [°C]

$u$
Velocity component in x-direction [m/s]

$u^+$
Normalized velocity in x-direction [-]

$U_{bare}$
Heat transfer coefficient relative to the bare tube area [W/m²-K]

$U_{fin}$
Fin heat transfer coefficient [W/m²-K]

$V$
Velocity [m/s]

$v$
Velocity component in y-direction [m/s]

$V_{free}$
Free flow velocity [m/s]

$V_{in}$
Inlet velocity [m/s]

$V_{max}$
Maximum velocity in the narrowest cross-section between tubes [m/s]

$w$
Velocity component in z-direction [m/s]

$X$
Variable related to the calculation of the fin efficiency [-]

$Y$
Variable related to the calculation of the fin efficiency [-]

$y^+$
Normalized wall distance [-]

$f_{lam}$
Laminar Darcy friction factor [-]

$NTU$
Number of Transfer Units [-]