PREDICTION OF NATURAL CONVECTION HEAT TRANSFER IN A PIPELINE BUNDLE

MOHAMAD TAUFIK MOHD SALLEHUD-DIN

A thesis submitted to Imperial College London in fulfilment of the requirements for the degree of Doctor of Philosophy

Department of Chemical Engineering & Chemical Technology
Imperial College London

2012
ACKNOWLEDGEMENT

First and foremost, I am most grateful to my supervisor Professor Geoffrey F. Hewitt for his excellent support, dedication and guidance throughout my PhD. I have been very fortunate to work under his supervision, and I thank him sincerely for his advice, encouragement and patience.

Secondly, I would like to thank Professor Joe Quarini and Dr. Dan Rhys from University of Bristol and Dr. Ryo Torii from Imperial College London, for their great support in Computational Fluid Dynamics (CFD) modelling and analysis work. My special thanks to all my friends and colleagues within my research group and department, in particular Mr. Maung Maung Myo Thant @ Feroz Sultan, Khairul Rostani, Yujie Zhao and Min Lu for their continuous supports and great companionship.

I am most indebted to my family members, particularly my parents Mohd Sallehud-din Hj. Abd Manan and Paridah Omar Baki, my loving wife, Aida Azni Zulkifli and my precious children Adam Mikhael and Mia Amani; without whose continuous supports, prayers, understanding and encouragement, I could not have achieved so much.

Finally, I would also like to express my appreciation to Petroleum Nasional Berhad (PETRONAS); a Malaysian oil and gas company for giving me this valuable opportunity to undertake this research project as well as for the significant contribution in funding this work.
ABSTRACT

The work described in this thesis addresses the issue of flow assurance in hydrocarbon recovery systems. Deposition of wax and formation of hydrates are amongst the critical flow assurance challenges that need to be resolved to avoid serious reduction in hydrocarbon flows emanating from wells and passing through flow lines (which are often sub-sea) to processing facilities. The hydrocarbon streams need to be maintained at temperatures above those at which solids formation occurs. The work described in this thesis is focussed on active heating of the flow lines by arranging them in a tube bundle in which a heat source is provided by a hot fluid (typically water) flowing in a separate tube. The objective of the work was to develop a generic methodology for the prediction of such bundle systems using Computational Fluid Dynamics (CFD) to generate a heat transfer data base and to interpolate this data base using neural network methods. It was convenient to develop this methodology by focussing on a specific geometry (namely a 3 meter long bundle with 4 internal pipes - product, test, heat flow and heat return respectively). Use of this geometry allowed direct validation of the computational method since an experimental investigation of an identical geometry was carried out in a parallel research project. The CFD methodology was first used to investigate the design of the experiments; it was shown that changes were needed in rig flows and in the sensitivity of the temperature measurements to ensure that the experiments complied closely enough with the basic assumption the each of the surfaces was isothermal. Once changes were made, the experimental and computational results were directly compared and satisfactory agreement was observed. For the bundle in a horizontal orientation, a two-dimensional (2D) CFD solution was used and a data base of 1683 solutions (covering the expected range of surface temperatures) was created. A further set of 1053 three-dimensional (3D) cases were calculated for the situation where the bundle was inclined at up to 90° to the horizontal. Both sets of data were fitted by means of neural networks which allowed prediction of bundle behaviour for a wide range of conditions.
# TABLE OF CONTENTS

**NOMENCLATURE** .................................................................................................................. 10

**LIST OF FIGURES** ............................................................................................................. 15

**LISTS OF TABLES** ............................................................................................................. 20

1.0 **CHAPTER 1: GENERAL BACKGROUND AND LITERATURE REVIEW** ................. 23

1.1 Transporting Hydrocarbons in Offshore Pipelines .......................................................... 24

1.2 Project Objectives and Deliverables................................................................................. 30

1.3 Summary of Subsequent Chapters ................................................................................. 31

2.0 **CHAPTER 2: CFD MODEL DEVELOPMENT** ............................................................ 35

2.1 General Background on Flow Assurance Challenges...................................................... 36

2.2 Solid Formations in Hydrocarbon Recovery ..................................................................... 36

2.2.1 Wax .......................................................................................................................... 36

2.2.2 Asphaltenes ............................................................................................................. 41

2.2.3 Hydrates .................................................................................................................. 42

2.3 Thermal Management System .......................................................................................... 47

2.3.1 Passive Insulation .................................................................................................. 48

2.3.2 Active Heating ......................................................................................................... 55

2.4 Literature Review on Bundle Heat Transfer Analysis...................................................... 66

2.5 Previous TMF Work ....................................................................................................... 77

3.0 **CHAPTER 3: CFD MODEL DEVELOPMENT** ............................................................ 80

3.1 Overview of Computational Fluid Dynamics (CFD).......................................................... 81

3.2 CFD Model Development ............................................................................................... 86

3.2.1 The Geometry ...................................................................................................... 86

3.3 Basis of CFD Calculations for Bundle Heat Transfer...................................................... 88

3.4 Mesh Generation ............................................................................................................ 91

3.5 Boundary Conditions .................................................................................................... 95

3.6 Mesh Sensitivity Studies ............................................................................................... 101

4.0 **CHAPTER 4: CFD SIMULATIONS TO INVESTIGATE PIPE THERMAL STRATIFICATION PHENOMENON** .......................................................... 105

4.1 Introduction to the Thermal Stratification Phenomenon.................................................... 106

4.2 Background on Pipe thermal Stratification Investigations.............................................. 108

4.3 Basic Calculations ......................................................................................................... 110

4.4 Geometry Development and Mesh Generation ............................................................... 112

4.5 Boundary Conditions and Physical Models .................................................................... 113

4.6 CFD Results Analysis ................................................................................................. 116

4.6.1 Comparison of “Ideal Case” and Case with Thermal Stratification ......................... 117

4.6.2 Effect of Inclination Angles .................................................................................... 119

4.7 The Effect of Increasing Flow Rate in Minimising Thermal Stratification Problem 123
Appendix 4: The Effect of Thermodynamic Inhibitors on Hydrates Formation (Bai and Bai, 2005)) .................................................................................................................. 239
Appendix 5: Example of Projects Using Electrical Heating As Means of Managing Solids Formation and Deposition (Cochran (2003)) ................................................................. 240
Appendix 6: Thermal conductivities of Typical Soil Surrounding Pipeline For Burial Method (Bai and Bai (2005)) ........................................................................................................ 241
Appendix 7: Properties of Typical External Insulation Materials (Bai and Bai (2005)) ...... 242
Appendix 8: Properties of Typical Insulation Materials For PIP System (Bai and Bai (2005)) ......................................................................................................................... 243
Appendix 9: Physical Propertise of Pipes And Bundle Configuration ................................. 244
Appendix 10: Thermal Propertise of Gas (Nitrogen) ............................................................. 245
Appendix 11: Selected Pipe Temperatures (°C) and CFD CALCULATED HEAT Flows (W) of Horizontal Bundle ......................................................................................... 247
Appendix 12: Respective Heat Transfer Coefficients (W/m².K) and Prandtl Numbers For Horizontal Bundle ........................................................................................................ 248
Appendix 13: Values of Constant ‘a’ (Equation 6.3) for All Pipes ........................................ 249
Appendix 14: Values for constant ‘c’ and ‘d’ (Equation 6.7) for all the pipes ....................... 251
Appendix 15: Example of Typical CFX CCL File (Physics, Boundary Conditions & Simulation Set-up) ............................................................................................................. 253
Appendix 16: Enlarged Figure of The Velocity Profile For the Comparison Study Between (a) ANSYS CFX and (b) STAR-CCM+ ............................................................ 259
**NOMENCLATURE**

* ≡ Dimensionless

**Main Roman Symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Area</td>
<td>$m^2$</td>
</tr>
<tr>
<td>$C$</td>
<td>Concentration of Wax</td>
<td>%</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Specific Heat Capacity</td>
<td>J</td>
</tr>
<tr>
<td>$D_m$</td>
<td>Molecular Diffusion Constant</td>
<td>$m^2.s^{-1}$</td>
</tr>
<tr>
<td>$g$</td>
<td>Gravitational Acceleration</td>
<td>$ms^{-2}$</td>
</tr>
<tr>
<td>$K_{eq}$</td>
<td>Equivalent Thermal Conductivity</td>
<td>$Wm^{-1}K^{-1}$</td>
</tr>
<tr>
<td>$K$</td>
<td>Kinetic Energy</td>
<td>J</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent Kinetic Energy</td>
<td>$m^2s^{-2}$</td>
</tr>
<tr>
<td>$L$</td>
<td>Length</td>
<td>m</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
<td>kg</td>
</tr>
<tr>
<td>$\dot{M}$</td>
<td>Mass Flow Rate</td>
<td>$kg.s^{-1}$</td>
</tr>
<tr>
<td>$N_L$</td>
<td>Number of Internal Pipe</td>
<td>*</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$\dot{q}$</td>
<td>Heat Flux</td>
<td>$Wm^{-1}$</td>
</tr>
<tr>
<td>$Q$</td>
<td>Heat</td>
<td>W</td>
</tr>
<tr>
<td>$r$</td>
<td>Radius</td>
<td>m</td>
</tr>
<tr>
<td>$S_E$</td>
<td>Energy Source</td>
<td>$kgm^{-1}s^{-3}$</td>
</tr>
<tr>
<td>$S_M$</td>
<td>Momentum Source</td>
<td>$kgm^{-2}s^{-2}$</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
<td>s</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
<td>°C or K</td>
</tr>
<tr>
<td>$T_b$</td>
<td>Bulk Temperature</td>
<td>°C or K</td>
</tr>
<tr>
<td>$T_{ref}$</td>
<td>Reference Temperature</td>
<td>°C or K</td>
</tr>
</tbody>
</table>
$U$ Overall Heat Transfer Coefficient $\text{W.m}^{-2}\text{.K}^{-1}$

$V$ Velocity $\text{m.s}^{-1}$

$x$ Distance along pipeline $\text{m}$

**Main Greek Symbols**

$\rho$ Density $\text{kg.m}^{-3}$

$\Delta z$ Length Increment $\text{m}$

$\tau$ Stress Tensor $\text{kgm}^{-1}\text{s}^{-2}$

$\mu$ Dynamic Viscosity $\text{Pa.s}$

$\nabla$ Nabla Operator *

$h_{tot}$ Total Enthalpy $\text{m}^2\text{s}^{-2}$

$\lambda$ Thermal Conductivity $\text{Wm}^{-1}\text{K}^{-1}$

$\varepsilon$-N Strain versus Number of Fatigue Cycles Tests Curve *

$\beta$ Thermal Expansion Coefficient $\text{K}^{-1}$

$\alpha$ Heat Transfer Coefficient $\text{Wm}^{-2}\text{K}^{-1}$

$\nu$ Kinematic Viscosity $\text{m}^2\text{s}$

$\kappa$ Thermal Diffusivity $\text{m}^2\text{s}$

$\varepsilon$ Turbulent Dissipation Rate $\text{m}^2\text{s}^{-3}$

$\omega$ Angular Velocity $\text{s}^{-1}$
Relevant Dimensionless Number Symbols

\[ Re = \frac{\rho UD}{\mu} \]

\[ Ri = \frac{g \beta (T_{hot} - T_{ref}) L}{V^2} \]

\[ Nu = \frac{\alpha D}{\lambda} \]

\[ Ra = \frac{g \beta D^3 \Delta T}{\nu \kappa} \]

Acronyms

2D Two Dimensional
3D Three Dimensional
AC Alternate Current
ACS Active Circulation System
ANN Artificial Neural Network
ANSYS CFX A CFD modeling software tool
ANSYS ICEM A software for extended meshing owned by ANSYS
BHOR Bundle Hybrid Offset Riser
BPD Barrel Per Day
CAD Computer Aided Design
CFD Computational Fluid Dynamics
CD-ADAPCO A CFD-focused provider of engineering simulation software, support and services.
E&P Exploration & Production
FLUENT A CFD software owned by ANSYS
FPSO Floating Production, Storage and Offloading
IFP  French Institute of Petroleum
JIP  Joint Industry Project
JT  Joule-Thompson
LCT  Liquid Crystal Thermography
LDPUF  Low Density Polyurethane Foam
MATLAB  (Matrix Laboratory) is a numerical computing environment and fourth-generation programming language developed by MathWorks
MEG  Mono-Ethylene Glycol
OCPUF  Open-Cell Polyurethane Foam
OD  Outer Diameter
OHTC  Overall Heat Transfer Coefficient
OLGA  One dimensional software for multiphase flow dynamics simulation developed by SPT
PDE  Partial Differential Equation
PIP  Pipe-in-Pipe
PVT  Pressure, Viscosity and Temperature
PWR  Pressurized Water Nuclear Reactor
RCS  Reactor Coolant System
RMS  Root Mean Square
ROV  Remote Offshore Vehicle
SECT  Skin Effect Current Tracing
SPT  Company which owns OLGA
SST  Shear Stress Transport
STAR-CCM+  CFD software developed by CD-ADAPCO
TACITE  Pipeline transient simulator developed by IFP
TLP  Tension Leg Platform
<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TMF</td>
<td>Transient Multiphase Flow</td>
</tr>
<tr>
<td>TransAT</td>
<td>A CFD software tool developed by ASCOMP GmbH</td>
</tr>
<tr>
<td>VBA</td>
<td>Visual Basic for Applications</td>
</tr>
<tr>
<td>WAT</td>
<td>Wax Appearance Temperature</td>
</tr>
</tbody>
</table>
CHAPTER 1: INTRODUCTION

Figure 1:1: Typical wax plug (Julie, 2007) ................................................................. 25
Figure 1:2: Hydrates plug being retrieved from a pipeline (Petrobras, Brazil) .................. 26
Figure 1:3: An example of a pipeline bundle configuration for thermal heating used in Statoil’s Gullfaks, North Sea (Knudsen, 1999). ................................................................. 28
Figure 1:4: An example of pipe-in-pipe system that employs the electrical heating mechanism ........ 29

CHAPTER 2: GENERAL BACKGROUND AND LITERATURE REVIEW

Figure 2:1: Waxy crude ........................................................................................................... 37
Figure 2:2: Typical Waxy Crude Behaviour ............................................................................. 38
Figure 2:3: Wax Formation Phase Diagram ........................................................................... 39
Figure 2:4: Wax deposition .................................................................................................... 40
Figure 2:5: Asphaltenes stability vs. pressure (Bai and Bai, 2005) ........................................... 41
Figure 2:6: Hydrate Structure .................................................................................................. 42
Figure 2:7: A burning methane hydrate snowball ................................................................. 43
Figure 2:8: Gas hydrate removed from a pipeline (Petrobras, Brazil) .................................. 43
Figure 2:9: Hydrate Stability Curve ....................................................................................... 44
Figure 2:10: Hydrate Plug Formation ..................................................................................... 45
Figure 2:11: Diagram of methods that typically used in flowlines thermal management systems .... 47
Figure 2:12: Hydrate zones of insulated and non-insulated flowline at different pressure and temperature (Julie, 2007) ................................................................. 48
Figure 2:13: Conventional Insulation (Julie, 2007) ................................................................ 49
Figure 2:14: Pipe in pipe insulation (Julie, 2007) ................................................................. 49
Figure 2:15: Typical arrangement of passive insulation in pipe in pipe bundle (Julie, 2007) ........ 50
Figure 2:16: Performance of different type of insulation material (Julie, 2007) ....................... 51
Figure 2:17: U-value against burial depth for bare pipe and a 2 inch polypropylene (PPF) coated pipe .. 55
Figure 2:18: Fully insulated electrical heating system (Julie, 2007) ......................................... 57
Figure 2:19: Earthed current electrical heating system (Julie, 2007) ....................................... 57
Figure 2:20: The arrangement of “Skin Effect Current Tracing (SECT) method. (Julie, 2007) .......... 58
Figure 2:21: General arrangement of conventional and deepwater application hot water circulation heating method ........................................................................................................ 60
Figure 2:22: The velocity profile of natural convection induced by thermal gradient in a bundle from FLUENT simulation (Brown et al, 1996) ................................................................. 69
Figure 2:23: Bundle schematic used in model of Danielson and Brown (1999) ......................... 70
Figure 2:24: Temperature profiles for the Britannia bundle calculated by Danielson and Brown (1999) using an analytical model ................................................................. 71
Figure 2:25: A simplified bundle geometry validation used with TACITE (Duret et al, 2000) .......... 72
CHAPTER 3: CFD MODEL DEVELOPMENT

Figure 3:1: Work flow for computational fluid dynamic (CFD) modelling and simulations......................... 86
Figure 3:2: General arrangement of the bundle selected for the project.......................................................... 87
Figure 3:3: Schematic diagram to show the heat transfer mechanism of a pipe bundle..................................... 89
Figure 3:4: Quadrilateral and triangle elements for 2D mesh (CFD-Online)...................................................... 92
Figure 3:5: Hexahedral and tetrahedral elements for 3D mesh (CFD-Online)...................................................... 92
Figure 3:6: Mesh statistic for the hexagonal mesh of the 3D bundle model .................................................... 93
Figure 3:7: The overall volume mesh for the 3m long bundle pipe bundle ...................................................... 93
Figure 3:8: 1mm thickness cell for 2D model .................................................................................................. 94
Figure 3:9: The hexahedral mesh for the 2D model ....................................................................................... 95
Figure 3:10: Type of boundary condition that can be set based on the bounding surface (ANSYS CFX-Solver Modelling Guide)........................................................................................................ 95
Figure 3:11: Fluid boundary conditions (ANSYS CFX-Solver Modelling Guide)................................................. 96
Figure 3:12: Wall boundary conditions .......................................................................................................... 97
Figure 3:13: Overall basic settings of solver control ..................................................................................... 100
Figure 3:14: Tetrahedral and hexahedral mesh that were generated for the mesh sensitivity study................. 102

CHAPTER 4: CFD SIMULATIONS TO INVESTIGATE PIPE THERMAL STRATIFICATION PHENOMENON

Figure 4:1: The cross-sectional picture of the tetrahedral mesh for the bundle thermal stratification model (a) and hexahedral mesh for the “ideal” bundle (b). ................................................................. 112
Figure 4:2: The overall tetrahedral mesh grid for the 3m length of the thermal stratification bundle model .................................................................................................................................................. 113
Figure 4:3: Cross-sectional temperature distribution of the bundle at 1.5m plane (ideal case) .................... 117
Figure 4:4: Cross-sectional temperature profile of product pipe at 1.5m plane............................................. 119
Figure 4:5: Temperature distribution along the product pipe wall ............................................................... 119
Figure 4:6: Effect of inclination angles on average temperature of outlet, product wall and gas ............. 121
Figure 4:7: Cross-sectional temperature distribution of product pipe at 1.5m plane for horizontal (left) and vertical (right) bundle.......................................................................................................... 122
Figure 4:8: Temperature profile along the product pipe wall for horizontal (left) and vertical (right) bundle ........................................................................................................................................ 123
Figure 4:9: Graphs showing average temperature (°C) of gas, outlet and product wall against flowrate (litre/minute) ....................................................................................................................... 124
CHAPTER 5: CFD MODEL VALIDATIONS

Figure 5:1: Graph plots showing the pipe heat flows at different inclination angles for temperature Set 1 for validation of CFD predictions against experimental results ........................................... 135

Figure 5:2: Graph plots showing the pipe heat flows at different inclination angles for temperature Set 2 for validation of CFD predictions against experimental results ........................................... 136

Figure 5:3: Cross-sectional temperature profile at mid-plane for both 2D (top) and 3D (bottom) models for 3 selected cases; Case 1 (a), Case 4 (b) and Case 7 (c). .................................................. 139

Figure 5:4: Schematic diagram showing the geometry of the bundle used for the comparative study between ANSYS CFX and STAR-CCM+ ............................................................................ 142

Figure 5:5: Mesh grid for the 3D bundle model; hexagonal mesh for CFX (left) and polyhedral mesh for STAR-CCM+ (right) ............................................................................................................ 143

Figure 5:6: Comparison on cross-sectional temperature distribution between ANSYS CFX (left) and STAR-CCM+ (right) ............................................................................................................ 146

Figure 5:7: Comparison on gas velocity profile between ANSYS CFX (left) and STAR-CCM+ (right). .... 147

Figure 5:8: 3D bundle model used to study the effect of radiation in conjugate convective heat transfer ........................................................................................................................................... 149

Figure 5:9: Comparison on cross-sectional temperature (taken at middle plane) between case with radiation (left) and case with no radiation (right). ............................................................................. 151

Figure 5:10: Graphs showing the pressure and temperature profile for different insulation thickness for the Joule-Thomson cooling study using OLGA version 6 ................................................... 154

CHAPTER 6: CFD HORIZONTAL BUNDLE ANALYSIS (2D)

Figure 6:1: Two dimensional model for horizontal bundle case ............................................................................................................................ 158

Figure 6:2: Mesh statistics of the simulated case ................................................................................................................................. 159

Figure 6:3: Summary of the calculation of all the equations at each timestep ...................................................................................... 161

Figure 6:4: A screenshot showing a typical window of the solver after a simulation is completed .... 161

Figure 6:5: An extract from the CFD output file showing the domain imbalance and global imbalance for the calculation on radiation ........................................................................................................... 162
CHAPTER 7: CFD INCLINED AND VERTICAL BUNDLE ANALYSIS (3D)

Figure 7:1: Three dimensional (3D) bundle model for inclined and vertical bundle ........................................ 174
Figure 7:2: Root Mean Square (RMS) residual plot for mass and momentum equations .............................. 175
Figure 7:3: Gas temperature distribution and velocity plot of an inclined bundle .......................................... 176
Figure 7:4: Gas temperature distribution and velocity plot for vertical bundle ............................................... 177
Figure 7:5: Irregular pattern in gas temperature distribution at the end walls of the bundle ......................... 179
Figure 7:6: Cross-sectional temperature distributions at the mid-plane (1.5) of the bundle for various inclination angles for Case 1 .............................................................................................. 182
Figure 7:7: Increase in heat flow of product pipe as the inclination ............................................................... 183
Figure 7:8: Cross-sectional temperature distributions at the mid-plane (1.5) of the bundle for various inclination angles for Case 2 .............................................................................................. 185
Figure 7:9: Increase in heat flow of product pipe as the inclination angle increase for Case 2 ...................... 186
Figure 7:10: Heat flow (Watt) for sleeve pipe at various inclination angles for Case 1 (a) and Case 2 (b) ............................................................................................................................................... 187
Figure 7:11: The temperature variation of product pipe is almost at a constant value over the length and around the perimeter ........................................................................................................... 189
Figure 7:12: Temperature profile along the 3m bundle length for different angle of inclinations for Case 1 (top) and Case 2 (bottom) ........................................................................................................... 191

CHAPTER 8: ARTIFICIAL NEURAL NETWORK (ANN) PREDICTIONS OF BUNDLE HEAT TRANSFER

Figure 8:1: Plot of the Nusselt number as a function of Rayleigh number for the free convection around smooth horizontal cylinders (Thibault and Grandjean, 1990) ......................................................................................... 196
Figure 8:2: Plot of the average and the local Nusselt number prediction errors for the natural convection along slender vertical cylinders with variable surface heat flux (Thibault and Grandjean, 1990) ........................................................................................................................................ 197
Figure 8:3: Schematic diagram for a simple neuron ......................................................................................... 200
Figure 8:4: A schematic diagram showing the structure of a layer of neurons ............................................... 201
Figure 8:5: A schematic diagram showing multiple layers of neurons (MATLAB Neural Network Manual) ............................................................................................................................................... 202
Figure 8:6: Simplified three layer network (MATLAB Neural Network Manual) ........................................... 202
Figure 8:7: Plots showing some transfer functions used in ANN: hardlim, purelin, tansig and radbas ........... 203
Figure 8:8: Flow chart showing the principle of supervised learning process (Rhys, 2005) ............................ 205
Figure 8:9: A schematic diagram showing a feedforward ANN with one “hidden” layer .............................. 206
Figure 8:10: A typical screenshot of a neural network analysis running in MATLAB ................................... 209
Figure 8:11: A typical screenshot after the network training process is completed and achieved convergence .............................................................................................................................................. 210
Figure 8:12: Performance plot after the network training process is completed for the bundle heat transfer datasets. .............................................................................................................................210

Figure 8:13: Plots showing a comparison between CFD results and neural network results. ..............211

Figure 8:14: Screenshot of the neural network excel solver which can function as a heat transfer look-up table. ................................................................................................................................213

Figure 8:15: A screenshot showing neural network predictions on bundle heat transfer for a given temmperature combinations and inclination angle. .................................................................214
LISTS OF TABLES

CHAPTER 3: CFD MODEL DEVELOPMENT
Table 3.1: Comparison results between tetrahedral and hexahedral type of mesh ...................... 102
Table 3.2: Results of mesh sensitivity study for the 3D bundle model .............................................. 103
Table 3.3: Results for the mesh sensitivity study for the 2D model for the analysis of horizontal bundle pipe .................................................................................................................. 104

CHAPTER 4: CFD MODEL VALIDATIONS
Table 4.1: Comparison on CFD results between ideal case and stratified case ............................... 118
Table 4.2: CFD results of various inclination angles for the thermal stratification study .................. 120
Table 4.3: CFD results on effect of flow rates towards the average temperature of gas, outlet and product wall ......................................................................................................................... 124

CHAPTER 5: CFD SIMULATIONS TO INVESTIGATE PIPE THERMAL STRATIFICATION PHENOMENON
Table 5.1: Set 1 pipe temperatures with associated heat flows (W) at different inclination angles from experiment and CFD predictions ............................................................ 133
Table 5.2: Set 2 pipe temperatures with associated heat flows (W) at different inclination angles from experiment and CFD predictions ............................................................ 133
Table 5.3: Temperature combinations used for validation study of 2D model against 3D model ...... 138
Table 5.4: Heat flow (W) for each temperature combinations of the 3m 3D model ......................... 138
Table 5.5: Heat flow (W) for each temperature combinations of the 1mm 2D model ....................... 138
Table 5.6: Comparison of heat flows from the pipe to the interstitial gas (W) calculated by ANSYS CFX and STAR-CCM+ respectively ........................................................................... 145
Table 5.7: Comparison on heat flows (Watt) of all pipes between case with radiation and case with no radiation ................................................................................................................. 150

CHAPTER 6: CFD HORIZONTAL BUNDLE ANALYSIS (2D)
Table 6.1: Constant ‘a’ and the standard deviations of each pipe calculated by correlating the CFD data with Equation 6.4 ........................................................................................................ 169
Table 6.2: Constant ‘c’, ‘d’ and standard deviations of each pipe calculated by correlating the CFD data with Equation 6.6 ........................................................................................................ 169
Table 6.3: Nusselt numbers and Rayleigh numbers for all the pipe surfaces of each CFD run .......... 171

CHAPTER 7: CFD INCLINED AND VERTICAL BUNDLE ANALYSIS (3D)
Table 7.1: Comparison of the heat flows and gas average temperature for cases with and without radiation at bundle end walls ........................................................................................................... 180
Table 7.2: Selected temperature combinations for the study on effect of inclination angle on heat transfer in the bundle.
CHAPTER 1
Introduction

1.0 SUMMARY OF CHAPTER

Pipeline transportation of hydrocarbons from wells to processing facilities such as offshore platforms can often be very challenging since, if the product stream cools down excessively during the transportation process, this can lead to deposition of solid phases and to a reduction or, in the worst case, cessation of the flow. Thus, flow assurance (that is, the design of the system such that the flow will not be significantly reduced) is a key concern in these systems. This thesis describes computational studies of the heat transfer in a particular type of solution to the flow assurance problem, namely actively heated pipeline bundles. The objective of this first Chapter is to set the scene for this work, starting with a brief introduction (Section 1.1) on typical flow assurance challenges such as the formation of waxes, hydrates and asphaltenes. The commonly used methods to mitigate these problems are reviewed and then the focus is narrowed down to the application of actively heated bundles to improve the thermal performance of offshore pipeline systems. The importance of heat transfer analysis for a bundle pipeline system is also emphasized and this leads to the establishment (see Section 1.2) of the main objectives of the work. A description on the methodology that was implemented to achieve these objectives is also presented. Finally, a brief summary of each chapter in this thesis is given (Section 1.3).
1.1 TRANSPORTING HYDROCARBONS IN OFFSHORE PIPELINES

Conventionally, hydrocarbon products produced from sub-sea wells is transferred to processing facilities through long and slender flowlines (pipelines and risers) which can span over many kilometres. The hydrocarbon stream emerges from the reservoir through the well at typically a relatively high temperature. The cold ambient temperature of the sea water will lead to significant cooling of the hydrocarbon stream if no precautions are taken to minimise such cooling. In addition, as the hydrocarbon passing through the long flowlines and particularly in any riser section, the pressure drops can result in a significant temperature fall due to the Joule-Thompson effect, even if the heat losses are minimised\(^1\). These processes of cooling the hydrocarbon can potentially lead to the formation and deposition of solid phases such as \textbf{wax}, \textbf{asphaltenes} and \textbf{hydrates} which can cause restriction to the hydrocarbon flow and, in the worst case, total blockage to the whole flowline system. Hence, these flow assurance challenges have the potential of causing a massive loss of production.

\textbf{Wax} is not a single compound but rather a combination of several high molecular weight paraffins that solidify from crude oil due to the decrease in hydrocarbon temperature. Wax typically has a density around 800 kg/m\(^3\) and thermal conductivity of approximately 0.140 W/(m.K) (Bai and Bai, 2005). Wax substances are found naturally in most crude oils and its deposition at the pipeline wall increases as the temperature of the hydrocarbon reduces. The temperature at which crystals first begin to form is known as cloud point or wax appearance temperature (WAT). Over the time, wax deposits tend to accumulate at the pipe wall and cause reduction in the internal pipe diameter which eventually can block the pipeline as clearly shown in Figure 1.1 below. The wax deposits on the wall can also increase the surface roughness of the

\(^1\) A detailed discussion of Joule-Thompson cooling is given in Chapter 4.
wall which causes an increase in the pressure drop. This reduces the throughput of hydrocarbon for a given source (reservoir) pressure.

Wax deposition is much more prominent in deep water fields due to the high temperature gradient between the hydrocarbon that leaves the well and the ambient temperature which can be as low as 4 °C. In this case, the large heat losses to the surroundings may cause significant reduction in temperature of the production fluid.

![Figure 1:1: Typical wax plug (Julie, 2007)](image)

**Hydrates** are crystalline compounds that are formed by a combination of water and natural gases such as methane, ethane, propane, nitrogen, carbon dioxide, hydrogen sulfide etc. Hydrates are most likely to form when the hydrocarbon gas contains water at high pressure and relatively low temperature. Therefore hydrates can actually form anytime and anywhere in an offshore pipeline system where there is natural gas, water and suitable temperature and pressure (Bai and Bai, 2005). It is also important to note that hydrate blockages can develop very rapidly. The hydrate formation problem is most likely to occur during transient operations such as start-up, shutdown and blowdown. During these operations, the production system is particularly vulnerable to falls in temperature leading to the hydrate formation region being entered. The formation of hydrates is also very much dependent on the gas composition. Richer gases (those with higher propane and butane concentrations) will tend to form hydrates at relatively higher temperatures and lower pressures.
Asphaltenes in general are dark brown to black solids. Unlike waxes, asphaltenes do not melt and are not soluble in n-heptane. Chemically, asphaltenes are high molecular weight polyaromatic compounds that contain carbon, hydrogen, oxygen, nitrogen, sulphur as well as some heavy metals such as vanadium and nickel. Nearly all crude oils contain a certain amount of asphaltenes and these may cause problems in the production system when they are unstable. The stability of asphaltenes is governed by the ratio of asphaltenes to stabilising factors in the crude such as aromatics and resins. Asphaltenes can also become unstable when acids or certain types of completion fluid are added.

The detailed properties and characteristics of wax, hydrates and asphaltenes are described in Chapter 2.1. There are several methods that are typically used for managing solids deposition and formation of wax, hydrates and asphaltenes. One of the more common methods is mechanical pigging or scraping. This method is used once the deposits are already formed. Frequent pigging activities can minimise the solid build-up in the pipeline. A dual pipeline system is required in order to facilitate pigging since the pigs have to be returned for reuse. The major disadvantage of pigging is that it requires shutdown of pipeline system. This entire process could cause the loss of several days of production.
The other common method is by controlling the temperature of the hydrocarbon to ensure that it does not fall below the wax appearance temperature (WAT) or the cloud point of the production fluid. Conventionally, chemical injection of thermodynamic inhibitors such as glycol, methanol, mono-ethylene glycol (MEG) and pour point depressant are used to mitigate the formation of solid hydrates and wax. These chemicals work by shifting the hydrate formation curve to lower temperatures. Chemical injection is the most common method used for the mitigation of solid formation, mainly because such chemical injection can work for practically any hydrocarbon system. However, as the length of flowlines has increased over the years, the economic viability of chemical injection is becoming ever more questionable.

A thermal management system is defined as a technology which maintains the temperature of the hydrocarbon at sufficiently high level to prevent solids formation and deposition of wax, asphaltenes and hydrates. Nowadays, exploration and production activities are penetrating into much deeper water and at distances further away from the shore leading to the requirement for a much longer pipeline system. As mentioned previously, the use of insulation material is relatively ineffective in limiting the loss of heat from the flow line and cannot, of course, compensate for falls in temperature (particularly at the riser section of the pipeline) due to Joule-Thompson effect; these are predominantly caused by pressure changes. An alternative solution is to use an active heating method and such a system can provide a much more effective and flexible approach to the mitigation of the solid formation and deposition in the pipeline. In an active heating system, the product fluid is heated by an external heat source such as a heated fluid or electrical heating.

Typically, active heating systems using a heated fluid are applied in a bundle type of pipeline as exemplified in Figure 1.3. The heating fluid (typically hot water) passes along a tube in the bundle imparting heat to the surrounding fluid (typically gaseous nitrogen) which, in turn,
imparts heat to the product tube, maintaining it at a sufficiently high temperature to avoid the formation and deposition of solid components. The heating fluid may return along the bundle in a separate pipe (the “cold water return” pipe shown in Figure 1.3). The temperature of the heating medium tends to decrease along the entire length of the bundle. In order for it to be efficient, the heated fluid has to be injected in a large volume at a relatively very high temperature. For direct electrical heating, the heat is typically generated in the tube wall or in cables by Joule heating and is conducted through the wall to the product fluid as shown in Figure 1.4. The electrical heating system is a lot more compact and is suitable for smaller pipeline systems. It is proven that both forms of active heating system are efficient in indefinitely prolonging the cool down period, particularly during shutdowns, and also for system start up.

Figure 1.3: An example of a pipeline bundle configuration for thermal heating used in Statoil’s Gullfaks, North Sea (Knudsen, 1999).
Figure 1.4: An example of pipe-in-pipe system that employs the electrical heating mechanism

The effectiveness of a bundle thermal management system may be judged by its success in keeping the hydrocarbon product stream above critical temperatures such as the wax appearance temperature (WAT). The thermal management system may be required not only to prevent heat loss from the product but also to provide heating to offset Joule-Thompson cooling, particularly in the riser section of a flowline system. In order to analyse and design a bundle system, it is important to have sufficient knowledge of the heat transfer processes to ensure the bundle can operate effectively. Though several bundles have been implemented in oil and gas fields (for example in BP’s King field in the Gulf of Mexico), there is a dearth of detailed information on the heat transfer in the open literature. Typically, bundle thermal performance varies from one application to another depending on the bundle configuration, the fluid properties and also the reservoir characteristics. Therefore, there was a significant need to undertake an extensive heat transfer study in order to understand the highly complex thermal interaction of the pipes inside a bundle system. This need formed the main motivation of the work reported in this thesis.
1.2 PROJECT OBJECTIVES AND DELIVERABLES

The primary aim of this project is to understand and to predict the thermal interactions between the pipes inside a bundle configuration. In the study, a bundle geometry was selected which is typical of those used for the application of the active heating method for a hydrocarbon recovery pipeline system; it should be stressed, however, that the methodology developed is applicable to any bundle geometry. It was possible to validate the methodology by carrying out, in a parallel study, experimental measurements on the selected geometry. The main objective was to carry out computational simulations using a typical commercial computational fluid dynamics (CFD) code (ANSYS CFX) and to embody the results of these calculations (which covered a wide range of temperature combinations for the bundle in the horizontal, inclined and vertical orientations) into a calculation structure capable of predicting the performance of long bundle systems. The following activities were carried out in order to achieve the project objective.

- Developing a computational model of the selected bundle geometry to predict the conjugate heat transfer of natural convection and radiation that occur between the surfaces inside the bundle.
- Validating the CFD model against the data generated from experiments on the selected bundle geometry. The experimental work was carried out by Mr. Maung Maung Myo Thant in a parallel PhD project. This validation was an important step in ensuring the reliability and accuracy of the CFD simulation results.
- Undertaking numerical experiments to investigate thermal stratification effects. In the experiments, the aim was to maintain each surface inside the bundle at a nearly uniform temperature. This was done by passing water streams through the pipes and over the outer sleeve tube. Significant thermal stratification of the water in the pipes would vitiate the assumption of uniform surface temperature. The numerical (CFD) experiments
established the conditions for thermal stratification and led to a redesign of the experiments to avoid the problem.

- Carrying out a wide range of steady state Computational Fluid Dynamics (CFD) calculations on the selected bundle geometry for horizontal, inclined and vertical positions in order to generate a database on heat transfer which could be used as a basis for bundle performance calculation.

- Fitting the heat transfer database using an artificial neural network in order to establish a correlation which could be used in calculations of bundle performance.

The principal outcomes of the work described in this thesis are as follows:

- A 2D and 3D bundle model for CFD heat transfer simulations.

- A CFD methodology to undertake convective heat transfer analysis for a chosen bundle configuration.

- An extensive heat transfer database of heat flows and heat transfer coefficients for each surface in the bundle as a function of the temperature of the surface itself and the temperatures of the surrounding surfaces. This database covers inclination angles ranging from 0 to 90 degrees.

- A neural network methodology for fitting the heat transfer database in order to establish a prediction tool which can be used in bundle simulations.

1.3 SUMMARY OF SUBSEQUENT CHAPTERS

Chapter 2 of this thesis provides a more detailed review of flow assurance challenges such as wax, hydrates and asphaltenes. It also explains in more detail the common thermal management methods that are used to mitigate the problem of solids formation and deposition in hydrocarbon recovery pipeline systems. Then, the focus is narrowed down to consider the specific case of pipeline bundles using active heating. Selected case studies and previous work on bundle heat
transfer modelling are reviewed and analysed. Finally, an assessment of the technology gap existing at the time of initiation of this project is given. The outcomes of this assessment were used as the main bases of the present project and will be further explained in the subsequent chapters of this thesis.

In Chapter 3, the detailed geometry and configuration of the selected bundle is described. The bundle that was used in this work is 3m long and has 4 internal pipes which are the product, test, heat flow (heat supply) and heat return pipes respectively. These pipes are inserted into a 30 inch sleeve pipe and held in their positions by a set of spacers at both ends of the bundle. A general overview on computational fluid dynamics (CFD) is also provided in this chapter followed by a description of the methodology used for the CFD simulation of the bundle. Detailed descriptions on the mesh, boundary conditions, physical property and also the solver control for the CFD simulations are provided. Finally, the outcome of the mesh sensitivity studies that were carried out is also discussed.

As was mentioned above, a key assumption in the experimental work was that the surface temperature of all the pipes (carrier, product, test, heat flow and heat return) is fixed at a constant value around and along the length of the pipe. The presence of thermal stratification in the water streams passing through the pipes in the experimental bundle would vitiate this assumption and Chapter 4 provides details of the work that was undertaken to investigate the thermal stratification effect. Preliminary calculations to determine the Reynolds number and Richardson number of the fluid that flows inside the product pipe are also presented in this chapter. The CFD calculations on thermal stratification indicated that, at the water flow rates originally planned for the experiments, thermal stratification effects would be significant. Thus, further work was carried out to study the effect of increasing the water flow rates to minimise the thermal stratification and this work is also described in Chapter 4. It was shown that the
thermal stratification effects could be reduced to an acceptable level by increasing the flow rate. However, the change in temperature of the water stream in passing through the pipes was reduced as a result and improved temperature measurement (namely the replacement of thermocouples with resistance thermometers) was needed to achieve the required accuracy.

Chapter 5 of this thesis describes the validation activities that were carried out in collaboration with Mr. Maung Maung Myo Thant to ensure the reliability of the computational model. Using the improved experimental system (which avoided thermal stratification effects by increasing the water flow rates – see Chapter 5), experimental data were generated by Mr. Myo Thant for the bundle in the horizontal, inclined and vertical positions. Several temperature combinations were selected and the heat flows measured for each pipe were compared with the predictions of the CFD simulations; the observed agreement between experiments and predictions served to validate the simulations. Other validation exercises which were carried out and which are reported in Chapter 5 were as follows:

- Comparisons were made between a two dimensional (2D) model of the bundle and the 3D bundle model. It was confirmed that these models gave identical results for the bundle in the horizontal orientation, as would be expected.
- Most of the CFD calculations described in this thesis were carried out using the ANSYS CFX code. However, additional calculations were carried out using the STAR-CCM+ produced by CD-Adapco and these are compared with equivalent calculations done with ANSYS CFX.

Finally, in Chapter 5, work aimed at investigating the effect of Joule-Thompson cooling in pipeline risers is reported; this work was done using the OLGA simulation package.
Chapter 6 focuses on closer analysis of the outcomes from the CFD steady state simulations for the horizontal bundle. Several case studies were chosen for the analysis. A set of results from the horizontal bundle simulations were selected to be fitted with a standard type of convective heat transfer correlation.

Chapter 7 is similar with Chapter 6 which provides the detailed analysis on the CFD steady state simulations. In Chapter 7, the focus is on the inclined and vertical vertical bundle. As is shown in Chapter 7, for the inclined and vertical case, the thermal interaction inside the bundle is probably too complex to be fitted by conventional forms of correlation; this is why artificial neural networks were employed for interpolation of the data.

Chapter 8 presents the results obtained using artificial neural networks; such networks appear to be a viable and efficient method of correlating the heat transfer data generated by the CFD simulations. The Artificial Neural Network (ANN) is a mathematical or computational model which has the capability to adapt with the changes in the external and internal information which flows through the network. This advantage makes ANN suitable for training to perform complex functions. Chapter 8 starts with a general overview of artificial neural networks. Selected case studies reported in the open literature on the application of neural networks for predicting heat transfer are also reviewed and analysed. Then, the results of the application of the ANN system to the present CFD data base are presented. The results were compared against the CFD calculations to ensure the reliability and accuracy of the neural network prediction.

Finally, Chapter 9 summarises the main conclusions from the work and gives recommendations for further work.
CHAPTER 2
General Background and Literature Review

2.0 SUMMARY OF CHAPTER

In this chapter a more detailed discussion is presented on flow assurance issues related to solids formation and deposition. Section 2.2 discusses the formation and deposition mechanism for wax, asphaltenes and hydrates. Some common methods employed in the industry to mitigate these deposition problems are also described. Chemical injections such as pour point depressants and inhibitors have been used widely in Oil and Gas industry to manage these flow assurance issues. However, as the operational water depths become greater, and the length of the subsea pipelines become longer, these methods are no longer efficient and economically viable. The use of a thermal management system is an alternative solution that has the capability of providing better thermal performance and ensuring that the temperature of the produced fluid is always sufficiently above the solids formation and deposition region. In general, thermal management systems consist of passive insulation and active heating; these are discussed in general in Section 2.3. Then, in Section 2.4, the focus is narrowed down to consider the specific case of pipeline bundles using active heating; selected case studies and previous work on bundle heat transfer modelling is reviewed and analysed. An assessment of the technology gap existing at the time of initiation of this project is given. The outcomes of this assessment were used as the main bases of the present project and will be further explained in the subsequent chapters of this thesis. Finally, the work that was conducted in previous Transient Multiphase Flow (TMF) projects on bundle heat transfer is described in Section 2.5
2.1 GENERAL BACKGROUND ON FLOW ASSURANCE CHALLENGES

Flow assurance is recognised as one of the critical elements in Exploration and Production (E&P) activities. It ensures the smooth flowing of the hydrocarbon from the well to the processing facilities. These facilities can be onshore, a conventional fixed offshore platform, or floating structures like a Floating Production, Storage and Offloading (FPSO). As the pipeline network becomes longer and goes into deeper water, the pressure drop becomes greater and the temperature gradient increases dramatically. Under these conditions, solids formation and deposition may present a great challenge to the exploration and production activity. For the past 10 years, the issues associated with hydrocarbon solids formation and deposition have become much more critical and they can affect the viability and economics, particularly for deepwater projects.

One major focus in flow assurance is on prevention and mitigation of solid deposits particularly wax, asphaltenes and hydrates. For a given reservoir fluid, these solids precipitate at certain combinations of pressure and temperature (Ellison et. al., 2000). The next section on this chapter will focus on the properties and also the thermodynamic and kinetic behaviour of wax, asphaltenes and hydrates.

2.2 SOLID FORMATIONS IN HYDROCARBON RECOVERY

2.2.1 Wax

Waxes consist of a multitude of higher molecular weight paraffinic hydrocarbons which are slightly soluble in the liquid phase of black oils and condensates. The waxes in crude oils primarily consist of paraffin hydrocarbons (C18 to C36) and napthenic hydrocarbons (C30 to C60). Wax solidifies in bulk oil as discrete crystals. Wax deposits may vary from soft to hard and from yellow to black. Wax content has been found in both black oils and condensates. Waxy crude has high paraffin content, high pour point and high molecular weight.
weight. Appearance of wax may dramatically increase the viscosity of crude oils and to their displaying non-Newtonian behaviour causing the crude oil to gel. According to Ellison et. al. (2000), wax varies in consistency from that of petroleum jelly to hard wax with melting point from near room temperature to over 100 °C.

There are certain crucial parameters of waxy crudes, a knowledge of which is needed in order to identify, understand and manage wax deposition issues in a flowline.

**Wax Appearance Temperature (WAT) or Cloud Point** is the temperature at which the first wax crystals precipitate from crude oil. A pipeline needs to be operated at a temperature higher than the wax appearance temperature of the crude in order to mitigate wax issues in the pipeline.

**Pour Point** is the lowest temperature at which crude oil can be poured under gravity. A pipeline needs to be operated at a temperature higher than the pour point of the crude in order to mitigate wax issues in the pipeline.

**Yield Stress** is the force required to break down the wax structure once it has developed. It determines the pumping pressure required to restart the flow in a pipeline.
**Wax Content** is the amount of wax in a crude oil and is measured by determining the wax precipitation as a function of temperature. This parameter can be incorporated in wax deposition models, however it is not used directly in system design.

**Wax Melting Point** is the temperature that is required to melt solid wax. This temperature determines the temperature to which pipe walls may need to be heated in order to remove wax deposits and blockages.

The critical problems with wax that give the greatest challenges to the operators are wax formation and deposition. According to Bai and Bai (2005), a gel will form the crude oil when the wax precipitates from the oil. This does not occur while the oil is flowing because the intermolecular structure is destroyed by the shear forces. However when the oil stops flowing, the wax particles will interact and combine together to form a network resulting in a gel structure if enough wax is out of solution. Generally, wax crystals begin to precipitate out of the crude oil when the temperature falls below Wax Appearance Temperature.
Temperature (WAT) of the fluid. Wax molecules diffuse to the walls, followed by crystallisation on the walls resulting in deposition.

![Wax Formation Phase Diagram](image)

**Figure 2:3: Wax Formation Phase Diagram**

The deposition rate and the formation of a wax layer depends on the flow regime, temperature gradient and crude oil conditions. Wax can form at temperatures lower than 60°C (140°F). Wax deposits have been found on surfaces such as well-bores and the walls of flowlines and export lines. Wax deposition starts with the deposition of small wax particles followed by a partial restriction in the flow leading, in some cases, to a complete blockage of the pipeline. However, it is important to note that the precipitation of wax deposits is a very slow process; nevertheless, such deposition will still give devastating outcomes if allowed to continue unchecked.

Bai and Bai (2005) explained that the propensity for wax deposition and the rate at which such deposition occurs can be predicted adequately by calculating the rate of molecular diffusion of wax to the wall by following equation:
where \( m \) is the mass of deposit [kg], \( \rho \) is the density of wax [kg/m\(^3\)], \( D_m \) is the molecular diffusion constant [m\(^2\)/s], \( A \) represents the deposition area [m\(^2\)], \( C \) is the concentration of wax in percentage (%) and \( r \) is the radial position [m].

The crystallisation of wax components out of the oil is responsible for the changes in the crude oil properties including the gellation of oil and increase in viscosity. Gelling of oil will lead to high restart pressures for shut-in flowlines. Deposited wax will also increase the pressure drop in the pipeline. For a given development field, depending on the specific fluid properties and system configuration, the rate of wax deposition can be predicted. This allows an effective wax management strategy to be developed. Generally, there are several common methods that have been practiced widely for wax control and management system as follows:

- Flowline pigging
- Thermal insulation and pipeline heating
- Injection of inhibitors
2.2.2 Asphaltenes

Asphaltene is a class of compound in crude oil that is not soluble in n-heptane. Aromatic solvents such as toluene, on the other hand are good solvents for asphaltenes (Bai and Bai, 2005). In general, all oil contains a certain amount of asphaltenes and these can become a problem when they are unstable during production. Crude oils with unstable asphaltenes typically lead to operational problems related to fouling, such fouling affecting valves, chokes, filters and tubing. Asphaltenes become unstable when the pressure of the well decreases and the volume fraction of aliphatic components increases (Ellison et. al., 2000). The pressure at which asphaltenes begin to flocculate and precipitate when the aliphatic fraction of the oil reaches the threshold limit is called flocculation point.

According to Ellison et. al., (2000), depressurisation of a live bottom hole sample provides the most direct measure of asphaltene stability for production systems. During the depressurisation, the live oil flocculation point or the pressure at which asphaltenes begin to precipitate in the system is determined by monitoring the transmittance of an infrared laser beam which passes through the sample. Onset of flocculation will produce a
recognisable reduction of light transmittance. If an oil has a flocculation point, then the asphaltenes are unstable at pressures between flocculation point to just below the bubble point. After the bubble point has been reached, the mass of precipitated asphaltenes and the mass of asphaltenes deposited in the cell is measured. Ellisonn et al (2000) also mentioned that these two measurements give a means to assess the the likelihood of problems due to deposition. The mass of deposited asphaltenes can also be used to predict the mass of deposition in a production system. Though this figure is highly likely to be an overestimate, it is still gives a good indication for design purposes and for contingency planning.

2.2.3 Hydrates

Hydrates are ice-like crystals that consist of water and light hydrocarbons or gas molecules - about 85 mol% water and 15 mol% hydrocarbon. The water molecule ‘traps’ the gas molecule like a ‘cage’ as illustrated in Figure 2.6 below.

![Figure 2.6: Hydrate Structure](image)

Hydrates contain as much as 180 volumes of gas per volume of hydrate. Hydrates can form naturally or as a result of man-made conditions. Hydrates form at temperatures lower than 18°C when the pressure is less than 17 bar. Hydrates may occur naturally in
sediments of the sea-floor, deep lakes and the subsurface of the Arctic permafrost regions. Methane hydrates look remarkably like ice but burns if lit (Figure 2.7).

Figure 2.7: A burning methane hydrate snowball

Hydrates can also form in wellbores, flowlines, valves and meter discharges. Hydrates have been found in wet gases, condensates and black oils. The formation of hydrates is a significant flow assurance problem and is mostly encountered during restart. Hydrates can cause pipeline blockage and, in the worst cases, can cause the pipeline to burst.

Figure 2.8: Gas hydrate removed from a pipeline (Petrobras, Brazil)
Hydrate forms from water and gas molecules and such formation is encouraged by low temperatures and high pressures (see Figure 2.9).

Figure 2.9: Hydrate Stability Curve

Figure 2.10 below shows the mechanism of hydrate plug formation. Hydrate plugs start with the formation of hydrate particles or nucleation. This is followed by a hydrate film growth. The film will then break, and the particles will start to agglomerate together forming larger chunks of solid hydrate. These chunks will then start to plug and block the pipeline.
Based on the above survey, it can be concluded that the formation and deposition of solids such as wax, asphaltenes and hydrates is the major flow assurance issue faced by the oil and gas industry. Various methods have been used to mitigate this issue, the most common methods being as follows (Bai and Bai, 2005):

- Thermodynamic control – the aim is to keep the pressure and temperature of the whole system out of the solids formation regions.
- Kinetic control – to control the conditions of solids formation so that deposits do not form.
- Mechanical control – this method allows solids to deposit but they are periodically removed by pigging.
Injection of thermodynamic inhibitors such as, glycol, methanol and mono-ethylene glycol (MEG) is one of the common methods used. Typically these compounds are used for restarts and for spot treatments during shutdown or displacement. Ethanol has been used in Brazil because it is less expensive (Cochran, 2003). Generally, all the developments have the capability to inject inhibitors into the subsea system. However, flowline (riser and pipeline) systems are getting much longer as the water depth increases or the reservoir/well is located further away from the shore. Thus, more inhibitors are required to mitigate the problem of solids formation and deposition in the pipeline system and this can significantly affect the economic viability of a project. For example, methanol is typically used for system restarts and also to treat and/or displace fluids in wellbores, trees, jumpers and manifolds during shutdowns. However, methanol is not commonly used for continuous operation of oil systems. If the system produces a significant amount of water, the associated cost can be exorbitant. According to Cochran (2003), given that a well produces 1000 BPD of water, 0.5 BBL methanol per BBL of water is needed to prevent hydrate formation and methanol cost approximately USD 100 per BBL, then the methanol costs will be USD 50,000 per day or USD 18,000,000 per year.

Insulation is another example of solids control strategies commonly implemented. Insulation is always applied to pipelines and risers. Insulation provides hydrates control by maintaining temperatures above those for hydrate formation. Insulation also provides a cooldown time period, which is also known as the “no touch time”, before the production fluid reaches the hydrate formation temperature. During a cold start, insulation helps to reduce the time to warm the produced fluids to temperatures above those for hydrate and wax formation. However as the flowline systems become longer, an insulation system is no longer efficient in providing good thermal management. This is particularly so in some riser sections where the fluid can cool without a change of enthalpy due to the Joule-
Thomson cooling effect. These are the circumstances in which active heating systems are becoming significant in providing much better performance for a flowline thermal management system. Hot fluid circulation is a commonly used technique to warm deepwater flowlines and risers particularly in a cold restart. Either depressurisation or displacement is commonly used following shutdowns to protect flowlines and risers.

2.3 THERMAL MANAGEMENT SYSTEM

A thermal management system is a method of controlling the temperature of a hydrocarbon which flows inside a flowline by means of heat containment and heat transfer. Over the last 10 years flowline design and the operation of thermal management systems have become increasingly important for preventing blockages in flowline system.

In general there are two types of thermal management system, one involves thermal insulation (passive mechanism) and the other one uses thermal heating (active mechanism). Each mechanism can be divided into a number of sub-categories as shown in Figure 2.11.

![Thermal Management System Diagram](image)

Figure 2:11: Diagram of methods that typically used in flowlines thermal management systems
2.3.1 Passive Insulation

Passive insulation is a conventional method that has been used widely for flowlines (horizontal pipelines and manifolds and vertical risers) in oil and gas fields to safeguard the flowline from corrosion and to minimise heat loss to the surroundings. Passive insulation also helps to prolong the cool down period of the pipeline flowline system during shutdown and restart activities.

The traditional polypropylene foam insulation was developed by Norsk Hydro for subsea insulation systems in the mid eighties (Hansen et al, 1999). After going through various and extensive tests and prototype fabrications, the polypropylene foam based insulation was introduced in 1989 and since then it has been used widely in the Gulf of Mexico and North Sea as a thermal insulation system (Hansen et al, 1999).

Good insulation can often keep the temperature of the fluid high enough to prevent the precipitation of wax and the formation of hydrates as shown in Figure 2.12.

![Flowline Insulation for Hydrate Avoidance](image)

Figure 2.12: Hydrate zones of insulated and non-insulated flowline at different pressure and temperature (Julie, 2007)
Conventionally there are 2 types of insulation, namely wet and dry insulation. The materials used for wet insulation are typically polyurethane, polypropylene, rubber or glass reinforced plastic. These materials have heat transfer coefficients (U-values) of approximately 2 W/m²K (Jaeyoung, 2002). The dry insulation systems use polyurethane foam and rockwool which have better U-value of approximately 1 W/m²K (Jaeyoung, 2002), therefore giving less heat loss to the surroundings and, in many cases, allowing the temperature of the hydrocarbon to be kept sufficiently high to mitigate the solids deposition problem. However, the presence of water in the insulation does cause a deterioration in its performance. This has led to the introduction of the pipe-in-pipe (PIP) system as shown in Figure 2.14. By partially evacuating the gap between the two pipes, reductions in the U-value to approximately 0.5 W/m²K may be achieved (Jaeyoung, 2002).

Figure 2:13: Conventional Insulation (Julie, 2007)

Figure 2:14: Pipe in pipe insulation (Julie, 2007)
As the water depth increases, the mechanical load on thermal insulation becomes more significant. Hence, a thicker insulation is required to ensure the durability and integrity of the flowline system because the environment is becoming more aggressive. Currently, a wide range of new thermal resistance materials and insulation techniques are being developed to improve the thermal performance of flowline systems.

Besides increasing the thickness of the insulation or using higher resistance insulation material, the thermal performance of the flowlines can be improved effectively by using an insulated pipe in pipe bundle system as shown in Figure 2.14. With this system, the insulated pipe in pipe system is enclosed in a carrier pipe which is filled with inert gas. In this bundle arrangement system the thermal performance is greatly improved with a lot less heat loss due to conduction and convection to the surroundings. Figure 2.16 compares the thermal performance of different types of material which are typically used for flowline insulation systems. The figure clearly shows that low density polyurethane foam (LDPUF), which is a type of dry insulation, has a heat transfer coefficient value at least half of the U-value of wet insulation material; use of this material will give significant improvement in heat insulation performance.

Figure 2:15: Typical arrangement of passive insulation in pipe in pipe bundle (Julie, 2007)
It’s vital to have low U-value and high thermal resistance material as the insulation for flowlines in order to reduce the thickness of the insulation. Thicker insulation will increase the overall size of the flowlines and will significantly increase the imposed mechanical load; this may drastically influence the economic viability of the project.

![Figure 2:16: Performance of different type of insulation material (Julie, 2007)](image)

Delafkaran and Demetriou (1997) describe work carried out by Kvaerner Oil and Gas Limited for a recent North Sea field development. This was for a high pressure, high temperature gas condensate field where the produced fluids are exported to an existing platform via an insulated multiphase pipeline for onward transmission. Thermal insulation was required to mitigate against problems of wax deposition and hydrate formation as the field aged. A pipe-in-pipe system was used with the annulus space between the pipeline and the outer sleeve filled with alumina silicate microspheres. Delafkaran and Demetriou discuss a wealth of practical details relating to the construction including stress analysis, hydrostatic testing, fatigue analysis, riser end closure, riser anchor systems etc. The focus
of the work described in this thesis is on the thermal aspects but it should be stressed that mechanical aspects are equally significant and must be firmly considered at the design stage.

Welch et al (1997) describe experience with the Texaco Erskine Multiphase Pipeline which was installed to transport a gas/condensate mixture from the Erskine wellhead platform to the Lomond platform at a maximum temperature of 150°C. The insulation material used was alumina silicate microspheres. The paper stresses the importance of field joint design. In this application, the field joint is insulated and the outer sleeve pipes are then joined by means of a sliding steel collar which is welded into place. Extensive use was made of finite element and fracture mechanics analysis and this was backed up by extensive experimental testing.

Beckmann et al (1998) describe the design and installation of the Troika towed-bundle flowline system. Troika is a deep water (2700 ft) oil development located in the Gulf of Mexico approximately 150 miles south of New Orleans, Louisiana. The combined flow from five wells is being produced to the Bullwinkle platform through two 14-mile long, 10.75 inch diameter pipe-in-pipe insulated flowlines. These flowlines were installed by the bottom-tow method in four 7-mile long segments. Each 7-mile segment was fabricated at a beach makeup site, laterally launched, bottom towed for 400 miles, positioned in the field and connected. Connection to the Bullwinkle platform entailed lifting the riser end to the surface and securing it to the jacket leg in a catenary configuration. Insulated steel pipe jumpers were used to join the 7-mile sections at the mid and sub-sea manifold end points.

The thermal insulation of the flowline was 3-inch thick open cell foam and the casing pipe was 24 inches in diameter. The insulation on the riser is 2-inch thick. The pipe-in-pipe
system was pressurised with nitrogen but adequate account was not taken of the effect of the nitrogen on the effective thermal conductivity of the insulating material. Beckmann et al give a wealth of detail about the fabrication of the bundles at the beach makeup site including bulkhead assemblies, welding, hydrotests etc. The towing and positioning of the bundles is also described and the installation by connection to the platform and manifold is outlined. The authors reached the following conclusions:

- Simplifying interfaces between platform, riser, flow segments, jumpers and the manifold was critical to maintaining the schedule on a project requiring concurrent design, procurement, fabrication and installation.
- The use of bulkheads along the bundles was prudent.
- Steel catenary risers can be installed on the leg of a platform using anchor handling vessels. Assist vessels are needed for contingency (power loss) and unexpected events (currents). Practice proved helpful.
- Completed steel catenary risers with strakes attached can be towed minimising offshore spread costs.

The in-place flowline alignment between end points should be as straight as possible to minimise length inaccuracies and facilitate towing adjacent bundles into position. Target areas for positioning the bundles in the field should be made as large as possible to accommodate the bundle length survey tolerances and connection methods. The use of ROV sonar for bundle placement within the target box worked well but required a considerable amount of time.

Bundle tow-loads were about 50% of what the designer expected. The authors hypothesised that this may be the result of the smooth abrasion coating achieved through the application process. This implied that longer or heavier bundles might be achievable.
Knudsen (1999) describes the technology used in tying back several satellite fields (Gullfaks South, Rimfaks and Gullveig) to the Gullfaks A-platform. The transport of the fluids presented notable challenges, particularly that from Gullfaks South. Hydrate control is handled by applying thermal insulation, by using methanol as a hydrate inhibitor and depressurising the flowlines. For the Gullfaks South field, these measures were insufficient and heated tube bundles were needed. Installation of the reeled flowlines proceeded satisfactorily with the exception of a minor accident with an 8 inch line to the Rimfaks satellite. This suffered buckling, which meant that the line had to be cut and flanges installed. The authors give details of the challenges in design of these systems, which, again, were mechanical as well as thermal in nature.

Pipelines are buried typically in deepwater subsea fields to provide thermal insulation, on-bottom stability and also physical protection from dropped objects and trawling. They are generally buried below the mudline level or have rocks dumped over them. The pipelines are often buried by trenching and backfilling. Pipeline burial is a project specific selection and very much depending on the pressure, viscosity and temperature (PVT) characteristic, soil properties, topography, geo-hazards and infrastructures.

The heat capacity of buried insulated pipelines can be relatively higher than a pipe-in-pipe (PIP) system and this can prolong the cool down time period. The installation cost of insulated and buried pipeline is approximately 35-50 % that of a PIP system (Bai et al, 2005). For a long pipeline system this can be a massive saving to the whole project. Several other advantages of pipelines burial system include:
Chapter 2: GENERAL BACKGROUND AND LITERATURE REVIEW

- More choices on vendor for pipe fabrication
- More choices on installation contractors & vessels.
- Improved schedule and reduced time for first oil
- Slower cool down time during shut down compared to PIP system
- Possibility for single pipe repair in comparison to PIP

Since soil has a high heat capacity, it will absorb the heat from the hydrocarbon flowing in a pipeline buried in it. It will retain the heat by acting as a natural heat storage system and is capable of retaining sufficient heat to keep the hydrocarbon warm after the flow has stopped. During a shut down period, the heat stored in the soil can keep the hydrocarbon warm enough to prevent wax gelling and hydrate formation for as much as 4 times longer than for a PIP.

![Figure 2:17: U-value against burial depth for bare pipe and a 2 inch polypropylene (PPF) coated pipe](image)

2.3.2 Active Heating

Active heating is a practical, effective and economical solution for flowline thermal management systems, especially in deep water fields where pipeline and riser lengths are
such as to span large temperature and pressure differences. By using active heating, the use of flowline pigging to manage and control wax deposition can be avoided (Khlefa et al, 2003). Active heating may be used to prevent the cooling down of a non flowing system (for example during shutdowns) and to maintain or increase temperature for a flowing system and thus to reduce chemical injection requirements. Active heating also is capable of efficiently maintaining the temperature of hydrocarbon above the wax and hydrates formation temperatures.

Typically a flowline active heating system is utilised for conditions such as (Julie, 2007):

- Warming up of non-flowing system
- Prevention of cool down of non-flowing system
- Maintaining temperature while flowing
- Increasing temperature while flowing
- Reduce chemical injection requirement

There are 2 types of active heating methods; one is electrical heating and the other one is circulation of hot fluid (Fouad et al, 2004). Active heating systems using electrical heating have been used for example in the Statoil Asgard field in the North Sea and in the DeepStar field in the Gulf of Mexico. Active heating utilising hot fluid circulation is being implemented at Statoil’s Gullfakes and ConocoPhillips’ Britannia fields (both in the North Sea) and BP’s King field in the Gulf of Mexico (Fouad et al, 2004). Another example of bundle riser application is the Bundle Hybrid Offset Riser (BHOR) concept in the West of Africa. The bundle arrangement incorporates multiple conduits conceived to enhance its flow assurance performance. Operational performance is further enhanced by the Active
Circulation System (ACS) to provide hydrate blockage removal capability following accidental or prolonged shutdowns.

**Direct Electrical Heating**

For electrical heating systems the heat is supplied by an electrical cable. In general there are two types of electrical heating methods; one is a fully insulated system which requires complete electrical insulation of the flowlines from sea water as shown in Figure 2.18. The second method is an earthed current system which requires electrical communication with the sea water through anodes or other means as shown in Figure 2.19 (Hansen et al, 1999 and Julie, 2007).

![Figure 2:18: Fully insulated electrical heating system (Julie, 2007)](image)

Another technique of electrical heating, as shown in Figure 2.20, is “Skin Effect Current Tracing (SECT)” which has an external heating tube containing an insulated electrical cable which supplies an AC current to heat up the hydrocarbon.

![Figure 2:19: Earthed current electrical heating system (Julie, 2007)](image)
In principle, heat is generated on the inner surface of a ferromagnetic tube that is thermally coupled to the pipe to be heat traced. An electrically insulated and thermal resistant conductor is installed inside the heat tube and connected at the end of the tube. The heat tube and conductor is connected in series to an AC voltage source. This method of heating known as skin effect heating because the return path of the circuit current is pulled to the inner surface of the tube by skin effect and proximity effect between the heat tube and the conductor.

![Diagram of Skin Effect Current Tracing (SECT) method](image)

**Figure 2:20:** The arrangement of “Skin Effect Current Tracing (SECT) method. (Julie, 2007)

**Hot Fluid Circulation**

The hot fluid circulation method is a proven technique of heating flowlines which has been used in deepwater projects. The hydrocarbon is heated by means of transfer of heat from hot water or another suitable fluid which circulates in a relatively small heating pipe within the bundle. The heated medium is heated in a heater at or adjacent to the subsea wellhead or by a heater in the above-surface processing facility. It general, a simple hot water circulation system is a long fixed tube heat exchanger with a heating medium on the outer side and the production fluid on the inner side of the tube similar to a typical pipe-in-
Chapter 2: GENERAL BACKGROUND AND LITERATURE REVIEW

pipe bundle arrangement. Several water-based heating media have been used in this heating system.

**Treated Sea Water (Khlefa et al, 2003)**

Sea water is an ideal medium as it is readily available and can be filtered and treated easily during the construction. However, it is considered as a high risk media because it is susceptible to corrosion attack, scale development and corrosion resulting from formation of sulfate-reducing bacteria.

**Treated Fresh Water (Khlefa et al, 2003)**

Fresh water is a conventional heating media in boiler operation. Hence, it is a well proven material that has been used for many years in heating systems. Typically, fresh water is used together with inhibitors, pH control and biocide treatments to control corrosion. In a closed loop heating and cooling system the primary concerns are microbiological and pitting corrosion. The current practice in mitigating these problems is by employing one of the following chemicals:

- Nitrite / Molybdate
- Sulfites + dispersants
- Borates and phosphates
- Molybdates + oxidizer (Synergistic agents)

With the use of any of these chemicals together with fresh water as the heating media, it is vital to control the chemistry of the water, oxygen content, pH and inhibitor concentration. This is to ensure the prevention of rapid corrosion and severe pitting.
Treated Glycol/Water Mixture (Khlefa et al, 2003)

Glycol / water mixtures have been used widely as heating or cooling media particularly in the applications where freeze protection is required. A heating system with glycol typically requires the use of an inhibitor to control corrosion. However, glycol inherently reduces corrosion and acts as a biocide when the concentration of glycol exceeds 20%.

Figure 2:21: General arrangement of conventional and deepwater application hot water circulation heating method

Brown et al (1999) described the design and implementation of a subsea heated bundle for Britannia project. This UK North Sea project, brought the first, 15 km subsea, heated bundle into operation in the 3rd quarter of 1998. A hot water heated bundle concept was used as the best technical solution to prevent hydrate and wax formation in the 15 km subsea tie back and to assist the environmental impact reduction programs. The subsea piping system is in a bundle configuration where a 37 inch diameter carrier pipe encloses a 14 inch production pipe, an 8 inch production test pipe, a 12 inch heated water pipe and a 3 inch methanol supply pipeline. The carrier wall and the 14 inch production pipeline are separated by less than 40 mm at the 6 o’clock position. The bundle configuration provides the best technical solution for the 140-meter deep subsea tie back. However, unlike typical bundles, 8300 m³ of water media actively heats the bundled piping as the water circulates between the carrier annulus, the 12 inch hot water pipeline, and the platform at
13,200 m$^3$/day. This circulation of water distributes heat in the subsea system and significantly increases the thermal mass of the system to improve the thermal control required for the subsea tie back. The topside equipment provides heating, expansion tanks, pumps, and system monitoring.

Zhang et al. (2002) discuss the issues in the design of hot water heated production flowline bundles using two typical offshore West Africa developments as case studies. Two design options, direct and indirect heating, are analyzed and compared. The indirect heating system consisted of a small hot water pipe attached to the production flowline, and the direct heating consisted of a large hot water pipe encasing the production flowline. Factors affecting the warm-up time, such as water flowrate and temperature, pressure limitations, and topsides heating capabilities are analyzed and compared.

In the short-distance tieback case studied by Zhang et al. (2002), the subsea wells, connected to a subsea manifold, are tied back with dual 12 inch flowlines to an FPSO 6.6 km away. The ambient temperature at the seabed is 4°C. The flowlines are efficiently insulated, with an overall heat transfer coefficient (OHTC) of 1.1 W/m$^2$.K, which provides the required heat conservation during normal production. An actively heated flowline bundle using foamed-in-place open-cell polyurethane foam (OCPUF) with nitrogen pressure was developed. In the long-distance tieback case, the total riser and flowline length is 32 km one-way and 64 km round-trip. All other conditions are the same as the short-distance tieback.

Direct and indirect hot water heated bundle designs have been analyzed and compared for typical West Africa subsea developments with different tieback lengths. The author
concludes that both heating options can meet the design heating conditions, while each has its advantages and disadvantages.

Direct heating has more efficient heat exchange between the heating water and the heated fluids than indirect heating. Also the flow area in the 12 inch and 16 inch designs is larger than the 3 inch indirect heating pipe. This allows the use of a higher flowrate, so the heating distance and warm-up time is generally better than indirect heating. However the topside heating capability may limit the flowrate or water temperature so the actual heating times may be only 50% or more than those predicted based on constant flowrate and water temperature. With either heating configuration, the warm-up time increases with tieback distance at a rate slightly faster than proportional. For tieback distance of 16-32 km, the warm-up time for the indirect tieback increases much faster than proportional with distance. Warm-up of the 8 km offset subsea tieback can be completed within 24 hours with a reasonable flowrate and water temperature. Warm-up time for the 32 km subsea tieback is about 100 – 150 hours even with direct heating.

For the long distance tieback with 32 km offset, the hot water pipe size of 3 inch may be too small and thus high pressure is required at the necessary flowrate. It is expected that increasing from 3 inch to 3.5 inch pipe size would reduce the pressure significantly and make indirect heating possible for the long distance tieback.

The indirect heating bundle has the advantage of smaller bundle size and lower cost. To alleviate the flowrate limit due to high pressure drop, the pipe size can be increased much more easily than with direct heating.

Another example of of a production bundle installation in West Africa is the Girassol field development; this is offshore from Angola in 1,400m deep water. This installation has
stringent operating requirements due to flow assurance issues and the need for flexibility. Rouillon (2002) details 11 production bundles with eight installed during the first phase of the offshore operations carried out during 2001. Each production bundle contains:

- Two 8 inch production flowlines.
- One 2 inch service line.

These lines are contained within two semi cylindrical modules of syntactic buoyancy foam, which ensure the thermal insulation of the pipes. A 30 inch carrier pipe houses the insulation foam and the production and service lines.

The carrier pipe ensures the mechanical and corrosion protection, and contributes to the overall thermal performance of the system as it avoids exchanges of seawater with the external environment. The carrier pipe is entirely closed and full of inhibited water. The length of the production bundles varies from 700m to 3,000m.

Esaklul et al (2003) describe the flow assurance challenges faced during the development of a multi-well field located approximately 27 km (17 miles) from the host facility and how those challenges have been solved using pipe-in-pipe active heating. The King field is located approximately 96 km (60 miles) East of Venice, Louisiana and is made up of four blocks: Mississippi Canyon MC 84, 85, 128 and 129. The water depth in the field ranges from 1500 to 1650 m (5000 to 5400 ft). The development consists of a subsea production system tied back to the Marlin TLP. Marlin is located in Viosca Knoll Block 915 at a water depth of 990 m (3250 feet). The subsea wells are tied back to the Marlin TLP using two flowlines. Each flowline connects to Marlin with a pipe-in-pipe steel catenary riser. Flow assurance was one of the key factors in the development of the field with hydrate and wax deposition as the primary design parameters.
Several development options were considered including insulation, chemical mitigation and active heating. Active heating was selected as the preferable option for its cost effectiveness, avoidance of pigging for wax control, and ability to maintain the flowlines above the hydrate formation and wax deposition temperatures during shut downs and prior to start-ups after prolonged shut downs. The King heated flowline system consists of two production flowlines, each an 203 mm (8inch) nominal diameter flowline inside a 305 mm (12 inch) nominal diameter jacket pipe. The 203 x 305 mm (8 x 12 inch) pipe-in-pipe (PIP) flowlines extend approximately 27 KM (17 miles) from the King field (water depth = 1646 m, 5400ft.) to the Marlin TLP (water depth = 990 m, 3250 ft.). The heating medium (a 30 wt% aqueous ethylene glycol solution with various inhibitors) is circulated from the topsides down the annulus of one flowline to the dual well tie-in base (also referred to as the manifold) located at the extreme far-end of the flowline system.

The authors conclude that active heating is a practical and effective flow assurance method for development of deepwater fields, providing hydrate and wax control during production and short-term shutdowns. In addition, it provides an easy means to condition the line after prolonged shut downs and an effective way to remove hydrate or wax plugs.

Based on the above survey, active heating seems to be the most practical, economical and viable solution in managing various flow assurance issues. Such systems have high operational flexibility and high efficiency, especially for the development of deepwater fields in comparison to conventional insulation methods.

As mentioned previously, active heating in general consists of hot fluid circulation method and direct electrical heating. The main advantage of water is that the technologies for heating water are very robust and basic. Also, it is very easy to store hot water in a tank. In
case of unexpected general power shut down, when using a hot water tank as a buffer, a minimum energy is needed to maintain the water circulating ensuring some additional time to fix the problem. The main disadvantage is that it is relatively voluminous. The active heating layer will increase by around 1 to 2 inches the outer diameter of the pipe compared to a standard pipe for similar thermal performance in passive mode. In case of a contingency plan with a hot water tank, the volumes of water may be substantial. For example for an 8 inch internal diameter, 1500 m long with an equivalent U value of 4W/m²K (based on the ID) the engineering study concluded that it was necessary to inject around 6m³/h of water at 60°C to maintain the fluid above 30°C in case of shut down. However, on an FPSO, hot water may be needed for many other functions and a reliable supply for active heating may need to be guaranteed by setting up an independent system using domestic boiler principles for example.

In comparison, the electrical heating is much more compact. A system equipped with the heat tracing using armour wires will have the same OD as a conventional insulated flexible pipe. The limitations of the armour concept will be on the maximum heat power available. For example, for an 8 inch pipe, replacing 12 % of the armour wires by heat tracing wires allow the addition of 200 W/m for a 1900 m long flexible pipe with a three-phase supply voltage of 1000V. By using an assembly with cables, the heating capability could be increased greatly. As an example, a single cable with three 10 mm² conductors would be able to provide 60 W/m on a length of 3000 m for a three-phase supply with a voltage of 1000V. This would open up a large range of possibilities.

It is not possible to state in general that either the electrical or the fluid active systems is the better. Both approaches have their advantages and disadvantages and the optimal choice between them will depend on the particular application. For the work described in
Chapter 2: GENERAL BACKGROUND AND LITERATURE REVIEW

In this thesis, the focus was on the bundle active heating system which uses hot fluid circulation to warm up the production pipe in order to keep the temperature above the solids formation and deposition region.

2.4 LITERATURE REVIEW ON BUNDLE HEAT TRANSFER ANALYSIS

Detailed information on the thermal analysis of a bundle active heating system is very limited in the open literature. As stated previously, the main focus of the present work has been on actively heated tube bundles. A model for such bundle has been developed by Brown et al (1996) and this model was subsequently incorporated in the OLGA system code. The work focused on numerical modelling to predict the transient heat transfer behaviour in a bundle system.

The overall model consists of two parts. The first part is OLGA multiphase flow simulator and the second part is a transient heat transfer model for the bundle which accounts for the axial heat convection and heat transfer between the pipes inside the bundle. The bundle model is coupled with OLGA through heat fluxes and temperatures. Generally OLGA performs all the hydraulic calculations and the heat transfer calculations out to the outermost surface of the internal pipes of the production fluids. The bundle model handles all the heat transfer calculations between the internal pipes and the annular fluid and between the annular fluid to the surrounding seawater. There is one equation for each of the internal lines including the carrier. For modelling purposes, in general there are three types of line; the first type is the carrier, a large pipe containing all the other lines and a fluid in the annular region. The second type is an internal line which is modelled as carrying a single phase flow with constant fluid properties. Examples of the fluids carried by this type of line are methanol and the heating medium supply or return lines. The third type is an internal, multiphase line which modelled by OLGA and is exemplified by the production and test lines.
The transient energy balance for the internal, single phase line is given by:

$$A_i \rho_i C_{pi} \left( \frac{\partial T_i}{\partial t} + V_i \frac{\partial T_i}{\partial z} \right) = -2\pi U_i r_i (T_i - T_c)$$  \hspace{1cm} \text{Equation 2.2}$$

Where $A_i$ is the cross section area of the pipeline, $\rho_i$ is the density of the fluid in it, $C_{pi}$ is the specific heat capacity of the fluid, $V_i$ represents the velocity and $T_i$ is its temperature. $U_i$ is the overall heat transfer coefficient (based on the internal tube radius $r_i$) between the single phase pipeline and the carrier line and $T_c$ is the temperature of the carrier line. The energy balance for the carrier line is given by:

$$A_c \rho_c C_{pc} \left( \frac{\partial T_c}{\partial t} + V_c \frac{\partial T_c}{\partial z} \right) = -2\pi U_c r_c (T_c - T_a) - \sum_{i=1}^{N_L} 2\pi U_i r_i (T_i - T_c)$$  \hspace{1cm} \text{Equation 2.3}$$

where the subscript $c$ refers to the carrier line parameters, and $T_a$ is the ambient temperature. $U_i$ is an overall heat transfer coefficient for heat transfer between the carrier tube fluid and the seawater and $N_L$ is the number of internal pipes. The sum in Equation 2.3 is over all the internal lines and accounts for the heat transfer between the carrier fluid and each of the internal lines. For the multiphase production and test lines, $T_i$ is the temperature of the outermost wall layer. In contrast to the case of the single phase lines, where $T_i$ is the fluid temperature.

Equation 2.2 and Equation 2.3 above ignore the thermal mass of the pipe walls for the carrier and single phase lines; this is relevant only for transient simulations and was accounted for separately. The procedure used in the OLGA code is to solve the above equations numerically. At each time step, OLGA passes the temperature of the outermost wall layer of the inner tubes to the bundle model and these temperatures are used in Equation 2.3. Then the bundle model will solve for the temperatures throughout the bundle, excluding the external pipes. When the
temperatures have been calculated, the bundle model calculates the heat that has been transferred at each line section to each of the internal lines using the following equation:

\[ Q_i = -2\pi r_i \Delta z_j (T_i - T_c) \]  

Equation 2.4

where \( \Delta z \) is the length increment. This increment of heat transfer \( Q_i \) is the used to calculate the change in enthalpy and hence the temperature of the fluid in the production and test lines at the next length step.

In order to solve the equations, initial and boundary conditions must be defined. Normally, a line will be specified to be at a constant temperature initially. For warm-up cases, this temperature will be ambient temperature. The fluids basically can flow in either direction and lines (for instance a heating and return line) can be connected at the end of the computational domain. Brown et. al. cited a number of analytical solutions for limiting cases and these were used to check the performance of the numerical scheme.

In their report, Brown et al also stress the importance of heat transfer coefficient. For flowing single phase fluids, the heat transfer coefficient can be calculated from conventional methods for single phase flows. There are certain conditions where the bundle is operating without any medium that flows inside the space between the internal lines and the carrier pipe. Hence the flow is fundamentally driven by natural convection and the heat transfer is relatively more complex. Brown et al also mentioned that, under these circumstances where data on natural convection in enclosed spaces in the literature is limited, the use of computational fluid dynamics (CFD) codes to predict the heat transfer seems more viable. The author reported the use of FLUENT for the computational prediction work.
Danielson and Brown (1999) emphasized that a correct prediction of overall heat transfer can have a significant influence on the both temperature gradients and lowest temperature attained in the produced fluid (relevant to hydrates formation). However, the film heat transfer coefficients, both in the carrier annulus and within the multiphase production and test lines, are the most poorly-characterised system parameters. The author also mentioned that even in a clean systems, standard correlations such as Dittus-Boelter equation cannot be expected to be within $\pm 30\%$. 

Further more, there could be fouling of the system over time that could have a profound and unaccounted-for, effect on the thermal resistance.

Danielson and Brown (1999) also emphasized the influence of the grid size on the steady state temperature profile. The equations for the temperature profiles in flowing bundle systems can be quite stiff and if the numerical grid is not sufficiently fine in regions of large axial temperature gradients, the predicted temperature profile can be significantly in error. This can result in a significant, non-conservative over-prediction of minimum line temperatures (and possible formation of hydrates and paraffin).
Danielson and Brown (1999) reported the development of an analytical model which has no grid dependence and capable of predicting the correct temperature profile regardless of the stiffness of the system. In addition to that, due to its speed and ease of use, the model is ideal for parametric sensitivity studies on variables that are not known with great accuracy, such as the film heat transfer coefficient. Danielson and Brown (1999) provided an example of a bundle system as given in Figure below.

Figure 2.23: Bundle schematic used in model of Danielson and Brown (1999)

Heat enters the system at $z$ and exits at $z+\delta z$; additional heat is lost to conduction through the wall. A heat balance from the point $z$ to $z+\delta z$ yields,

\[
\dot{M}_p c_{p,p} T_p (z) = \dot{M}_p c_{p,p} T_p (z + \delta z) + U_p \pi D_p \delta T_p (T_p - T_c)
\]

Equation 2.5

\[
\dot{M}_t c_{p,t} T_t (z) = \dot{M}_t c_{p,t} T_t (z + \delta z) + U_t \pi D_t \delta T_t (T_t - T_c)
\]

Equation 2.6

\[
\dot{M}_r c_{p,r} T_r (z) = \dot{M}_r c_{p,r} T_r (z + \delta z) + U_r \pi D_r \delta T_r (T_r - T_c)
\]

Equation 2.7

\[
\dot{M}_c c_{p,c} T_c (z) = \dot{M}_c c_{p,c} T_c (z + \delta z) + U_p \pi D_p \delta T_p (T_p - T_c) + U_t \pi D_t \delta T_t (T_t - T_c) + U_r \pi D_r \delta T_r (T_r - T_c) + U_c \pi D_c \delta T_c (T_c - T_d)
\]

Equation 2.8
where $M_i$ is the mass flow rate of flow of line $i$, $c_{p,i}$ is the heat capacity for the fluid in line $i$, $T_i(z)$ is the temperature of the fluid in line $i$, $D_i$ is the diameter of line and $U_i$ represents the overall heat transfer coefficient from pipe $i$. The subscripts are as follows:

$i = p$ (production), $t$ (test), $r$ (heating medium return/supply), $c$ (carrier) and $a$ (ambient).

Equations 2.5-2.8 are a set of linear, coupled ordinary differential equations which can be solved analytically as detailed out in the paper by Danielson and Brown (1999). An important part of the solution is to specify boundary conditions which are the inlet production and test line temperatures and setting the carrier and return/supply line temperatures equal at the two ends of the pipe. The solution by Danielson and Brown (1999) is particularly convenient one for evaluation of the influence of various parameters. It reduces to a solution for the co-axial system when two of the lines are moved. Typical results obtained by the model used by Danielson and Brown (1999) are presented in Figure 2.24 below.

![Figure 2.24: Temperature profiles for the Britannia bundle calculated by Danielson and Brown (1999) using an analytical model.](image-url)
Duret et al (2000) attempted to take into account a bundle model in a transient multiphase flow model using TACITE. The TACITE compositional code is able to simulate the behaviour of transient multiphase flow for the design and the control of oil and gas production pipelines and wells. TACITE can also be used for simulating process equipment such as controllers, valves, pigs, separators, lateral injectors and bundles. A transient pipeline bundle model has been developed within TACITE to analyse the thermal interactions between several single phase lines and the main production line. The model is based on the numerical solution of a set of energy balance equations, defined for each line and annulus medium within the bundle, the carrier pipeline and the outside. Natural convection is taken into account in the annulus and the thermal inertia is taken into account only for the production line. Thermal calculations performed with TACITE bundle model have been compared against results produced from SYSTUS software (a finite element code with mechanical and thermal modules) which has been validated by a test as part of a deep water project in West Africa; the results were very satisfactory.

Figure 2:25: A simplified bundle geometry validation used with TACITE (Duret et al, 2000)

Chin et al (2000) conducted a study using the OLGA code to determine the steady state performance of each flowline in the bundle, including the multiphase product line and two single
Chapter 2: GENERAL BACKGROUND AND LITERATURE REVIEW

phase heating lines. The pipeline used for the analysis is an insulated bundle system installed in Gulf of Mexico. The heat transfer between different lines in a bundle is simulated with the bundle module in OLGA. The multiphase flowline is analysed in the OLGA code and the injection and return lines are treated as auxiliary lines in which the flows are single phase. Heat transfer coefficients between fluids are required for the calculation of heat transfer rate among the different flowlines. The overall heat transfer coefficients includes the thermal resistance values from the wall and insulations. The internal film heat transfer coefficients for the injection line and return line are calculated with a standard single phase correlations. The internal film coefficients for the multiphase flow lines are calculated within OLGA are based on the flow regimes, fluid properties and fluid mass flowrates.

The “bundle” model in OLGA is a convection model which was originally designed for the presence of the carrier fluid instead of insulation around the flowlines. It was found that the bundle module has difficulty in modelling the multiple pipelines encased by an insulation. Because of this, the original bundle model needed to be modified to reflect the physical process. To handle such configuration, a “virtual carrier’ is generated by separating the bundle into two parts. The virtual carrier has the same outer diameter as the actual casing and has a thickness equal to the width of the nitrogen annulus plus the thickness of the casing wall and has an equivalent thermal conductivity, Keq. The resultant equivalent cross section of the model is shown in Figure 2.26 below.
The numerical results from the steady state simulation carried out by Chin et al were compared against the actual production data and showed good accuracy. Results for temperature profiles of the pipelines inside the multiple flowline bundle demonstrate the importance of thermal interaction among these pipelines. Finally, Chin et al concluded by emphasizing the significance of steady state simulation in providing a useful tool to predict operating conditions under active heating.

Zhang et al (2002) carried out comparative studies between direct and indirect hot water heated bundle systems as shown in Figure 2.27 below. Two typical offshore West Africa field developments were chosen as case studies. The aim was to determine the transient heat transfer between the heating water and the heated fluids taking account of the complex insulation configurations. Factors affecting the warm up time such as flowrate and temperature, pressure limitations and topsides heating capabilities were analysed and compared. A thermal analysis method was employed using an in-house computer program. The implemented thermal analysis method represents the heat transfer process among different parts of a bundle with good accuracy; the model used a pseudo-steady state lumped thermal mass approach and neglects the heat transfer in the axial direction of the flowline.
In the pseudo-state approach, the heat transfer rate among the pipes at a particular instant during the warm up process is assumed to be the same as the steady state value and the temperature of the pipes are assumed to be held at the average temperatures at the instant of the transient process. This approach ignores the heat-storing capacity (thermal mass) of the insulation. A limitation of the approach is, therefore, that it can only be applied with good accuracy to systems with low insulation thermal mass compared with the pipes containing the water (or other heating fluid) and the production fluid. To use this method, the heat transfer coefficients are required. These can be determined by using either analytical formula and empirical correlations for the direct heating or by using CFD methods. In this context, Zhang et al used CFDDesign, a finite element analysis based software.

Based on the analysis, Zhang et al concluded that both heating options can meet the design heating conditions, while each has its own advantages and disadvantages. The analysis also indicated that direct heating has more efficient heat exchange between the heating water and the heated fluid than does indirect heating, though using indirect heating gives less loss of heat to the environment. Indirect heating gives comparable warm-up times to those obtained with direct heating for short to medium distance tiebacks. Finally Zhang et al emphasized the advantage of
having smaller bundle size and lower cost for the indirect heating. In order to alleviate the flowrate limit on indirect heating due to high pressure drop, the pipe size can be increased.

Based on the above literature survey, it will be seen that several numerical modelling studies have been carried out to analyse the heat transfer behaviour inside a pipeline bundle. However the detailed analysis data and also the methodology used are still limited in the open literature particularly in the area of natural convection heat transfer for a bundle. It is also important to note that most of the models are case specific and focussed on conduction. According to Danielson and Brown (1999), there is major difficulty in predicting natural convection driven heat transfer particularly for a bundle system. To put this into perspective, it should be noted that even standard correlations for in-tube heat transfer such as the Dittus-Boelter equation cannot be expected to be within ± 30% even in a clean system.

Almost all of the numerical models in the open literature fail to mention the impact of radiation on the heat transfer behaviour in a conjugate natural convection condition. According to Sharma et al (2006), given the enclosure is sufficiently large, the natural convection is turbulent in nature. Natural convection in enclosures is often coupled with surface radiation heat exchange among the walls of the enclosures in transparent fluid media such as air. Surface radiation basically will modify the wall temperature distribution which in turn affects the natural convection bringing in interaction effects. Such conjugate heat transfer conditions are very complicated and there is a dearth of correlations to address this situation. These factors provided the motivation for the present work. There was a need to investigate and predict the complex thermal interactions between pipes in a bundle in a buoyancy driven flow with transparent fluid media in which the influence of radiative heat transfer could be significant.
2.5 PREVIOUS TMF WORK

Some work has been successfully carried out by Imperial College London in collaboration with University of Bristol under The Transient Multiphase Fluid (TMF) Joint-Industry-Project (JIP). The focus of previous work was mainly on horizontal flow lines. The work started in TMF2 by using a 1 meter long tube bundle and using water to fill up the gap between the internal pipes. The tube bundle consisted of two simulated production pipes and one simulated heating pipe mounted in a carrier tube with the inter-surface space filled with water as shown in Figure 2.28. Some CFD calculations were also carried out as can be observed from Figure 2.29 below.

Figure 2:28: Bundle arrangement in TMF2

Figure 2:29: CFD results for water filled bundle (Liu et al, 2006)
The work was continued in TMF3 to cover a wider range of applications such as more complex bundle configurations, manifolds and partially buried pipelines. In TMF3, a 3 meter long tube bundle was constructed and air was used as the interstitial gas medium between the internal pipes.

Under the TMF3 project, Imperial College London was responsible for the experimental work and the University of Bristol focussed on the computational fluid dynamics (CFD) calculations and also the development of 1-D spreadsheet solver to predict the temperature profile over the length of the flow line system. Neural networks were used to fit the large set of data generated from the CFD work.

For the current research project, the work was focused on inclined and vertical bundle systems, though the work on horizontal bundles has also been extended. A similar bundle configuration to that used in the TMF3 work was used as the basis for developing the methodology for both experimental and CFD work. However, it is important to note that the methodology developed
for the selected bundle case is also valid for other bundle configurations which makes it applicable for any other type of pipeline bundle applications.
CHAPTER 3
CFD Model Development

3.0 SUMMARY OF CHAPTER

In this chapter, work on the development of the bundle model for the computational fluid dynamics (CFD) is described. The chapter begins with an overview of computational fluid dynamics and the governing equations and methodology used are described (Section 3.1). There then follow descriptions of the development of a model for the geometry of selected bundle configuration (Section 3.2) and the bases of the calculation methods that were used to calculate the heat transfer for the bundle (Section 3.3). Section 3.4 describes the mesh generation for both the 2D and the 3D bundle models; the 2D model was used for the calculations on the horizontal bundle position whereas 3D model was used for the calculations for the inclined and vertical bundle positions. To complete the framework for the CFD analysis, the boundary conditions and the convergence criteria have to be defined and these matters are discussed in Section 3.5. An essential part of CFD calculations is to establish the number of grid points used is sufficient to render the results insensitive to further increases in this number; Section 3.6 describes the grid sensitivity studies that were carried out for the 2D and 3D bundle models respectively.
3.1 OVERVIEW OF COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics (CFD) is a computational simulation tool that is used to solve, for a specified system, problems related to fluid flow, heat transfer and other physical processes. In general, CFD works by solving the governing laws of fluid dynamics numerically over a chosen domain of interest with specified boundary conditions. Nowadays, with the help of increasingly powerful computing capability, CFD modelling and analysis has become a much more attractive and efficient design and analysis tool, particularly for the engineering industry. CFD can also provide an alternative to scale model testing or experimental work; CFD simulation can be less time consuming, more cost-effective and often more reliable than conducting physical test or experiments.

**Governing Equations**

The fundamental basis of fluid dynamics is the Navier-Stoke equations which are derived from Newton’s Second Law of conservation of momentum. These equations are supplemented by equations for conservation mass (also known as continuity equations) and conversation of energy. These basic principles are illustrated in the following sketch:

<table>
<thead>
<tr>
<th>Physical Principles</th>
<th>Mathematical Equations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass is conserved</td>
<td>Continuity equation</td>
</tr>
<tr>
<td>Newton’s Second Law</td>
<td>Momentum equation</td>
</tr>
<tr>
<td>Energy is conserved</td>
<td>Energy equation</td>
</tr>
</tbody>
</table>

In reference to, both the continuity and momentum equation are given by Equations 3.1 and 3.2 in the form used in the CFX code (“ANSYS CFX-Solver Theory Guide”):
\begin{equation}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0
\end{equation}
Equation 3.1

\begin{equation}
\frac{\partial (\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla p + \nabla \cdot \mathbf{\tau} + S_M
\end{equation}
Equation 3.2

Where the stress tensor, \( \mathbf{\tau} \), is related to the strain rate given by Equation 3.3, \( \rho \) is the density, \( \mathbf{U} \) is the vector of velocity \( U_x, U_y, U_z \) in x, y and z-direction respectively, \( p \) is pressure, \( S_M \) is the momentum source, \( T \) is the temperature, \( \mu \) is the dynamic viscosity and the nabla operator, \( \nabla \), defined by Equation 3.4.

\begin{equation}
\mathbf{\tau} = \mu \left( \nabla \mathbf{U} + (\nabla \mathbf{U})^T - \frac{2}{3} \delta \nabla \cdot \mathbf{U} \right)
\end{equation}
Equation 3.3

\begin{equation}
\nabla = i \frac{\partial}{\partial x} + j \frac{\partial}{\partial y} + k \frac{\partial}{\partial z}
\end{equation}
Equation 3.4

where \( i, j \) and \( k \) are unit vector in \( x, y \) and \( z \) directions respectively.

The total energy equation used in CFX is as follows:

\begin{equation}
\frac{\partial (\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{U} h_{tot}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (\mathbf{U} \cdot \mathbf{\tau}) + \mathbf{U} \cdot S_M + S_E
\end{equation}
Equation 3.5

Where \( h_{tot} \) is the total enthalpy, related to the static enthalpy \( h(T, p) \) given by Equation 3.6 below, \( \lambda \) is the thermal conductivity and \( S_E \) is the energy source.

\begin{equation}
h_{tot} = h + \frac{1}{2} \mathbf{U}^2
\end{equation}
Equation 3.6
The term $\nabla \cdot (\rho \tau)$ represents the work due to viscous stresses and is called the viscous work term. The term $U S_M$ represents the work due to external momentum sources and is currently neglected. An alternative form of energy equation (thermal energy equation) which is suitable for low-speed flows is also available in ANSYS CFX. To derive it, an equation for kinetic energy $K$ is required.

$$K = \frac{1}{2} U^2 \quad \text{where } U^2 = U_x^2 + U_y^2 + U_z^2$$  \hspace{1cm} \text{Equation 3.7}

The kinetic energy equation is derived by taking the dot product of $U$ with the momentum equation as given by Equation 3.8 below.

$$\frac{\partial (\rho K)}{\partial t} + \nabla \cdot (\rho UK) = -\rho \nabla p + U \cdot (\nabla \tau) + U S_M$$  \hspace{1cm} \text{Equation 3.8}

**CFD Methodology in General**

Before carrying out a CFD modeling and analysis, it is vital to thoroughly understand the problem that is going to be simulated in order to accurately define it in CFD. It is also important to note that in CFD it is not always necessary to simulate the whole body or domain for a particular problem, sometimes it is sufficient to analyse it based on a specific area of interest. By doing this, it can significantly reduce the computational time without compromising the reliability of the results. Generally, in order to carry out a CFD simulation, there are 4 working steps that have to be followed.

1. **Creating Geometry and Generating Mesh**

This activity is part of the work that needs to be done at the pre-processing stage. The first activity is to create a closed geometric solid for the specific area of interest. Most of the CFD
software has its own design modeler; however this modeling stage can also be done with other typical computer-aided design (CAD) or other geometry/mesh creation tools. During the model design stage, it is necessary to specify all the regions involved in the model such as solid, fluid and interface. The final aim of this stage is to generate the mesh for the body for input to the physics. Typically meshing is done automatically based on the input given by the user.

Flow variables are calculated more accurately with a fine mesh compared to a coarse one. However it is important to note that the finer the mesh, the more computation time is required to complete the CFD simulation. Generally, a mesh sensitivity study is often carried out before the number of nodes (fineness of the mesh) is finalized for a particular case. During the mesh sensitivity study, the fineness of the mesh is refined until the simulation result is no longer influenced by any further mesh refinement.

2. Defining the Physics of the Model

At this stage, the mesh is being uploaded in the pre-processor. Physical models, fluid properties and boundary conditions that are required by the solver are all specified. It is important for the user to have as much information as possible on the problem in order to accurately define the required parameters.

3. Obtaining the Numerical Solution

After all the information required for the simulation has been specified, the user has to set the solver control which determines the stopping or convergence criteria of the simulation. Then, the CFD software will perform the numerical calculation iteratively until the convergence criteria are met.
4. Analysing and Visualising the CFD Result in Post-Processor

Once the solutions are being obtained, the user can analyse the results to make sure that they are satisfactorily and meeting the criteria as expected. It is important for the user to have as much information or knowledge as possible on the problem in order to make a good judgment on whether the results calculated by the CFD code are reliable. If the result is not satisfactorily, the user has to identify the possible source of the error such as poor quality of mesh, or the fluid and/or physical models.

Use of CFD for Bundle Heat Transfer Project

CFD is one of the important elements in this project. Generally, the CFD work is carried out to complement the experimental work which has a limitation on the number of possible temperature combinations of all the pipes inside the bundle that can be run during the project duration. This is simply because an experiment with a given combination of temperatures takes much longer than an equivalent CFD calculation. Generally speaking, the experimental work in this project is aimed at validating the CFD methodology; once validation is achieved, the CFD methodology can be used for a wide range of cases. Thus, CFD is used here to simulate many temperature combinations so as to ensure the reliability of the heat transfer database for interpolation in solving bundle flow line problems. Once validated, the CFD methodology can also be used to cover a much wider range of inclination angles than that covered in the experiments, again because of the lower cost and higher speed of such calculations relative to the cost of experimental testing. However it is important that model used in the CFD work is validated against the experimental results in order to ensure the accuracy of the heat transfer database. The validation activity that was carried out for this work is described in Chapter 5 of this thesis.
Chapter 3: CFD MODEL DEVELOPMENT

**CFD Workflow for Bundle Heat Transfer**

The flow diagram below summarises the steps that were carried out to develop the geometry for the computational fluid dynamic (CFD) calculations, to generate the mesh and also to define the CFD models required for the simulation.

![Flow diagram for CFD modelling and simulations](image)

**Figure 3.1: Work flow for computational fluid dynamic (CFD) modelling and simulations**

### 3.2 CFD MODEL DEVELOPMENT

A computational model of the whole 3m long bundle was developed and the related mesh generated in order to conduct the CFD simulations. The aim was to calculate the heat transfer parameters (heat transfer coefficients and heat flows) used in predicting the overall heat transfer behaviour of the bundle.

#### 3.2.1 The Geometry

The chosen bundle used as the basis for the present work is a large diameter (“carrier”) pipe which carries 4 smaller diameter internal pipes. In the field, nitrogen gas is used to fill up the gap between the pipes inside the bundle. In the experimental validation studies, air is used instead of nitrogen for convenience. The 4 internal pipes are the product pipe which carries the product fluid, the test pipe, the heat flow pipe that carries the heating
medium fluid and the heat return pipe which transports the heating medium back to the source.

The geometry is shown in Figure 3.2 and is based on the following dimensions:

- **Sleeve** – 762mm (internal diameter)
- **Product** – 355.6mm (outside diameter)
- **Test** – 203.2mm (outside diameter)
- **Heat Flow** – 101.6mm (outside diameter)
- **Heat Return** – 101.6mm (outside diameter)

![Figure 3:2: General arrangement of the bundle selected for the project](image-url)
3.3 BASIS OF CFD CALCULATIONS FOR BUNDLE HEAT TRANSFER

A bundle is constructed in such a way that it can reduce excessive temperature changes along the pipeline that carries hydrocarbon from the well to collection point or processing facilities. This can be sometimes be achieved by incorporating conventional passive insulation methods within the bundle which increases the thermal resistance of the pipe wall to limit the heat flow from hydrocarbon to the cold surrounding sea water.

Alternatively, a hot fluid or electrical source is supplied in a pipe inside the bundle to increase the temperature in the bundle. This active heating method is a proven mechanism that can efficiently maintain the temperature of the hydrocarbon at a sufficiently high value to prevent solid formation and deposition from occurring inside the pipe.

The heat transfer rate to or from a given pipe surface in a bundle is dependent not only on the local temperature of the tube but also on the local surface temperature of all other tubes which are in contact with the interstitial gas. Natural convection and thermal radiation are the main heat transfer mechanisms between the internal pipe surfaces through the interstitial gas. Such heat transfer is highly complex and involves solving for the full range of surface temperatures which are likely to be encountered.

It is anticipated that the heat transfer from the in-pipe fluid to the pipe wall as well as the conduction through the pipe wall is relatively straightforward and the use of an approximate heat transfer coefficient would normally suffice. It is also important to know the in-tube fluid temperature. From this temperature, the skin temperature of the pipe can be calculated and will be used as the initial basis for this CFD work. A fixed surface wall temperature is assumed for each of the internal pipes in order to calculate the heat rate to or from the respective pipe surfaces. In a steady state, the net heat flux of all the interrelated pipes (product, heat flow, heat
return, test and sleeve) should equal to zero and the amount of net heat being absorbed by the sleeve pipe is approximately equal to the amount of heat lost to the sea water.

To illustrate the principle of the methodology, it is helpful to consider a simplified system as shown in Figure 3.3; this consists of a large carrier pipe which contains pipes carrying the hydrocarbon (the product pipe) and the heating fluid (the heating pipe)

![Figure 3:3: Schematic diagram to show the heat transfer mechanism of a pipe bundle](image)

The energy balance equations which already taken into account both convective and radiative heat transfer for the pipe bundle in Figure 3.3 can be written as follows:

\[
A_i \rho_i C_p_i \frac{dT_i}{dt} + A_i \rho_i V_i C_p_i \frac{dT_i}{dx} = \sum Q = -2\pi i U_i (T_i - T_c)
\]

Equation 3.9

There is one equation for each of the internal pipes in the bundle. Subscript \(i\) is the pipe identifier, for examples, \(p\) for product, \(t\) for test, \(f\) for heat flow and \(r\) for heat return. The
velocity (V) is positive if the flow is in the same direction with the production fluid. Otherwise it will be negative. The energy balance equation for the carrier pipe is given as follows:

\[
A_c \rho_c C_p \frac{dT_c}{dt} + A_c \rho_c V_c C_p \frac{dT_c}{dx} = \sum Q = -2\pi r_i U_c (T_c - T_a) - \sum_{i=1}^{N_L} 2\pi r_i U_i (T_c - T_i)
\]

Equation 3.10

where

- \( A = \) Cross-sectional area \((m^2)\)
- \( \rho = \) Density \((kg/m^3)\)
- \( C_p = \) Specific heat capacity \((J/Kg.K)\)
- \( V = \) Fluid velocity \((m/s)\)
- \( T = \) Temperature \((^\circ C)\)
- \( Q = \) Heat \((W)\)
- \( U = \) Overall heat transfer coefficient \((W/m^2.\cdot ^\circ C)\)
- \( t = \) time \((s)\)
- \( x = \) Distance along pipeline \((m)\)
- \( N_L = \) Total number of pipeline in the bundle
- \( r = \) Internal radius of pipeline \((m)\)

Subscripts

- \( i = \) Pipe identifier within the bundle
- \( c = \) denotes the carrier line
- \( a = \) Total number of pipelines in the bundle
3.4 MESH GENERATION

The governing equations for the fluid flow and heat transfer are in the form of partial differential equations (PDE) to represent the conservation of laws for mass, momentum and energy. These PDEs are usually transformed into algebraic equations that can be recognised and solved by digital computers. Typically the fluid domain is split into smaller sub-domains (made of geometrical shape elements such as tetrahedra and hexahedra for 3D meshes and quadrilaterals and triangles in 2D). The the governing equations are discretised and solved inside each of these sub-domains. This process of discretisation of the PDEs is normally described as “numerical discretisation”.

Mesh Classifications

The sub-domains normally known as cells or elements and a collection of cells or elements are called the mesh or grid, whereas the process of developing the mesh is called mesh or grid generation. In general, meshes are classified into one of two categories which are respectively the connectivity-based and the element-based types of mesh.

Connectivity Based Meshes

The most basic forms of connectivity-based meshes are structured meshes and unstructured meshes respectively. Generally, in structured meshes the node connectivities are regular, in order words, they have a fixed pattern. This restricts the element choice typically to quadrilaterals in 2D and hexahedra in 3D. On the other hand, an unstructured mesh has irregular node connectivities which allow the solver to use any possible element that is available. However, the storage requirements for unstructured meshes are relatively larger in comparison with structured meshes because the neighborhood connectivities need to be stored explicitly.
Element Based

Meshes can also be differentiated by their geometric dimensions and shapes. Depending on the type of analysis as well as the solver requirements, the generated mesh could be in 2D or 3D. Typically, quadrilaterals and triangles elements are used for 2D meshes (Figure 3.4), whereas for 3D meshes, hexahedral and tetrahedral types of elements are used (Figure 3.5).

![Figure 3.4: Quadrilateral and triangle elements for 2D mesh (CFD-Online)](image)

![Figure 3.5: Hexahedral and tetrahedral elements for 3D mesh (CFD-Online)](image)

Mesh Generation For 5-Pipe Bundle

ANSYS ICEM was used to construct the grid for the selected bundle model. A hexagonal type of mesh is chosen for both 2D and 3D bundle model for its efficiency, robustness and fast meshing capability.

3D Mesh For Inclined and Vertical Bundle

The 3D mesh is shown in Figures 3.6 and 3.7 below. The mesh is sufficient to provide grid insensitive numerical results in an acceptable computation time. The 3m long 3D model has
327,525 nodes and 313,168 elements as shown in Figure 3.6. The 3m length of the pipe is divided into 50 equal length grids.

Figure 3:6: Mesh statistic for the hexagonal mesh of the 3D bundle model

Figure 3:7: The overall volume mesh for the 3m long bundle pipe bundle
Chapter 3: CFD MODEL DEVELOPMENT

2D Mesh For Horizontal Bundle

2D model was developed for the simulation of horizontal bundle pipe. In practical, CFX can not really model in 2 dimensions, therefore in order to carry out the simulations, a very thin slice of the pipe cross-section is extruded into 3 dimensions where the thickness is relatively small length scale used for the mesh. 1mm thickness was chosen as this gives good cell aspect ratio for the solution.

Hexahedral type of mesh was also selected. A fine mesh with prismatic layers was constructed with 51,764 nodes for the 2D bundle model. For the 2D model, the computation time required to achieve convergence was much shorter than that for the 3D model for the inclined and vertical bundles.
3.5 **BOUNDARY CONDITIONS**

**General Overview on Boundary Conditions**

According to ANSYS CFX-Solver Modelling Guide, boundary conditions are a set of properties or conditions on the surfaces of a domain that are required to fully define the flow simulation. The figure below gives a representation of the different types of boundary conditions required; these depend on the nature of the bounding surfaces.

![Figure 3:10: Type of boundary condition that can be set based on the bounding surface (ANSYS CFX-Solver Modelling Guide)](image)

Figure 3:9: The hexahedral mesh for the 2D model
In general, boundary conditions are split into two categories which are fluid boundaries and solid boundaries.

**Fluid Boundary**

![Fluid Boundary Diagram](image)

Figure 3:11: Fluid boundary conditions (ANSYS CFX-Solver Modelling Guide)

Based on ANSYS CFX-Solver Modelling Guide, a fluid boundary is defined as an external surface of a fluid domain and consists of different boundary conditions as shown in Figure 3.11 above. The types of boundaries are as follows:

- **Inlet** – Fluid predominantly flows into the domain
- **Outlet** – Fluid predominantly flows out the domain
- **Opening** – Fluid can simultaneously flow in and out the domain. However, this type of boundary condition will not work for domains with more than one fluid present.
- **Wall** – Impenetrable boundary to fluid flow
- **Symmetry plane** – A plane of both geometry and flow symmetry
Solid Boundary

A solid boundary is an external surface of a solid domain which consists of the following boundary conditions:

- Wall – Impenetrable boundary to fluid flow
- Symmetry – A plane of both geometry and flow symmetry

Boundary Conditions For The Bundle Model

1 domain with 5 boundary conditions has been specified for the bundle model. A gas domain filled with nitrogen is used with no slip wall type of boundary condition (zero velocity at wall) for the outer surface of all 4 internal pipes (product, test, heat flow and heat return) and also for the inner surface of the sleeve pipe. An image of the screen used in specifying these boundary conditions is shown in Figure 3.12

Figure 3:12: Wall boundary conditions
**Wall Boundary Conditions**

All 5 fluid boundary walls were modelled as ‘no-slip’ boundaries. For the 3D calculations, adiabatic walls were specified for each end of the 3 m zone calculated. It has been assumed that the 3D zone to be calculated is limited to 3 m since spacers are positioned at every 3m to hold the internal pipes in position and it is assumed that these spacers block the flow of the interstitial gas that fills up the space between the pipes. Though the spacers are not totally impermeable to the interstitial gas, it is believed that dividing up the domain in this fashion is a reasonable approximation and avoids the complications which would arise if the details of the spacers and the flows between adjacent zones were taken into account.

**Pipe Temperatures**

In the model, 5 wall surfaces have been specified; namely the product, test, heat flow, heat return and sleeve. These walls were modelled as smooth walls with an emissivity of 0.9. The emissivity value was chosen at 0.9 based on the sensitivity studies done in previous work by Liu et. al. (2006). A fixed wall temperature is used for each of the surfaces. It is important to choose the temperature range covered so as to be relevant to typical wax and hydrates formation regions The following temperature (°C) ranges were used in the simulations of inclined and vertical bundles (3D model):

- **Product**: 20-40
- **Heat Flow**: 40-60
- **Heat Return**: 35-55
- **Test**: 20-40
- **Sleeve**: 5-15
For the 2D model, since the computation time to complete 1 simulation is relatively small than that for the 3D model, a wider temperature range (0 to 100 °C) was chosen for each of the surfaces.

**Fluid and Turbulence Models**

The interstitial nitrogen was modelled as a constant property gas at atmospheric pressure respectively. A Boussinesq buoyancy model was chosen (Further description is given in Appendix 1). The thermal interaction in the bundle is a natural convection driven heat transfer. Therefore, even with a small change in the temperature of the gas, there is a significant increase in the velocity. Hence, a Shear Stress Transport (SST) turbulence model (Appendix 2) was specified due to its capability in switching between the k-epsilon and the k-omega turbulence models when necessary in order to give better accuracy especially for the near wall treatment in the low-Reynolds number flow regime.

**Radiation Model**

Radiation between the surfaces is a significant contributor to inter-surface heat transfer and it is important to include radiation in the calculations. An emissivity at 0.9 was specified for all the metal surfaces including the interfaces. It is assumed that the fill gas is transparent to radiation and therefore a Monte Carlo radiation model was specified which calculates the radiation between all the respective surfaces. Solving for radiation consumes much computational time so the program was set such that the radiation was solved at every 50th iteration with 5,000 ray histories being calculated using a gray spectral model.
**Solver Control**

The solver control needs to be specified before a simulation can be run. The parameters of the solver will control how the ANSYS CFX Solver will be executed. This includes the time step and convergence control.

Simulations were set for a maximum of 750 iterations for the 3D model and 1,000 iterations for the 2D model where the radiation was calculated at every 50th iteration. The figure is chosen after conducting several simulations with different number of iterations during the mesh sensitivity study until it satisfied the standard convergence criteria where the residual target is $1e^{-4}$. Another parameter used to determine the convergence level by monitoring the net heat transfer of all the surfaces; the total of the heat flows for all the surfaces should equal to zero in steady state. Convergence was deemed to have occurred when the residual was below $1e^{-4}$ or the average domains temperatures were stable and the domain energy imbalances were below 3%. Figure 3.13 is an image of the solver control screen.

![Figure 3.13: Overall basic settings of solver control](image)

Figure 3.13: Overall basic settings of solver control
3.6 MESH SENSITIVITY STUDIES

Generally, the finer the mesh the more accurate the simulation result will be. However, the finer the mesh, the longer it takes the solver to achieve convergence. Therefore, it is vital to determine the optimum mesh quality to economise on computational time whilst retaining the required accuracy and reliability of the results. Mesh sensitivity studies were carried out to obtain the ideal mesh quality (node numbers) and the total number of iterations required. 2 types of mesh sensitivity studies were carried out. The first one was aimed at determining the optimum type of mesh for the CFD model. Even though conventional wisdom is that hexahedral meshes give better result than tetrahedral meshes, it was still worth to running simulations to show how significant the choice of type of mesh in this particular CFD simulation. The second study was undertaken to determine the ideal number of nodes for both 2D model for horizontal bundle case and 3D model for the inclined and vertical bundle case. The study is also used to obtain the optimum number of iterations required in order for a simulation to achieve convergence.

Tetrahedral vs. Hexahedral

A set of temperature combination was chosen for this study as given below

- Product : 65 °C
- Heat Flow : 30 °C
- Heat Return : 25 °C
- Test : 20 °C
- Sleeve : 17 °C

2 types of mesh which are tetrahedral and hexahedral were generated with about the same number of nodes for the selected bundle geometry as shown in Figure 3.14 below. The figure on the left is the bundle model with tetrahedral mesh and on right is the hexahedral mesh. Both have prism layers around all the edges in order to capture the near wall flow and heat transfer. The overall result of the study is presented in Table 3.1 below. Both models were compared on the domain imbalances and also the net heat flow (Watts). Domain imbalance is one the
parameters that is used to check the quality of the convergence. Small values of global domain imbalance indicate that conservation (mass, momentum and energy) has essentially been maintained. For a steady state simulation, the total amount of heat being released should equal to the total amount of heat being absorbed in the domain. Hence, the net heat flow in theory should equal to zero. However, due to numerical residual errors in CFD calculations the net heat flow may not be zero; however, it should be kept as close to zero as possible. By comparing the domain imbalance and net heat flow, the Table 3.1 below clearly shows that hexahedral mesh gives better results for a similar number of nodes and iteration numbers.

![Figure 3:14: Tetrahedral and hexahedral mesh that were generated for the mesh sensitivity study](image)

<table>
<thead>
<tr>
<th>Case</th>
<th>Type of Mesh</th>
<th>No. of Nodes</th>
<th>Iteration</th>
<th>Domain Imbalance (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Hexahedral</td>
<td>327,525</td>
<td>750</td>
<td>-0.2075</td>
</tr>
<tr>
<td>2</td>
<td>Tetrahedral</td>
<td>318,963</td>
<td>750</td>
<td>-0.2648</td>
</tr>
<tr>
<td>3</td>
<td>Tetrahedral</td>
<td>318,963</td>
<td>1,000</td>
<td>-0.4162</td>
</tr>
<tr>
<td>4</td>
<td>Tetrahedral</td>
<td>571,656</td>
<td>750</td>
<td>-0.6183</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Product</th>
<th>Test</th>
<th>HeatFlow</th>
<th>HeatReturn</th>
<th>Sleeve</th>
<th>Net HF</th>
<th>Ave. Gas Temp (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1363.450</td>
<td>-159.323</td>
<td>26.517</td>
<td>-4.608</td>
<td>-1227.030</td>
<td>-0.994</td>
<td>28.35</td>
</tr>
<tr>
<td>1363.830</td>
<td>-165.842</td>
<td>28.403</td>
<td>-1.953</td>
<td>-1223.850</td>
<td>0.594</td>
<td>29.15</td>
</tr>
<tr>
<td>1359.090</td>
<td>-164.564</td>
<td>27.518</td>
<td>-2.177</td>
<td>-1220.960</td>
<td>-1.091</td>
<td>29.23</td>
</tr>
</tbody>
</table>

Table 3.1: Comparison results between tetrahedral and hexahedral type of mesh
The results also show that even after increasing the number of iterations (Case 3) and improving the fineness of the mesh (Case 4) for the model with tetrahedral mesh, the domain imbalance and net heat flow for the model with hexahedral mesh still gives the better results (better convergence).

**3D Model**

The Table 3.2 below shows the results of the mesh sensitivity study that was carried out for the 3D bundle model with the hexahedral type of mesh. Several cases were simulated to determine the ideal mesh numbers and total number of iterations required for the 3D bundle in order to obtain a reliable and accurate result within a reasonable computation time.

<table>
<thead>
<tr>
<th>Case</th>
<th>No. of Nodes</th>
<th>Iterations</th>
<th>Completion Time</th>
<th>H-Energy Domain Imbalance (%)</th>
<th>Net Heat Flux (W)</th>
<th>AveTemp Gas (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2,583,400</td>
<td>1,000</td>
<td>15 hrs 35 mins 49.428 sec</td>
<td>0.6238</td>
<td>-0.8866</td>
<td>294.749</td>
</tr>
<tr>
<td>2</td>
<td>769,200</td>
<td>1,000</td>
<td>4 hrs 19 mins 6.132 sec</td>
<td>0.4052</td>
<td>2.05288</td>
<td>295.122</td>
</tr>
<tr>
<td>3</td>
<td>769,200</td>
<td>500</td>
<td>2 hrs 8 mins 14.739 sec</td>
<td>0.3904</td>
<td>0.44167</td>
<td>295.205</td>
</tr>
<tr>
<td>4</td>
<td>218,350</td>
<td>1,000</td>
<td>50 mins 35 sec 14.47 sec</td>
<td>0.1313</td>
<td>6.27649</td>
<td>294.623</td>
</tr>
<tr>
<td>5</td>
<td>327,525</td>
<td>500</td>
<td>57 mins 18 sec</td>
<td>0.2654</td>
<td>2.90642</td>
<td>294.725</td>
</tr>
<tr>
<td>6</td>
<td>327,525</td>
<td>750</td>
<td>1 hr 18 mins</td>
<td>0.1664</td>
<td>0.010796</td>
<td>294.722</td>
</tr>
</tbody>
</table>

Table 3.2: Results of mesh sensitivity study for the 3D bundle model

The most ideal bundle model was chosen based on the convergence criteria where the energy domain imbalance should be less than 3% (absolute value) for a heat transfer simulation and net heat flow should equal or very close to zero in a steady state condition. Based on Table 3.2, it is seen that Case 6 gives the lowest domain imbalances as well as a small net heat flow with a very reasonable computation time for the simulation to achieve convergence.
Chapter 3: CFD MODEL DEVELOPMENT

2D Model

Table 3.3 below shows the results of the mesh sensitivity study that was also carried out for the 2D model. Similar criteria were used as those employed for the 3D model. The mesh (51764 nodes) and number of iterations (1000) corresponding to Case 4 in Table 3.3 was chosen for carrying out the main set of calculations for the horizontal bundle.

<table>
<thead>
<tr>
<th>No.</th>
<th>Geometry File Name</th>
<th>No. Of Nodes</th>
<th>Iterations</th>
<th>Completion Time</th>
<th>H-Energy Domain Imbalance (%)</th>
<th>Net Heat Flux (W)</th>
<th>Ave. Temp Gas (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Bundle</td>
<td>200,000</td>
<td>500</td>
<td>2 hrs</td>
<td>-4.8596</td>
<td>-0.0133</td>
<td>302.651</td>
</tr>
<tr>
<td>2</td>
<td>Bundle01</td>
<td>100,000</td>
<td>500</td>
<td>1 hour</td>
<td>-6.5375</td>
<td>-0.018</td>
<td>302.541</td>
</tr>
<tr>
<td>3</td>
<td>Bundle02</td>
<td>51,764</td>
<td>500</td>
<td>26 mins 55 sec</td>
<td>-5.27</td>
<td>-0.014</td>
<td>302.503</td>
</tr>
<tr>
<td>4</td>
<td>Bundle02</td>
<td>51,764</td>
<td>1,000</td>
<td>23 mins 32 sec</td>
<td>1.7139</td>
<td>0.0041</td>
<td>301.561</td>
</tr>
</tbody>
</table>

Table 3.3: Results for the mesh sensitivity study for the 2D model for the analysis of horizontal bundle pipe
CHAPTER 4
CFD Simulations to Investigate The Pipe Thermal Stratification Phenomenon

4.0 SUMMARY OF CHAPTER

In the experimental work carried out as part of this project by Mr. Maung Maung Myo Thant, the heat flows into or out of the surfaces of pipes in a simulated bundle were measured. An important assumption in the experimental work was that each of the surfaces was maintained at a nearly constant temperature by passing water streams of known temperature through the pipes and around the sleeve. The (minimal) changes in water temperature between the inlet and outlet of the pipes and of the sleeve were used to estimate the heat gained or lost by the water and hence by the surface. A challenge to the assumption of constant surface temperature could arise as a result thermal stratification which can lead to the separation of colder and hotter water in the pipes and, hence, to the existence of significant variations in pipe surface temperature. The investigation focused on the product pipe which was expected exhibit the greatest thermal stratification. It was, in fact, found that thermal stratification effects could occur at the water flowrates originally planned for the experiments and this necessitated an increase in these flow rates. At the higher flow rates, though the calculations showed that thermal stratification effects were minimal, the (already small) difference between the inlet and outlet temperature of the water streams was further reduced and improved instrumentation (namely platinum resistance thermometry) was necessary to achieve the required accuracy. Thus, the CFD investigation was important in establishing the validity and design of the experimental method.
Chapter 4: CFD SIMULATIONS TO INVESTIGATE THE PIPE THERMAL STRATIFICATION PHENOMENON

In the first section of the chapter (section 4.1), a general introduction to the thermal stratification phenomenon is presented and a review of published information on this topic is given. Section 4.2 describes the general background to the investigation described in this chapter and Section 4.3 presents some (non-CFD) calculations that were carried out to determine the relevant Reynolds and Richardson numbers. The CFD studies of thermal stratification for the present experiments are described in Sections 4.4 to 4.7; the mesh generation activities are presented in Section 4.4 and section 4.5 provides details of the boundary conditions, physical models and solver control. Section 4.6 reports the results from the CFD calculations for the experiments originally planned and show that thermal stratification would have been significant. Section 4.7 describes the further CFD work that was undertaken for increased water flowrates; these calculations showed that variations in the pipe surface temperature would be reduced to an acceptable level at these increased flow rates and the experimental design was changed accordingly. The conclusions drawn from this study are summarised in Section 4.8.

4.1 INTRODUCTION TO THE THERMAL STRATIFICATION PHENOMENON

Thermal stratification is a phenomenon that typically occurs when cold and hot streams of a given fluid flow inside a pipeline. In this condition, the cold fluid which is relatively denser will occupy the lower part of the pipe while the hot fluid flows in the upper part. Thermal stratification is opposed by conduction between one region and the other and by mixing induced, say, by turbulence. The temperature gradient in the fluid due to the thermal stratification produces unwanted excessive thermal stress on the pipeline in all directions (radial, axial and circumferential) which will affect the structural reliability of the pipeline system. Some studies reported in the literature have shown that thermal stratification may be significant in pressurized water nuclear reactors (PWR’s).
Jo and Kang (2010) carried out three-dimensional (3D) computational fluid dynamics (CFD) analysis to predict the transient temperature distributions in a PWR for the geometrically complex bent piping of the pressuriser surgeline. The surgeline is a curved piping system which connects the reactor coolant system (RCS) that carries relatively cold fluid and the pressuriser which contains hotter fluid. In the work described by Jo and Kang, a three-dimensional CFD calculation involving conjugate heat transfer was conducted to predict the convective heat transfer coefficient and temperature distribution in the wall of an actual PWR surgeline subjected to internal thermally stratified fluid flow either during out- or in-surge operations. The in-surge condition typically occurs during the starting up of the reactor at cold condition where the hot fluid from the pressuriser flows into the surgeline which initially is filled up with cold fluid. Conversely, an out-surge situation occurs during the cooling-down process after reactor shutdown. Here, cold fluid from the pressuriser flows into the surgeline which initially is filled up by hot water. Three major simulations were carried out to study the flow regimes in the surgeline and also the effect of the thermal capacity of the pipe wall. The convective heat transfer coefficients at the pipe inner wall surfaces were determined. All the CFD calculations were conducted for both out-surge and in-surge cases. The CFD analytical approach was successful and was then implemented to analyse the thermal stratification of an actual pressurizer surgeline (Kori Unit 1).

Jo et al (2001) conducted a study which employs a numerical analysis to predict a stratified flow inside horizontal circular pipelines. The method implements a body-fitted non-orthogonal grid system to accommodate the pipe wall of the circular pipe geometry and the interface between the fluid regions at two different temperatures. The numerical method presented in the paper was used to analyse thermal stratification in the pressuriser surgeline in a pressurized water reactor (PWR). The outcomes of the analysis show that the numerical method successfully simulates the thermal stratification in a circular pipe as well as predicting the temperature distributions in the
The authors also mention that the method used in the study can also be employed for applications to various cases of thermally stratified flows in pipes and tanks with complex geometry and different flow conditions.

Da Silva et al (2007) report an experimental study that was carried out on a section designed to represent an injection nozzle of the steam generator of a PWR. The main aim of the experimental work was to study the effect of thermal fatigue due to thermal stratification. Deformation fatigue tests were also carried out on the experimental section which had been subjected to the effects of thermal stratification and also the virgin pipe in order to compare the structural reliability of the pipe. The results from the fatigue test were used to plot the strain versus number of fatigue cycles tests $\varepsilon$-N curves for both the experimental section and the virgin pipe. Numerical simulations on thermal stratification were also carried out using computational fluid dynamics (CFD). The experimental results were used to validate the computational modeling and analysis.

4.2 BACKGROUND ON PIPE THERMAL STRATIFICATION INVESTIGATIONS

As was mentioned above, one of the assumptions for the experiment that was carried out by Mr. Maung Maung Myo Thant as part of the present project was that the skin surface temperature of each pipe is fixed at a constant value. It is thus necessary to make sure the experimental system is capable of maintaining the temperature at a nearly constant value around and along the pipe length. Nevertheless, the (small) differences between the inlet and outlet temperatures of the water streams passing through the pipes (product, test, heat flow and heat return or through the coil at round the carrier pipe) should be sufficiently large to allow an accurate enough calculation of the heat lost or gained by the water streams. This allows an adequate determination of the rate of heat exchange between the respective surfaces and the gas inside the bundle.
The main objective of the study described in this section was to investigate, using computational fluid dynamics (CFD) modelling and simulation, the validity of the assumption of a constant temperature on each of the surfaces. Even when the inlet and outlet temperatures of the water streams differ by only a small amount, there is the possibility that thermal stratification will lead to regions of the surfaces which differ significantly in temperature from the mean of the inlet and outlet values, thus vitiating the assumption of a nearly constant surface temperature for each surface and producing an error in the apparent heat transfer coefficient for that surface. The study was focussed on the product pipe at horizontal, inclined and vertical positions. The inlet water temperature was fixed at a constant temperature and, thus, thermal stratification (if it occurs) would develop along the pipe. This is different to the PWR cases reviewed above where water streams at different temperatures are introduced into the pipe as a result of operation of the reactor system. Since, in the experiments carried out in the present project, the flow rate is relatively low, thus creating a laminar flow (particularly in the product pipe which has the largest diameter), remixing of the fluid due to turbulence would not occur and the system may be prone to thermal stratification. To study the effects of thermal stratification in the product pipe, the following combination of temperatures was selected:

- Sleeve : 17
- Test : 20
- Heat Flow : 30
- Heat Return : 25
- Product Inlet : 65

The wall temperature of the product pipe was allowed to vary as a result of thermal stratification but the other surface temperatures remained constant. The temperature values were chosen to simulate the worst situation where the temperature gradient between the product pipe and the
surroundings is high. Under these conditions, the thermal stratification effect would be maximised.

Two types of simulations were carried out in the thermal stratification study. One was a comparative study between a bundle experiencing thermal stratification in the product pipe and an “ideal” (non-stratified) case bundle model. For the stratified model, the temperature of the test, heat flow, heat return and sleeve pipes were specified while the wall temperature distribution of the product pipe is calculated by applying CFD to the inside-tube flow and heat transfer. For the “ideal” case, the surface temperature of all the 5 fluid boundary walls were each set to a constant value around and along the pipe length. For this particular simulation, only a horizontal pipe bundle was studied.

The second simulation was to study the effect of inclination angles on thermal stratification in the product tube. The angles studied ranged from 0 to 90 degrees from the horizontal in 10 degree increments. The temperature combinations listed above were used but the mass flow rate in the product tube was increased from 1/6 kg/second to 1/3 kg/second due to the complexity of the CFD model and the need to achieve quicker convergence.

4.3 BASIC CALCULATIONS

Before undertaking the CFD simulation, it was important to do a basic calculation to estimate the Reynolds Number which will determine the type of fluid flow model. The Reynolds Number is calculated based on the equation below:

\[
Re = \frac{\rho VL}{\mu} \tag{Equation 4.1}
\]
where $\rho$ is the density of the fluid, $V$ is mean velocity of the fluid, $L$ is the characteristic length (equal to the diameter for a tube) and $\mu$ is the dynamic viscosity of the fluid. For a water flow at 10 litres/min through the product tube (the rate planned originally), the Reynolds number is 1256, and the flow is thus laminar.

It is also important to determine whether, inside the product tube, natural convection or forced convection is the more dominant. An estimate of the significance of natural convection can be made by calculating the Richardson Number by using the equation below:

$$Ri = \frac{g\beta(T_{hot} - T_{ref})L}{V^2}$$  

Equation 4.2

where $g$ is the gravitational acceleration, $\beta$ is the thermal expansion coefficient, $T_{hot}$ is the hot wall temperature, $T_{ref}$ is the reference temperature, $L$ is the characteristic length (equal to the diameter for a tube) and $V$ is the characteristic velocity.

It is known that, natural convection is negligible when $Ri < 0.1$, forced convection is negligible when $Ri > 10$ and neither is negligible when $0.1 < Ri < 10$. It is found that the calculated $Ri$ is a function of temperature difference and is approximately $3663 \Delta T$ where $\Delta T$ is the difference between $T_{hot}$ and $T_{ref}$. Therefore, it is confirmed that natural convection is more dominant in the product fluid flow.

Based on these calculations, it can be qualitatively predicted that thermal stratification is highly likely to occur in the product pipe due to the fact that the flow regime is laminar and natural convection is dominant in comparison with forced convection.
Chapter 4: CFD SIMULATIONS TO INVESTIGATE THE PIPE THERMAL STRATIFICATION PHENOMENON

4.4 GEOMETRY DEVELOPMENT AND MESH GENERATION

A three-dimensional model of the product pipe was developed using ANSYS ICEM. The overall length of the pipe is 3m and the outer diameter of the product pipe is 0.3556m and its wall thickness is 15mm. The mesh grids were generated using ANSYS ICEM and a tetrahedral type of mesh was used. A mesh with 1, 972, 056 nodes was created in order to get a good convergence whilst retaining a sufficient accuracy in the CFD calculations.

The use of a tetrahedral type of mesh is not efficient in capturing the shear and boundary layer physics. Therefore prism layers have been created to effectively capture the effect of these components near the surface while maintaining the ease of mesh generation away from the surfaces using the tetrahedral mesh.

![Figure 4:1](image1.png)

(a) (b)

Figure 4:1: The cross-sectional picture of the tetrahedral mesh for the bundle thermal stratification model (a) and hexahedral mesh for the “ideal” bundle (b).
4.5 BOUNDARY CONDITIONS AND PHYSICAL MODELS

3 domains have been created which consist of 1 product fluid domain (pink region), 1 solid domain (yellow region) and 1 gas domain (blue region) as shown in Figures 4.1a and Figure 4.2 above. All fluid-solid boundary walls in the fluid domains (between the product fluid and the product tube wall and between the gas and the solid surfaces in contact with it) were modelled as ‘no-slip’ boundaries. Adiabatic walls were specified for each end for each domain. This clearly is appropriate for the experiment which was arranged to be consistent with this assumption. However, even for inclined tube bundles, it is not inappropriate since lengths of bundle typified by the 3m length used in the experiments are separated by spacers which hold the internal pipes in position. It is reasonable to assume that these spacers block the flow of the gas that fills up the space between the internal pipes.

In the case of a horizontal pipe, the gas space flows induced by natural convection are two-dimensional in nature. If the pipe is vertical or inclined, the gas space flows become three-
dimensional but they are occurring over a length which is restricted in the experiments to 3m (the distance between the ends of the test section) and, in the actual bundle application, to the distance between spacers (also likely to be of the order of 3m). In the practical application of tube bundle systems, the tube-side flows are likely to well mixed and turbulent and are thus capable of being represented through a one-dimensional model. However, in the experiments, there is a need to restrict the tube-side (water) flows so as to obtain a sufficient temperature difference to measure accurately the heat loss or gain through the tube wall. Typically, a temperature change of around 1-2 °C is needed for this purpose. The assumption in the experiments is that the wall temperature will be maintained at a nearly constant value (taken as the average of the inlet and outlet temperatures). However, thermal stratification effects can cause parts of the surface to differ significantly in temperature from this average value and, indeed, from the inlet/outlet temperatures themselves. This could invalidate the assumption of a constant surface temperature in calculating the tube-to-wall heat transfer coefficient from the experimental data.

In modelling the heat transfer in the simulated product pipe, it is important to take account of heat conduction in the pipe wall. In the computations, the 15mm thick steel product pipe wall was modelled as a continuous solid. A mass flow rate of 10 litre per minute or equivalent to 1/6 kg per second was specified at the inlet of the product tube and a pressure boundary condition was specified for the outlet. A similar mass flow rate was also used for the experimental work; it is important that the CFD model replicates the actual bundle set-up for the experiments to ensure the validity of the CFD model as well as the accuracy and consistency of the simulation results. After running several simulations with different mass flow rates, the value of 1/6 kg/s was finally chosen because it gives acceptable temperature differences between the inlet and outlet for calculating the heat loss and also a reasonable computation time to achieve good convergence with relatively low domain imbalances. With this flow rate through the 14 inch
Based on initial calculations that were performed prior to the start of the CFD simulations, laminar flow was specified for the product fluid domain.

There are 2 interfaces in the CFD model, one is between the product fluid and the inner product pipe wall and the other is between the outer product pipe wall and the fill gas (air). Therefore fluid-solid type of interfaces was specified for both interfaces. It is specified that heat is allowed to flow between the interfaces through the simulated pipe wall. The product fluid and the filled gas were modelled as a constant property water and air respectively at atmospheric pressure. A Boussinesq buoyancy (Appendix 1) model was chosen for both fluid regions. The Shear Stress Transport (SST) turbulence model (Appendix 2) was specified for the gas region. According to CFX, SST turbulence model is ideal for a low-Reynolds number flows and gives high accuracy boundary layer simulations.

It is expected that net radiative transfer between the outer surface of the product tube and its surroundings will be an important part of the overall heat transfer process. Thus, radiation was included in the CFD calculations. An emissivity at 0.9 was specified for all the metal surfaces. It is assumed that the fill gas is radiatively transparent; therefore the Monte Carlo radiation model was specified which calculates the radiation from surface to surface. Solving the radiation equations consumes much computational time; therefore the calculation was arranged such that the solutions of the radiation equations were implemented only at every 50th iteration with 2,500 ray histories calculated at each implementation using a gray spectral model.

The wall temperatures of the test, heat flow, heat return and sleeve surfaces were specified at fixed and typical values. The emissivity for these surfaces was also set as 0.9. For the product pipe, the surface temperature was not specified; rather, the distribution of temperature over this
surface was determined from the CFD calculations in order to observe whether there will be significant temperature variations due to thermal stratification effects. The product pipe outer surface (in contact with the gas) was modelled as smooth wall with an emissivity of 0.9.

The simulations were set for a maximum of 2,000 iterations where the radiation was calculated at every 50th iteration. The level of convergence of the calculations can be evaluated by monitoring the sum of the heat transfer rates of the respective surfaces; the total heat flow of all the pipe walls should equal to zero for a steady state analysis. The convergence was deemed to have occurred when the residual fell below 1e-4 or when the average domain temperatures were stable and the domain energy imbalances were below 3%.

4.6 CFD RESULTS ANALYSIS

Two CFD models were developed for the purpose of this study, one is for the “ideal” case as shown in Figure 4.1 (b) (Model 1) and the other one is stratified model as shown in Figure 4.1 (b) (Model 2). The first model has one gas domain with 5 fluid boundary walls (product, heat flow, heat return, test and sleeve) whose wall temperatures are fixed at constant values around the circumference as well as along the pipe length. In the second model (which takes account of thermal stratification in the product tube) the surface temperatures of the heat flow, heat return, test and sleeve surfaces are fixed; however, the full distribution of surface temperature on the product tube is determined as part of the calculation. The second model is relatively more complicated and requires more computation time to achieve convergence. The model has 3 domains which are inter-connected to each other as explained previously in the above section.

Figure 4.3 below shows the overall temperature profile at cross-section half way along the 3m bundle (i.e. at 1.5m from each end) for the ideal case (fixed product tube temperature). Generally, it shows that regions of low gas temperature can be found at the bottom part of the
product pipe as well as the bundle. The product pipe seems to be heating up the top part of the sleeve pipe due to its proximity as well as the large temperature difference between the two surfaces (65 °C for product tube and 17 °C for sleeve). The average interstitial gas temperature for “ideal” (Model 1) case is 28.35 °C whereas for the stratified (Model 2) case it is 30.7 °C.

![Cross-sectional temperature distribution of the bundle at 1.5m plane (ideal case)](image)

**Figure 4.3:** Cross-sectional temperature distribution of the bundle at 1.5m plane (ideal case)

### 4.6.1 Comparison of “Ideal Case” and Case with Thermal Stratification

After 2,000 iterations, the simulation achieved the convergence criteria with approximately between 3-5% domain imbalances and 0-1 W net heat flow in steady state condition. The results are shown in the Table 4.1 below. A negative wall heat flow indicates that the heat is being absorbed while a positive value indicates that the surface is losing heat.
Chapter 4: CFD SIMULATIONS TO INVESTIGATE THE PIPE THERMAL STRATIFICATION PHENOMENON

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Unit</th>
<th>&quot;Ideal&quot; Case</th>
<th>Stratified Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average Gas Temperature</td>
<td>°C</td>
<td>28.35</td>
<td>30.70</td>
</tr>
<tr>
<td>Average Product Wall Temperature</td>
<td>°C</td>
<td>65.00</td>
<td>63.15</td>
</tr>
<tr>
<td>Average Product Wall Heat Flow</td>
<td>Watt</td>
<td>1363.45</td>
<td>729.445</td>
</tr>
<tr>
<td>Average Test Wall Heat Flow</td>
<td></td>
<td>-159.320</td>
<td>-257.498</td>
</tr>
<tr>
<td>Average Heat Return Wall Heat Flow</td>
<td></td>
<td>-4.60800</td>
<td>-91.2239</td>
</tr>
<tr>
<td>Average Sleeve Wall Heat Flux</td>
<td></td>
<td>-1227.03</td>
<td>-362.496</td>
</tr>
<tr>
<td>Net Wall Heat Flow</td>
<td></td>
<td>-0.994</td>
<td>-0.6943</td>
</tr>
</tbody>
</table>

Table 4.1: Comparison on CFD results between ideal case and stratified case

Generally, in terms of the heat flow direction (positive means heat is going out of the domain and negative indicates that heat is being absorbed), the CFD results of stratified model show good agreement with the “ideal” bundle model. The calculated net heat flow (the sum of the heat flows between the surfaces and the gas space) should, of course, be zero; the calculated values are less than 1 W for both cases which indicates that a satisfactory simulation has been achieved. The calculated average product tube outside wall temperature for the stratified case is 63.15 °C which represents around a 3% drop from the 65 °C of the fixed wall temperature case. However there is a significant drop in the average wall heat flow for the product pipe. This is because the temperature variation around the product pipe wall is large with the lowest temperature (61 °C) being found in the bottom part of the pipe as clearly shown in Figures 4.4 and Figure 4.5 below.
4.6.2 Effect of Inclination Angles

For the calculations on the effect of tube inclination, a mass flow rate in the product tube of 1/3 kg/second (20 litre per minute) was used for the inlet boundary condition instead of 1/6 kg/second as specified in the first stratification test. Due to the complexity of the CFD model, the flow rate was increased in order to achieve faster convergence. The temperature of the sleeve pipe was also changed from 17 °C to 5 °C. This was done to increase the temperature gradient so that the effect of thermal stratification can be seen more clearly. The simulation was carried out with 2,000 iterations again solving for
radiation at every 50th iteration. Each simulation took approximately 25 hours to achieve convergence. In total, 10 simulations were carried to cover the range of inclination angle from 0 to 90 degrees at 10 degree intervals, spanning the range from horizontal to vertical.

The general trend of the simulations indicates that the thermal stratification effect becomes more severe as the inclination angle increases. This is shown by the reduction of both average outlet and product wall temperature as the inclination angle increases as represented by graphs in Figure 4.6 below. The overall results are compiled and tabulated in Table 4.2.

### Table 4.2: CFD results of various inclination angles for the thermal stratification study

<table>
<thead>
<tr>
<th>Angle of Inclination</th>
<th>Average Temperature (deg C)</th>
<th>Wall Heat Transfer Rate (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Outlet Temp</td>
<td>Product Wall Temp</td>
</tr>
<tr>
<td>0</td>
<td>64.73</td>
<td>63.87</td>
</tr>
<tr>
<td>10</td>
<td>64.54</td>
<td>63.81</td>
</tr>
<tr>
<td>20</td>
<td>64.48</td>
<td>63.78</td>
</tr>
<tr>
<td>30</td>
<td>64.46</td>
<td>63.77</td>
</tr>
<tr>
<td>40</td>
<td>64.45</td>
<td>63.77</td>
</tr>
<tr>
<td>50</td>
<td>64.44</td>
<td>63.77</td>
</tr>
<tr>
<td>60</td>
<td>64.42</td>
<td>63.73</td>
</tr>
<tr>
<td>70</td>
<td>64.41</td>
<td>63.69</td>
</tr>
<tr>
<td>90</td>
<td>64.40</td>
<td>63.67</td>
</tr>
</tbody>
</table>

As can be observed from Table 4.2 above, the product tube loses heat at a rate which increases with increasing angle of inclination; all of the other tubes experience a heat gain. The magnitude of the calculated net heat flow is not zero (due to residual errors in the calculations) but is small compared with the heat losses from the product pipe and heat gains through the other surfaces. The sleeve and the test pipes are absorbing the most heat which reflects the fact that their surface areas are higher and their surface temperatures are lower than those of the other two surfaces (heat flow and heat return).
Figure 4.6: Effect of inclination angles on average temperature of outlet, product wall and gas

Based on the CFD calculations, the average temperature for the outlet is reduced from 64.73 to 64.4 °C as the bundle is moved from the horizontal to the vertical position. The corresponding temperature drop in the product tube water stream thus changes from 0.27 °C (minimum) to 0.6 °C (maximum) as the bundle inclination changes from horizontal to vertical. These results show that more thermal stratification will occur as the inclination angle increases; this arises since the thermal stratification effect arises from natural convection effects inside the product tube and this become stronger as the gravitational force increases with increasing tube inclination. Figures 4.7 and 4.8 show the circumferential and longitudinal product pipe surface temperature distributions for the horizontal and vertical pipe cases.
For vertical case, in the product pipe, an interesting phenomenon was seen where one side of the pipe is cold and the opposite side is slightly hotter. The low temperature region was found at the side which is close to the test pipe (smaller pipe on the right of product pipe). This is because the temperature gradient between the product pipe (65 °C) and the sleeve pipe (5 °C) is far greater than the temperature gradient between the product pipe (65 °C) and the test pipe (20 °C). Therefore more heat is going towards the region closer to the sleeve pipe to heat up the sleeve pipe until equilibrium is achieved in the whole bundle system.

Figure 4.7: Cross-sectional temperature distribution of product pipe at 1.5m plane for horizontal (left) and vertical (right) bundle
4.7 THE EFFECT OF INCREASING FLOW RATE IN MINIMISING THERMAL STRATIFICATION PROBLEM

For the previous thermal stratification study, a 10 litre/minute flow rate was used for the product pipe fluid and there was a significant effect on the rate of heat loss from the pipe as a result of thermal stratification; as was explained above, this change in heat loss applies to the experiments. In the actual bundle case, the surface of the product tube would be expected to be nearly isothermal and the system behaviour would be expected to be close to the “ideal” case. The question thus arises: can the thermal stratification effect in the experiments be reduced to an acceptable by increasing the water flow rate through the tube? Further calculations were therefore done for the horizontal bundle case with the product tube water flow rate successively increased from 10 litre/minute to 20, 30, 40 and 50 litre/minute. Hence, in total there were 4 further simulations carried out and each run was completed after 2,000 iterations where radiation was calculated at every 50th iteration. All 4 simulations achieved the convergence criteria satisfactorily. Table 4.3 below shows the overall results of the study.
Table 4.3: CFD results on effect of flow rates towards the average temperature of gas, outlet and product wall.

<table>
<thead>
<tr>
<th>Flow rate (litre/minute)</th>
<th>Gas Average Temp (°C)</th>
<th>Outlet Average Temp (°C)</th>
<th>Product Wall Ave. Temp. (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>29.29</td>
<td>64.46</td>
<td>63.33</td>
</tr>
<tr>
<td>20</td>
<td>29.37</td>
<td>64.62</td>
<td>63.77</td>
</tr>
<tr>
<td>30</td>
<td>29.39</td>
<td>64.61</td>
<td>63.93</td>
</tr>
<tr>
<td>40</td>
<td>29.44</td>
<td>64.70</td>
<td>64.17</td>
</tr>
<tr>
<td>50</td>
<td>29.46</td>
<td>64.77</td>
<td>64.27</td>
</tr>
</tbody>
</table>

Both graphs below clearly show that the average temperature of the gas, the outlet and also the product wall increase as the flow rate increases (fluid velocity also increases). An increase in temperature provides a good indication that the effect of thermal stratification on the pipe surface has been minimized.

Figure 4:9: Graphs showing average temperature (°C) of gas, outlet and product wall against flowrate (litre/minute)

It can be seen from the CFD results that, at high water flow rates through the product tube, though the effect of thermal stratification is minimised, the outlet temperature is very close to the inlet temperature which is 65 °C. To measure this small temperature difference with
sufficient accuracy to use it to calculate the heat loss from the pipe is obviously a challenge! This challenge was met in the laboratory work by using a more sensitive temperature sensor, namely a platinum resistance thermometer.

The detailed calculated internal water temperatures in the product tube are presented in Figures 4.10 to 4.15 below. As the flow rate is increased, the temperature variations decrease and the effects of thermal stratification are minimised (note that there are changes of scale between the successive diagrams).

![Cross-sectional temperature distribution of the product pipe at 1.5m plane for the original case (10 litre per minute flowrate)](image)

Figure 4.10: Cross-sectional temperature distribution of the product pipe at 1.5m plane for the original case (10 litre per minute flowrate)
Chapter 4: CFD SIMULATIONS TO INVESTIGATE THE PIPE THERMAL STRATIFICATION PHENOMENON

Figure 4:11: Cross-sectional temperature distribution of the product pipe at 1.5m plane for 20 and 30 litre/minute flow rate.

Figure 4:12: Cross-sectional temperature distribution of the product pipe at 1.5m plane for 20 and 30 litre/minute flow rate.

Figure 4:13: Axial temperature distributions of the 3m long product pipe at the mid-plane for the 10 litre/minute flow rate.
Figure 4:14: Axial temperature distributions of the 3m long product pipe at the mid-plane for the 20 and 30 litre/minute flow rate.

Figure 4:15: Axial temperature distributions of the 3m long product pipe at the mid-plane for the 40 and 50 litre/minute flow rate.
In general, the temperature variation becomes much less from the top to the bottom and from the inlet to the outlet of the product pipe as the flow rate increases. For example, for the 50 litre/minute flow rate, the temperature of the product fluid ranges from 64.75 °C to 64.90 °C which is almost constant. In contrast, where the flow rate was 10 litre/minute, the temperature varied from 61 °C to 64.6 °C and a significant effect was observed on the net heat loss.

4.8 CONCLUSIONS FROM THERMAL STRATIFICATION STUDY

The CFD calculations indicated that the impact of thermal stratification is significant at the low water flow rates originally planned for the experiments. The angle of inclination also has an important impact to the thermal stratification of the product pipe. This is clearly shown in Figure 4.6 where it is seen that the average outlet product pipe wall temperature falls as the inclination angle increases. This is because as the inclination angle goes up, the influence of gravity is getting stronger. The fact that, in the planned experiments (where the water flow rate was intended to be 10 litres/min), the product tube-side fluid is in laminar flow also contributes to the increase in temperature variations in the product pipe as it goes up from horizontal to vertical position.

For the experiments, it is important to take the necessary actions to mitigate the thermal stratification effect because of its impact on the overall heat transfer calculation for the bundle system. One possible method is to increase the fluid velocity by using a heat transfer enhancement device in the form of a spiral-grooved insert that will be slotted in the pipes to create a high velocity flow that will increase the tube-side heat transfer coefficient. Another alternative is to simply increase the flow rate of the fluid. However, it is also important to make sure that the (small) temperature difference between the inlet and the outlet of the pipe is large enough to be measured with sufficient precision to give an adequately accurate calculation of the heat lost and gained by the water stream.
The effect of increasing the fluid flow rate was studied and the CFD calculations indicate that, the thermal stratification effect in the experiments could be reduced to a sufficiently low level by increasing the flow rate of the stream that passes through the pipe. Implementing an increase in water flowrate was relatively straightforward but the original method for measuring the difference between the inlet and outlet temperatures of the water stream (i.e. the use of thermocouples) had to be changed.

The temperature variations along and around the pipe are insignificant when the flow rate of the product fluid is at 40 & 50 litre/minute as shown in the tabulated results. The CFD calculations for both cases show that the average temperature of the product wall is 64.17 °C and 64.27 °C respectively. At these flow rates, the flow velocity is much higher and the flow is well into the turbulent regime. The turbulence gives better mixing which distributes the heat more evenly along and around the product pipe.

At flow rates of 40 & 50 litres/min, the temperature difference between the inlet and outlet is relatively very small compared to what it would have been with the initially planned flow rate (10 litre/minute flow rate). However, the temperature difference it is still sufficiently large to be measured with acceptable accuracy using Resistance Temperature Detectors (RTD’s). This temperature difference can be used to calculate the heat loss from the product fluid that passes through the product pipe.
CHAPTER 5
CFD Model Validation

5.0 SUMMARY OF CHAPTER

In computational modelling, it is vital to ensure the model used is validated against analytical or experimental data to give confidence that the results produced are reliable and accurate. The primary focus of this chapter is to report the validation activities that were carried out in collaboration with Mr. Maung Maung Myo Thant (who carried out a wide range of experiments on the selected bundle geometry) to ensure the reliability of the computational model. Section 5.1 presents the results of the validation activity with reports of comparisons of computed and measured heat flows for the bundle in the horizontal, inclined (at 5, 30, 45 and 60 degrees) and vertical orientations. For each orientation, two experimental data sets were used; the heat flows were predicted using the CFD code from the measured mean temperatures of the surfaces and these heat flows were compared with the measured values.

Three further validation exercises are described in Sections 5.2, 5.3 and 5.4 are were as follows:

- Comparisons were made between a two dimensional (2D) model of the bundle and the 3D bundle model (Section 5.2).

- Most of the CFD calculations described in this thesis were carried out using the ANSYS CFX code. However, additional calculations were carried out using the STAR-CCM+ produced by CD ADAPCO and these are compared with equivalent calculations done with ANSYS CFX (Section 5.3).
• A study was conducted to verify the importance of radiation in conjugate heat transfer in the bundle. Two models were prepared where one includes the radiation model and the other one excludes the radiation (Section 5.4).

Finally, in Section 5.5, work aimed at investigating the effect of Joule-Thompson cooling in pipeline risers is reported; this work was done using the OLGA simulation package.
5.1 VALIDATION OF 3D CFD MODEL AGAINST EXPERIMENTAL RESULTS

It was vital to validate the CFD bundle model against the experimental results to ensure the reliability and accuracy of the CFD predictions which are used for to generate the bundle heat transfer database (Chapters 6 and 7) on which the later for the neural network analysis was based (Chapter 8).

The 3D CFD bundle model that was described in Chapter 3 was used for this validation activity. The boundary conditions, physical models, fluid properties and solver control were as described in Chapter 3. The validation work was done for bundle inclination angles of 0 (horizontal), 5, 30, 45, 60 and 90 (vertical) degrees. Two sets (Set 1 and Set 2) of temperature combinations were chosen for the study. For each set, the temperatures for each surface at each inclination angle were approximately constant. However, the temperatures used in the CFD simulations were the exact values measured in the experimental work done by Mr. Myo Thant. It is important to use the exact temperatures that were used for the experimental work to ensure the reliability of the outcomes from the validation work.

The overall surface heat flow results for both experiment and CFD predictions are gathered in Table 5.1 and Table 5.2 for Set 1 and Set 2 respectively. The complete results with the net heat and average temperature of the gas calculated from both experiment and CFD were given in the Annex at the end of this chapter.
<table>
<thead>
<tr>
<th>Angle</th>
<th>Temperatures</th>
<th>EXP</th>
<th>CFD</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>31.49 31.88 50.84 45.36 20.31</td>
<td>58.38 19.23 244.30 160.04 -441.65</td>
<td>100.50 70.49 241.70 169.40 -581.15</td>
</tr>
<tr>
<td>5</td>
<td>29.62 29.78 50.71 45.21 20.35</td>
<td>-19.48 -17.30 258.28 179.76 -368.77</td>
<td>65.33 40.24 246.88 171.49 -524.07</td>
</tr>
<tr>
<td>30</td>
<td>30.00 21.79 50.40 45.14 20.66</td>
<td>74.17 -146.36 268.85 164.90 -339.58</td>
<td>106.84 -77.78 252.39 174.67 -453.13</td>
</tr>
<tr>
<td>45</td>
<td>30.50 25.32 50.42 45.40 22.65</td>
<td>26.55 -80.90 228.06 177.06 -320.37</td>
<td>78.80 -45.35 234.11 163.60 -431.31</td>
</tr>
<tr>
<td>60</td>
<td>30.18 25.67 50.39 45.30 22.80</td>
<td>-8.64 -79.22 238.04 186.02 -309.44</td>
<td>74.80 -39.10 229.63 162.95 -430.40</td>
</tr>
<tr>
<td>90</td>
<td>30.64 25.31 50.74 45.16 21.92</td>
<td>14.85 -83.86 237.93 180.19 -322.74</td>
<td>116.22 -35.71 230.50 161.93 -471.07</td>
</tr>
</tbody>
</table>

Table 5.1: Set 1 pipe temperatures with associated heat flows (W) at different inclination angles from experiment and CFD predictions

<table>
<thead>
<tr>
<th>Angle</th>
<th>Temperatures</th>
<th>EXP</th>
<th>CFD</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>25.93 25.86 49.8842 45.5732 20.24</td>
<td>-99.83 -47.60 269.60 188.62 -318.31</td>
<td>-9.39 -2.01 248.15 186.99 -424.65</td>
</tr>
<tr>
<td>5</td>
<td>25.1072 24.606 49.6862 45.3822 19.98</td>
<td>-96.85 -59.33 268.26 182.89 -283.24</td>
<td>-22.56 -14.78 247.11 189.09 -400.60</td>
</tr>
<tr>
<td>30</td>
<td>23.1217 22.964 50.3012 44.8347 20.81</td>
<td>-162.12 -39.78 280.87 206.28 -271.08</td>
<td>-73.89 -36.15 258.68 178.79 -325.79</td>
</tr>
<tr>
<td>45</td>
<td>25.6922 25.023 50.5912 45.0882 22.43</td>
<td>-86.81 -56.67 259.69 166.39 -267.57</td>
<td>-38.38 -28.12 243.29 169.09 -345.01</td>
</tr>
<tr>
<td>60</td>
<td>25.6672 25.633 50.4802 45.1522 22.51</td>
<td>-80.91 -66.83 250.37 178.21 -262.08</td>
<td>-37.89 -18.23 241.11 164.46 -350.07</td>
</tr>
</tbody>
</table>

Table 5.2: Set 2 pipe temperatures with associated heat flows (W) at different inclination angles from experiment and CFD predictions
The results for Set 1 (from Table 5.1) are plotted in Figure 5.1. Though there are some discrepancies both in sign and magnitude between the measured and predicted values, the majority of the results show satisfactory agreement and the plots in Figure 5.1 show that the general trends have been captured.

The results for the CFD simulations for Set 2 (Table 5.2) are illustrated in Figure 5.2. Again the agreement between the experiments and the CFD predictions is generally good. For the product, test and sleeve pipe the CFD predictions heat flows give somewhat higher values than experimental results whereas for the heat flow and heat return, the CFD predictions are generally somewhat lower than the experimental values. under-predict as can be seen from Figure 5.2.
Figure 5:1: Graph plots showing the pipe heat flows at different inclination angles for temperature Set 1 for validation of CFD predictions against experimental results.
Based on the results presented in this section, it can be concluded that reasonable agreement was obtained between the CFD predictions and the experimental results. It can thus be concluded that the CFD model used in the bundle heat transfer analysis has been successfully validated. This is vital to ensure the reliability and accuracy of the CFD calculations which will be the
main input for the neural network analysis that will be further explained in Chapter 8 of this thesis.

5.2 VALIDATION OF 2D AGAINST 3D CFD MODELS

In this work, there were two types of model used in undertaking the CFD simulations to calculate the heat transfer for horizontal, inclined and vertical bundle position. For horizontal case, the heat transfer does not change significantly over the length of the pipe. Hence, a 2D model is adequate for the heat transfer calculation for the horizontal bundle.

Being able to use 2D model to simulate the horizontal case of the bundle gives a great saving in the computational time required to complete the simulations. This is because with 2D model each simulation can be completed within half an hour with 4 processors; compared to the 3D model, this represents a time saving of around 70%. It is therefore economic to predict bundle heat transfer for more temperature combinations for the horizontal case to make the heat transfer database more complete and accurate. It is possible to calculate horizontal bundles using both the 2D and 3D methods and it is of interest to compare the two methods in this context. A few temperature combinations were selected (for the horizontal bundle case) and calculations done with both the the 3D and 2D models. Details of the selected cases are given in Table 5.3 and the results using the 2D and 3D models are shown in Tables 5.4 and 5.5 respectively. for the validation study between the 2D and 3D CFD model as shown in Table 5.3.
The values extracted from the CFD results are the heat flow (Watt) for each of the pipe surfaces in the bundle, namely the product, test, heat flow, heat return and sleeve pipe. Generally, in
terms of the heat flow directions (positive means heat loss and negative value represents heat gains) the 2D results are similar those obtained with the 3D model. In steady state case, the amount of heat being absorbed by the sleeve pipe also represents the amount of heat goes out of the bundle to the environment. For the validation study, the bundle heat loss in Watt per metre was used in order to compare the results between the 2D and 3D models.

Generally, the bundle heat losses (W/m) for the 2D model show good agreement with the results for the 3D model, the difference being approximately 7% on average. Figure 5.3 below shows clearly the comparison on the temperature distribution at the mid-plane of the 2D and 3D models for a few of the selected cases (i.e. temperature combinations).

Figure 5.3: Cross-sectional temperature profile at mid-plane for both 2D (top) and 3D (bottom) models for 3 selected cases; Case 1 (a), Case 4 (b) and Case 7 (c).
Essentially, the cross-sectional temperature distribution predicted by the 2D model is reasonably similar to that predicted by the 3D model. However, there are some differences in the quality of the contour plots between the 2D and 3D model; the temperature contours for the 2D are much smoother. This is because the 2D model has much better quality (finer) mesh, i.e., greater number of nodes per unit cross-sectional area in comparison to the 3D model. In practice, the 2D model can be implemented with a much finer mesh due to the fact that the computational time to complete the simulation is much smaller than that for the 3D model.

In conclusion, the validation study has proven that the 2D model has an acceptable level of accuracy for use in carrying out heat transfer simulations for the horizontal bundle. This has given a great advantage in reducing the computational time to complete the simulations for the horizontal bundle case. Hence, a lot more temperature combinations can be simulated in order to make the heat transfer database more extensive and complete.

5.3 COMPARISON BETWEEN CFX AND STAR-CCM+ ON BUNDLE HEAT TRANSFER ANALYSIS

There are quite a number of commercial computational fluid dynamics (CFD) software packages available in the market such as CFX and Fluent from ANSYS, STAR-CCM+ and STAR-CD from CD-Adapco and TransAT from Ascomp. Fundamentally they are all relatively similar and capable of undertaking simulation work related to fluid dynamics and heat transfer. Each of it has its own advantages and disadvantages and in the end, it is up to the user to decide which tool is the most practical and economic for the work that the user would like to carry out.

Prior to this project, there was a discussion on which CFD software would be used to undertake the CFD simulations for the bundle heat transfer analysis. Previously, ANSYS CFX had been used at Imperial College and Bristol University to carry out simulations of the 1m water-filled and the 3m air-filled horizontal bundles used in the earlier experiments. The current research
work followed on from this earlier work (as explained in Chapter 2) and it was therefore decided to use CFX as the fundamental model. This choice was consistent with the fact that ANSYS CFX is commonly used in academic work and the fact that the code and was already available in the Chemical Engineering Department of Imperial College at the start of this research project. However, as part of the basic validation of the CFD methodology, it was decided to repeat the calculations with another code. For this purpose, the STAR-CCM+ code from CD-Adapco (which was also available in the Department of Chemical Engineering, Imperial College) was selected.

**Geometry Development**

For this comparative study, a CFD model was developed using each of the software based on the 3m long bundle pipe used in the experimental work undertaken in a parallel research work by Mr. Myo Thant as illustrated in Figure 5.4 below. The bundle consist of one 14 inch product pipe, one 8 inch test pipe and two 4 inch heat flow (heater) and heat return pipes. These internal pipes are inserted in a 30 inch sleeve pipe. The 3D CFD model was created based on the external diameter of the internal pipes which are the product, test, heat flow and heat return whereas for the sleeve pipe, it was drawn based on the internal diameter.
Chapter 5: CFD MODEL VALIDATION

Figure 5:4: Schematic diagram showing the geometry of the bundle used for the comparative study between ANSYS CFX and STAR-CCM+

Mesh Generation

Once the geometry was drawn for both softwares, a set of mesh was generated using the meshing tool of each software. As described in Chapter 3, a hexagonal type of mesh was used for ANSYS CFX. For STAR-CCM+, a polyhedral mesh was set up. These types of mesh are considered as the most reliable that can be implemented in the respective codes for the heat transfer analysis. The following figure shows the mesh grids generated for the 3D bundle model for both softwares.
For the CFX hexagonal mesh, a total of 327,525 nodes were generated whereas for STAR-CCM+ polyhedral mesh a total of 954,789 nodes were generated. It is also important to note that, during the initial stage of the study, the total number of nodes for the STAR-CCM+ polyhedral mesh was set at a value which was approximately similar to the number of hexahedral nodes used in CFX. However, the STAR-CCM+ simulation results did not meet the convergence criteria. After varying the number of nodes a few times, the number of polyhedral nodes in STAR-CCM+ was finally set at 954,789 where the convergence criteria were duly met.

**Boundary Conditions and Physical Model**

Similar boundary conditions were defined for both the CFX and the STAR-CCM+ simulations. One fluid domain with 5 fluid boundary walls were specified for both models. The fluid boundary walls were respectively the product, test, heat flow, hereturn and sleeve pipes. All fluid boundary walls were modelled as “no-slip” boundary (zero velocity at wall). A set of adiabatic walls were defined at both ends of the 3m zone calculated. As mentioned previously in Chapter 3, it has been assumed that the 3D zone to be calculated is limited to 3m since there are spacers positioned at both ends to hold the internal pipes in their position. The gas domain was filled with nitrogen gas. The interstitial nitrogen was modelled as a constant property gas at
atmospheric pressure respectively. A Boussinesq buoyancy model was chosen and Shear Stress Transport (SST) Turbulence model was specified for both cases (see Appendices 1 and 2 for further details). The emissivity of the walls for both cases was set at 0.9. It is assumed that the fill gas is transparent to radiation and therefore a Monte Carlo radiation model was specified which calculates the radiation between all the respective surfaces.

**Temperature Combinations**

One of the assumption in the work is that all the surface temperature of the walls are fixed at a constant value around and along the bundle length. Similar temperature combinations was used for both models and are given as follows:

- **Product**: 65 °C
- **Test**: 20 °C
- **Heat flow**: 30 °C
- **Heat return**: 25 °C
- **Sleeve**: 17 °C

**Solver Control**

Simulations were set for a maximum of 750 iterations for the ANSYS CFX model and 1500 iterations for the STAR-CCM+ model. For ANSYS CFX the radiation was calculated at every 50th. iterations. In STAR-CCM+ there is no option to specify the interval for radiation calculation, hence it was calculated at every iteration. The number of iterations for STAR-CCM+ was chosen after conducting several simulations with different number of iterations until the results satisfied the standard convergence criteria where the residual target is 1e-4. Another parameter used in judging the convergence level was the net heat transfer of all the surfaces; the
sum of the heat flows for all the surfaces of all the pipe walls should equal to zero in for a steady state analysis.

Simulation Results

Steady state simulations were carried out for both cases at horizontal position. Both simulations achieved convergence and met the criteria. However, it was noted that the ANSYS CFX was completed much quicker than that of STAR-CCM+. This would be expected since the total number of nodes for the STAR-CCM+ model is much higher and radiation is calculated at every iteration for STAR-CCM+. In CFD, calculation for radiation is a very time consuming.

The heat flows for each pipe was extracted from the simulation results for both cases and presented in the Table 5.6 below. As will be seen, there is reasonable agreement between the two codes. The results also show that CFX always gives higher heat flow for all the pipes. The net heat flows for both cases were also calculated. For ANSYS CFX the calculated net heat flows was -0.988602 Watts whereas for STAR-CCM+, the net heat was a lot higher which is 5.189711 Watts. For steady state analysis, the closer the net heat to zero, the better is the result.

<table>
<thead>
<tr>
<th>Pipe</th>
<th>ANSYS CFX</th>
<th>STAR-CCM+</th>
</tr>
</thead>
<tbody>
<tr>
<td>Product</td>
<td>1363.45</td>
<td>1199.69</td>
</tr>
<tr>
<td>Test</td>
<td>-159.323</td>
<td>-147.8917</td>
</tr>
<tr>
<td>Heat Flow</td>
<td>26.5169</td>
<td>26.503</td>
</tr>
<tr>
<td>Heat Return</td>
<td>-4.60788</td>
<td>-0.986963</td>
</tr>
<tr>
<td>Sleeve</td>
<td>-1227.03</td>
<td>-1082.506</td>
</tr>
</tbody>
</table>

Table 5.6: Comparison of heat flows from the pipe to the interstitial gas (W) calculated by ANSYS CFX and STAR-CCM+ respectively
Figure 5.6 below shows the cross-sectional nitrogen temperature distribution for both cases taken at the middle plane of the domain (at 1.5m distance from both end walls). The nitrogen velocity distribution for the two cases, taken at the same plane as the cross-sectional temperature distribution, is illustrated in Figure 5.8. Both temperature distribution and gas velocity plot for both cases are reasonably similar.

It can be seen from Figures 5.7 and 5.8 that, for the chosen temperature combination (where the product pipe has a high temperature, i.e. 65 °C) the heat loss from the product pipe is high compared to the other pipes as is consistent with Table 5.6. In Figure 5.7, it is seen that, in both predictions, the product pipe is heating up the top section of the sleeve pipe due to the large temperature difference (product at 65 °C and sleeve at 17 °C) and the close proximity of the two surfaces.

The gas velocity plot in Figure 5.7 shows how the movement of the gas promotes redistribution of temperature in the bundle (enlarged version of the figure is shown in Appendix 16). The hotter fluid moves up to occupy the top section of the domain whereas the relatively colder fluid
moves to the lower part of the bundle. Eventually, an equilibrium distribution of temperature is reached with the upper zone temperature being higher than that of the lower zone. High velocity gradients are observed around the circumference of the hot of the product pipe.

Figure 5:7: Comparison on gas velocity profile between ANSYS CFX (left) and STAR-CCM+ (right)

It can be concluded from the comparative study that both CFX and STAR-CCM+ are capable of undertaking reliable bundle heat transfer analysis. The agreement between the two codes gives encouragement to the view that the general CFD methodology is a viable one in addressing this problem.

5.4 THE EFFECT OF RADIATION

As was mentioned previously in the literature review, almost all of the numerical models in the open literature fail to mention the impact of radiation on the heat transfer behaviour. Natural convection heat transfer in enclosures often occurs simultaneously with surface radiation heat exchange among the walls of the enclosures in transparent fluid media such as air. Surface radiation will modify the wall heat fluxes and affect the rate of loss or gain of heat by the surfaces in the bundle. In many circumstances, this would lead to changes in the surface
temperatures and therefore to an *interaction or coupling* of the radiation and natural convection heat transfer processes. However, in the present work, the heat transfer data base is based on a matrix of *specified* surface temperatures for the respective surfaces. The heat losses or gains from or by the surfaces are a result of a summation (for the specified surface temperatures) of the heat transferred by the natural convection and radiation processes. For a given set of surface temperatures, the heat transfer by radiation through the (transparent) interstitial gas is fixed and is simply added (within the CFX or STAR-CCM+ code) to the heat transfer by natural convection to give the total heat loss or gain by the surface. However, it is of interest to consider the relative contributions of natural convection and radiation to the total heat loss or gain by the surfaces. The relative contributions of radiation and natural convection can be determined by repeating bundle calculations assuming the standard value of emissivity (0.9) and zero emissivity respectively.

To evaluate the effect of radiation, two cases were prepared using a 3D model (illustrated in Figure 5.11 below) with the mesh, boundary conditions, physical model, fluid properties and solver control as described in Chapter 3. In the first case (Case 1), the emissivity was set to the standard value of 0.9. However, in the other case (Case 2) radiation was eliminated by setting the emissivity of all the wall boundaries to zero.
Five temperature combinations were selected for the comparative studies. Table 5.7 below provides the heat flows of all the pipes with the net heat flow for both cases. Then the two cases were compared in terms of the pipe heat flows for all the temperature combinations. Case 2 which has no radiation model show a great reduction in the amount of heat flow showing that the contribution of radiation is very significant.
Chapter 5: CFD MODEL VALIDATION

Temperatures

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>20</td>
<td>40</td>
<td>35</td>
<td>18</td>
<td>45.07</td>
<td>21.54</td>
<td>177.67</td>
<td>114.67</td>
<td>-226.70</td>
<td>-0.97</td>
</tr>
<tr>
<td>30</td>
<td>20</td>
<td>40</td>
<td>35</td>
<td>18</td>
<td>191.24</td>
<td>47.08</td>
<td>164.79</td>
<td>105.92</td>
<td>-414.93</td>
<td>-0.05</td>
</tr>
<tr>
<td>20</td>
<td>20</td>
<td>50</td>
<td>45</td>
<td>18</td>
<td>-92.31</td>
<td>50.09</td>
<td>277.59</td>
<td>200.66</td>
<td>-337.78</td>
<td>-1.94</td>
</tr>
<tr>
<td>30</td>
<td>20</td>
<td>50</td>
<td>45</td>
<td>18</td>
<td>142.38</td>
<td>76.25</td>
<td>257.57</td>
<td>186.78</td>
<td>-510.31</td>
<td>0.17</td>
</tr>
<tr>
<td>65</td>
<td>20</td>
<td>30</td>
<td>25</td>
<td>17</td>
<td>1363.45</td>
<td>-159.32</td>
<td>26.52</td>
<td>-4.61</td>
<td>-1227.03</td>
<td>-0.99</td>
</tr>
</tbody>
</table>

Heat Flows Case 2 - With No Radiation

<table>
<thead>
<tr>
<th>Product</th>
<th>Test</th>
<th>Heat Flow</th>
<th>Heat Return</th>
<th>Sleeve</th>
<th>Net</th>
</tr>
</thead>
<tbody>
<tr>
<td>-32.71</td>
<td>-15.28</td>
<td>72.03</td>
<td>39.40</td>
<td>-62.68</td>
<td>0.76</td>
</tr>
<tr>
<td>32.70</td>
<td>-22.55</td>
<td>68.48</td>
<td>38.98</td>
<td>-117.98</td>
<td>23.39</td>
</tr>
<tr>
<td>-55.89</td>
<td>-30.33</td>
<td>113.86</td>
<td>70.82</td>
<td>-99.94</td>
<td>-1.47</td>
</tr>
<tr>
<td>8.76</td>
<td>-35.41</td>
<td>106.73</td>
<td>69.51</td>
<td>-148.66</td>
<td>0.92</td>
</tr>
<tr>
<td>128.29</td>
<td>-32.95</td>
<td>29.12</td>
<td>9.22</td>
<td>-55.84</td>
<td>77.83</td>
</tr>
</tbody>
</table>

Table 5.7: Comparison on heat flows (Watt) of all pipes between case with radiation and case with no radiation

As would be expected, the cross sectional distribution of gas temperature (Figure 5.9) is essentially the same for Cases 1 and 2. Since the gas is treated as being transparent, the radiation has no effect on the gas temperature which is essentially governed by the surface temperatures and the associated natural convection processes. However, as seen in Table 5.7, the radiation contributes strongly to the heat losses and gains from the various surfaces.
Chapter 5: CFD MODEL VALIDATIONS

Figure 5.9: Comparison on cross-sectional temperature (taken at middle plane) between case with radiation (left) and case with no radiation (right)

It can be concluded from the study that radiation does play an important role in the heat transfer that occurs inside the bundle pipe. Eliminating the radiation from the CFD calculation, reduces the pipe heat flows which would cause a significant inaccuracy in bundle performance calculations. Thus, in generating the CFD data bases on which bundle performance predictions are based (see Chapters 6 and 7) it is vital to include both natural convection and radiation.

5.5 OLGA SIMULATIONS ON JOULE-THOMSON COOLING

Insulation is the most widely employed thermal management strategy to contain the heat inside a pipeline system in order to mitigate solids formation and deposition problems. However as the exploration and production (E&P) activities go into deeper water and further away from shore, the pipeline system becomes longer and the environment becomes more challenging (low temperature and high pressure conditions). Thus heat losses from the pipeline system become more significant and active heating systems (as investigated in the present work) become of greater interest. However, there is a phenomenon which can lead to a reduction in temperature of the produced fluid even in the absence of heat losses; this is the Joule-Thomson (JT) effect. Fundamentally, the Joule-Thomson effect is associated with changes in temperature caused by pressure changes when the production fluid contains gas. This effect can cause significant
temperature drops, in the riser section of the flowlines. It may lead to the flowline temperature falling below the ambient temperature and more importantly below the wax appearance temperature (WAT) or cloud point which will induce the solids formation such as hydrates, wax and asphaltenes deposition. This effect cannot be dealt with by insulation; some form of active heating is required.

As part of the present work, a limited study was carried out of JT cooling of a production fluid passing through a riser section of a flowline. The study was carried out using a dynamic and multiphase flow simulator, namely the OLGA code. Two cases were prepared for the comparative study. One case is a bare pipe and the other case uses insulation for a randomly created pipeline profile. The fluid that flows inside the pipe was chosen from the internal database of OLGA. The fluid used was multiphase fluid of hydrocarbon with some gases.

Figure 5.13 below shows the pipeline profile which is represented by the black line. A horizontal pipe runs for approximately 5km (16,000 feet) at roughly 500m water depth. Then riser section of the pipeline consists of a vertical pipe which rises for about 600m (i.e to 50m above sea level). The study was carried out for various thickness of insulation to see the effect of the insulation on the Joule-Thomson cooling. The chosen insulation thicknesses were 10mm, 20mm, 40mm and 50mm. The graphs clearly show that even the insulation was increased 5 times higher than the original one which is at 10mm, the pressure drop at the riser section is still very significant (drop from 60 bar to 5 bar) which causes a temperature reduction at the receiving end of the riser.

From these limited OLGA simulations, it can be deduced that Joule-Thomson cooling is inevitable particularly at the riser section of a pipeline system even with a perfect insulation. Therefore it is important to employ an alternative thermal management method such as active
heating together with insulation to provide better thermal performance to keep the temperature of the pipeline outside the solids formation and deposition regions.
Figure 5:10: Graphs showing the pressure and temperature profile for different insulation thickness for the Joule-Thomson cooling study using OLGA version 6
## ANNEX

### Full results data on pipe heat flows (W) at different inclination angles from experimental work and CFD predictions for temperature Set 1

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Angle</th>
<th>Product Temperature (°C)</th>
<th>Test Temperature (°C)</th>
<th>HeatFlow Temperature (°C)</th>
<th>HeatReturn Temperature (°C)</th>
<th>Sleeve Temperature (°C)</th>
<th>Gas (°C)</th>
<th>Q (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>13/0/E</td>
<td>0°</td>
<td>25.8513</td>
<td>26.009</td>
<td>25.93015</td>
<td>25.8257</td>
<td>25.905</td>
<td>25.8635</td>
<td>50.1194</td>
</tr>
<tr>
<td>13/0/C</td>
<td>0°</td>
<td>25.9302</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13/5/C</td>
<td>5°</td>
<td>25.107203</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6/30/E</td>
<td>5°</td>
<td>26.5245</td>
<td>25.76</td>
<td>25.692194</td>
<td>25.6977</td>
<td>25.709</td>
<td>25.3335</td>
<td>45.2654</td>
</tr>
<tr>
<td>6/30/C</td>
<td>5°</td>
<td>25.692194</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6/45/E</td>
<td>5°</td>
<td>25.6025</td>
<td>25.732</td>
<td>25.66715</td>
<td>25.5677</td>
<td>25.698</td>
<td>25.3285</td>
<td>45.2542</td>
</tr>
<tr>
<td>6/45/C</td>
<td>5°</td>
<td>25.66715</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6/60/E</td>
<td>5°</td>
<td>25.5353</td>
<td>25.613</td>
<td>25.57415</td>
<td>25.4317</td>
<td>25.498</td>
<td>25.4685</td>
<td>45.2204</td>
</tr>
<tr>
<td>6/60/C</td>
<td>5°</td>
<td>25.57415</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Full results data on pipe heat flows (W) at different inclination angles from experimental work and CFD predictions for temperature Set 1

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Angle</th>
<th>Product Temperature (°C)</th>
<th>Test Temperature (°C)</th>
<th>HeatFlow Temperature (°C)</th>
<th>HeatReturn Temperature (°C)</th>
<th>Sleeve Temperature (°C)</th>
<th>Gas (°C)</th>
<th>Q (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>12/0/E</td>
<td>0°</td>
<td>22.9923</td>
<td>23.251</td>
<td>23.12165</td>
<td>22.9667</td>
<td>23.001</td>
<td>22.9638</td>
<td>50.5524</td>
</tr>
<tr>
<td>12/0/C</td>
<td>0°</td>
<td>23.1217</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12/5/E</td>
<td>5°</td>
<td>22.5057</td>
<td>22.657</td>
<td>22.51715</td>
<td>22.5557</td>
<td>22.667</td>
<td>22.6635</td>
<td>45.6594</td>
</tr>
<tr>
<td>12/5/C</td>
<td>5°</td>
<td>22.51715</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5/30/E</td>
<td>5°</td>
<td>26.5245</td>
<td>25.76</td>
<td>25.692194</td>
<td>25.6977</td>
<td>25.709</td>
<td>25.3335</td>
<td>45.2654</td>
</tr>
<tr>
<td>5/30/C</td>
<td>5°</td>
<td>25.692194</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5/45/C</td>
<td>5°</td>
<td>25.66715</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5/60/E</td>
<td>5°</td>
<td>25.5353</td>
<td>25.613</td>
<td>25.57415</td>
<td>25.4317</td>
<td>25.498</td>
<td>25.4685</td>
<td>45.2204</td>
</tr>
<tr>
<td>5/60/C</td>
<td>5°</td>
<td>25.57415</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Full results data on pipe heat flows (W) at different inclination angles from experimental work and CFD predictions for temperature Set 1
CHAPTER 6

CFD Horizontal Bundle Analysis (2D Model)

6.0 SUMMARY OF CHAPTER

The primary focus in this chapter is on the analysis of the two-dimensional (2D) CFD simulations that have been carried out for the horizontal bundle. In principle, the flow of the gas inside a horizontal bundle is two dimensional (2D), i.e. it can be represented in terms of a plane normal to the axis of the bundle. Therefore, a 2D CFD model is sufficient to calculate the heat transfer between the pipes inside the bundle. This greatly reduces the computational time needed to complete the required set of simulations covering a wide range of surface temperature combinations. Some CFD work on the horizontal bundle configuration had already been undertaken within the TMF project, as explained in Chapter 2. For the sake of completeness, and to confirm the reproducibility of the calculations, it was important to rerun the earlier simulation again; the results obtained are discussed in Section 6.1. Section 6.2 gives an overview of the present study with sample CFD results being presented. In total some 1,926 temperature combinations were simulated for the horizontal bundle case. The heat transfer parameters such as heat flows and heat transfer coefficients for each of the surfaces inside the bundle were obtained and gathered as a database. Section 6.3 gives further details of cases selected to exemplify the results. Finally in Section 6.4, discusses the application of a standard correlation for convective heat transfer to some of the data generated from the CFD simulations for the horizontal bundle case.
6.1 BACKGROUND ON 2D MODEL FOR HORIZONTAL BUNDLE

As explained in section 2.4, Chapter 2, the work on horizontal bundle was previously carried out in Transient Multiphase Flow (TMF) Joint Industrial Project (JIP). In the TMF work, in total, 1,683 temperature combinations were simulated using CFD. In order to have an extensive and a complete heat transfer database, it is necessary to also conduct CFD simulations for the horizontal bundle with similar temperature combinations for this particular work. However, it is also important to note that in the TMF work for the horizontal bundle, the temperature of the sleeve pipe was set at a constant value of 10.15 °C for all the temperature combinations.

Previously, the computing power and the number of available licenses for CFD software were limited. Therefore, it was really important to have a practical size of temperature matrix to ensure the work could be completed within the reasonable duration. By fixing the temperature of the sleeve pipe (which also represents the ambient temperature of the sea) to a constant value, the total number of temperature combinations required for the work was reduced significantly. The fact that a 2D model was used to solve the heat transfer for the horizontal bundle led to a reduction of the total computational time to complete the whole simulations.

In the present work, the simulations of the 1,683 temperature combinations carried out in the previous work were repeated and an additional 243 temperature combinations were also simulated. These additional temperature combinations are important to amplify the key temperature vectors for the horizontal bundle in order to cover a higher temperature range for the sleeve pipe (sea ambient temperature); which is between 15-25 °C. This temperature range was selected to solve the heat transfer calculations for the horizontal bundle at relatively shallow water depth conditions where the sea ambient temperature is slightly higher. For the inclined and vertical bundle cases, which will be further discussed in the next chapter, the sleeve temperature
ranged between 5-15 °C. These wide ranges of sea ambient temperature that are covered in the present work for both horizontal and inclined and vertical bundles will ensure a more reliable heat transfer database in order to adequately predict (using neural network interpolation) the thermal interactions inside the bundle for various water depth conditions from shallow to deep water.

Theoretically the flow of the air that fills up the space between the pipes along the horizontal bundle length is two dimensional. Hence, it is sufficient to use a simpler and less complicated two dimensional (2D) model (Figure 6.1) to carry out the heat transfer calculation using CFD. Before the simulations were carried out, the 2D model was first validated against the 3D model as discussed in Chapter 5, in order to ensure the accuracy and reliability of the results generated from the 2D model CFD simulations.

Figure 6:1: Two dimensional model for horizontal bundle case
6.2 GENERAL OVERVIEW OF THE CFD RESULTS

A total of 1,926 simulations (temperature configurations) were carried out on 2D bundle model for the horizontal bundle pipe. All the CFD simulation files, which are in the format of .def files (which consist of the geometry, the mesh, the physical model, fluid properties, the boundary conditions and also the solver control) and .ccl files (which provide the temperature combination) were compiled in a batch file. Rather than preparing one simulation file at a time which could be a time consuming activity, the whole simulations files were combined in one batch file to be run serially and continuously. Considering the large number of temperature combinations to be simulated, this particular working step helps to improve the efficiency of the preparation activity for each of the simulations. Hence, it reduces the overall time taken to complete the whole range of CFD simulations. However, it is vital, to ensure the accuracy of the CFD results by checking that the base case of the simulation which is the .def file is correct before undertaking the batch simulations.

In general, each simulation takes approximately 30 minutes to achieve convergence with a net heat flow (heat balance) close to zero Watt and with less than 3% domain imbalances.

Figures 6.2 and 6.3 below show screenshots taken from the output file (.out) of a CFD simulation. It provides the detailed breakdown of the mesh for a particular simulated case.

```
+-------------------------------------------------------------------+
| Mesh Statistics                                                   |
+-------------------------------------------------------------------+
Domain Name : Default Domain
  Total Number of Nodes  =  51764
  Total Number of Elements =  35571
  Total Number of Prisms  =  330
  Total Number of Hexahedrons =  22241
  Total Number of Faces   =  52160
+-------------------------------------------------------------------+
```

Figure 6:2: Mesh statistics of the simulated case
Figure 6.3 below shows a typical screenshot from the CFX solver after a simulation has been completed. The left hand side of the solver window, typically provides the plots for the Root Mean Square (RMS) residuals for the governing equations which are the mass, momentum and energy equations, turbulence equation and also the user defined parameters which in this case are the temperatures and also the heat flows for all the pipes inside the bundle.

The right hand side of the solver window provides all the detailed information such as the physical model, fluid properties, boundary conditions, intitial conditions, mesh statistics and the solver control that have been defined during the pre-processing stage of the work as presented in Appendix 15. It also gives the summary of the calculations for all the equations that are being solved at each timestep until the convergence is achieved as shown in Figure 6.4 below.

After the simulation has been successfully completed, typically at the end of the output file, it shows the total time taken to complete the simulation and also the location of the results file that has been saved. However, if the simulation is stopped in the middle of simulation or convergence is not achieved, an error message will be displayed at the end of the output file. The output file always gives a good indication or hint when it comes to debugging errors in CFD.
Chapter 6: CFD HORIZONTAL BUNDLE ANALYSIS (2D MODEL)

Figure 6.3: Summary of the calculation of all the equations at each timestep

Figure 6.4: A screenshot showing a typical window of the solver after a simulation is completed

Figure 6.5 below shows the global imbalance percentage for the radiation component of the heat flux and also the global imbalance for the total energy transfer. Domain imbalance percentage is one of the parameters that is used as a good check on the quality of the convergence. Small values of global domain imbalance indicates that conservation (mass, momentum and energy)
has essentially been maintained. The global domain imbalance is simply calculated with

equation below:

\[
\% \text{imbalance} = \frac{\text{Total In} - \text{Total Out}}{\text{Total In}}
\]

Equation 6.1

The calculation of imbalance percentage is considerably straight forward for the heat transfer
bundle simulation as there is only one domain which is the fluid gas domain.

```
<table>
<thead>
<tr>
<th>Boundary</th>
<th>Flux</th>
<th>1-Radiation</th>
</tr>
</thead>
<tbody>
<tr>
<td>HeatF</td>
<td>2.94E-05</td>
<td></td>
</tr>
<tr>
<td>Product</td>
<td>-3.64E-03</td>
<td></td>
</tr>
<tr>
<td>Sleeve</td>
<td>-1.75E-02</td>
<td></td>
</tr>
<tr>
<td>Test</td>
<td>-4.1E-02</td>
<td></td>
</tr>
</tbody>
</table>

Global Imbalance : 1.35E-18

Global Imbalance, in %: 0.0000 %
```

```
<table>
<thead>
<tr>
<th>Boundary</th>
<th>Flux</th>
<th>1-Energy</th>
</tr>
</thead>
<tbody>
<tr>
<td>HeatF</td>
<td>-5.35E-04</td>
<td></td>
</tr>
<tr>
<td>Product</td>
<td>-9.07E-03</td>
<td></td>
</tr>
<tr>
<td>Sleeve</td>
<td>-5.35E-03</td>
<td></td>
</tr>
<tr>
<td>Test</td>
<td>-1.31E-03</td>
<td></td>
</tr>
</tbody>
</table>

Domain Imbalance : 1.741E-04

Domain Imbalance, in %: 1.0147 %
```

Figure 6.5: An extract from the CFD output file showing the domain imbalance and global
imbalance for the calculation on radiation

6.3 ANALYSIS OF 2D BUNDLE CFD SIMULATIONS

Figure 6.6 below shows an example of a temperature distribution (left) and gas velocity plot
(right) that were extracted from the CFD simulation results. The gas temperature and gas
velocity plots were taken at the cross-section of the bundle at the middle plane of the gas domain
(0.5mm from each end). The heat flow (heater) pipe has the highest temperature amongst the
pipes in the bundle; therefore it gives out heat the most in the systems. A positive heat flow
(measured in Watt per metre of the bundle) indicates that the heat is being lost from the pipe.

In CFD, the average temperature of the gas was calculated by averaging all of the temperature of
the nodes in the domain. The average temperature of the gas gives a good reference point in
order to determine whether the heat is going in or out of all the pipe surfaces in the bundle. In general, pipes that are hotter than the average temperature of the gas (in this case the product, heat flow and heat return pipes) will give out heat (+ve Watts); which, whereas colder pipes (test and sleeve) will be absorbing heat (-ve Watts). The figure also shows the minimum temperature of the gas that fills up the space between the pipes inside the bundle.

The thermal interactions between the pipes inside the bundle will occur until equilibrium is achieved where the net heat flow should equal (or be very close to) zero. It is also important to note that the amount of heat being absorbed by the sleeve pipe indicates the total heat loss of the whole bundle to the surroundings. It can be observed clearly from the temperature distribution as shown in the figure below that the relatively hotter gas (less dense) occupies the top section of the bundle, whereas the cold gas (indicated by blue region) fills up the bottom section of the bundle. In Figure 6.6, the picture on the right shows the velocity profile of the gas at the middle plane of the computational domain (0.5mm from each end of the domain). Higher velocity can be observed at the surfaces which have high temperature and also in the narrow gaps between the pipes such as the top left section of the bundle between the product and the sleeve pipe. The flow in the gas is fundamentally driven by buoyancy force. Thus, even a small change in the temperature causes significant increase in the velocity of the gas.
Figure 6.6: Typical temperature and gas velocity plot being extracted from the CFD simulation

Figure 6.7 below demonstrates clearly a typical condition during bundle operation where the heater pipe heats up the product pipe, keeping the product pipe temperature sufficiently above the wax appearance temperature (WAT) of the production fluid to prevent the formation and deposition of solids. As can be observed from the Figure 6.7 below, even though the average gas temperature is lower than the temperature of the product pipe by approximately 1 °C, the product pipe is absorbing heat (-33.6 Watt per metre). This proves that the heater pipe can still efficiently warm up the product pipe due to its proximity to the product pipe and also the high temperature gradient.
In the case illustrated in Figure 6.8, the temperature of the product pipe is much higher than in the example illustrated in Figure 6.7. In this case, the product pipe is heating up the top section of the sleeve pipe due to very high temperature gradient (70 °C difference in temperature) and the proximity of the product pipe to the sleeve pipe. In this particular condition, the sleeve pipe absorbs a lot more heat (703.8 Watt) in comparison with the previous case (259.1 Watt); in both cases, the heat is lost to the surroundings. The results show in Figures 6.7 and 6.8 illustrate the complexity of the thermal interactions between the surfaces. This is why a relatively complex procedure (namely neural network fitting) is required to interpolate the results.

Figure 6.7: Typical temperature combinations for a bundle during operation
Chapter 6: CFD HORIZONTAL BUNDLE ANALYSIS (2D MODEL)

6.4 CORRELATION FOR 2D CFD PREDICTION DATA

After completing the CFD simulations for all the temperature combinations for the horizontal, inclined and vertical bundles, the results were compiled as a heat transfer database. The database consists of the temperatures for each pipe in the bundle (product, test, heat flow, heat return and sleeve) with their respective heat fluxes (W) and heat transfer coefficient (W m\(^{-2}\).K\(^{-1}\)), the latter being defined as the ratio of the heat flux to the difference between the surface temperature and the average gas temperature. For the horizontal bundle case, 1,926 heat transfer data sets have been generated using CFD simulation. Due to the complexity of the thermal interaction between the surfaces, which is driven by combined convective and radiative heat transfer in the bundle system, the neural network approach seems to be the best method to establish a working correlation and this approach will be discussed further in Chapter 8.

However, it is also interesting to correlate the CFD generated data using a form of natural convection heat transfer correlation that has been widely reported in the literature. For this purpose, a set of CFD results from the horizontal bundle case (2D model) was chosen. Given the

Figure 6.8: Other example of temperature plot for different temperature combination
temperatures and the respective heat flows, the heat transfer coefficients for each pipe surface were calculated using the equation below.

\[ \alpha_p = \frac{\dot{q}_p}{T_p - T_b} \]  

Equation 6.2

where subscript \( p \) represent the pipe in the bundle which are the product, test, heat flow, heat return and sleeve, \( \dot{q}_p \) is the heat fluxes (W/m) and \( T_p \) is the surface temperature of the respective pipes (K), whereas \( T_b \) is the mean bulk temperature of the gas (K). The Nusselt number of each pipe surface is given by Equation 6.3

\[ Nu_p = \frac{\alpha_p D_p}{\lambda} \]  

Equation 6.3

where \( D_p \) is the pipe diameter (m) of the respective surfaces and \( \lambda \) is the thermal conductivity (W/m K) of the gas taken at a mean of the surface and bulk temperatures. The common method correlating in correlating a set of natural convection data is to correlate the Nusselt number as a function of Rayleigh number which is defined as

\[ Ra = \frac{g \beta D^3 \Delta T}{\nu \kappa} \]  

Equation 6.4

where \( g \) is the acceleration of gravity, \( \beta \) is the volumetric thermal expansion of the gas (K\(^{-1}\)), \( D \) is the diameter (m) of each pipe, \( \Delta T \) is the difference between the temperature of pipe surfaces and the bulk temperature, \( \nu \) is the kinematic viscosity (m\(^2\)/s) and \( \kappa \) is the thermal diffusivity (m\(^2\)/s). In the present data, there is a difficulty in that radiative heat transfer can play a
significant role. However, assuming (very tentatively!) that the radiation component can be subsumed into the coefficient, correlations of the Nu/Ra form can be derived for the present data. Table 6.3 (at the end of this section) shows values of Nu and Ra calculated for a selection of the data obtained from the CFD calculations. Separate values are calculated for each of the surfaces. There are several correlations for natural convection heat transfer available in the literature. For the present analysis, a correlation form developed by Churchill and Churchill (1975) was used in order to correlate the present (CFD) data. A number of CFD results were selected to establish the correlation for the specific bundle configuration using the chosen correlation. The basic form of Churchill and Churchill correlation for natural convection is defined as

\[ Nu = aRa^{1/3}[f(Pr)]^{1/3} \]  

Equation 6.5

where \( Nu \) is the average Nusselt number for the surface, \( Ra \) and \( Pr \) are the Rayleigh Number and Prandtl Number respectively. Churchill and Churchill defined the function of \( f(Pr) \) as follows

\[ f(Pr) = \left[1 + \left(\frac{0.5}{Pr}\right)^{16}\right]^{-9/16} \]  

Equation 6.6

Equation 6.5 was fitted to the CFD data for each surface, the optimal value of \( a \) and the standard deviation being calculated in each case. The calculated values of \( a \) and of the standard deviation for each pipe are given in Table 6.1. The results show that the product pipe data is fitted relatively well with a 6.4% standard deviation. The standard deviations for the other pipes are considerably higher due to the highly complex flow and the thermal interactions inside the bundle. Each pipe interacts thermally interacted with all the other pipes in the bundle. In order to
take into account of these highly complex thermal interactions, the Nusselt number is correlated by multiplying the Rayleigh number (to the conventional power of 1/3) of a given pipe with the function of Prandtl number (to the conventional power of 1/3) and the product of multiplication of the Rayleigh number of the other pipes in the bundle as given by the Equation 6.6 below.

\[ \text{Nu} = c \text{Ra}^{\frac{1}{3}} f(\text{Pr})^{\frac{1}{3}} (\text{Ra}_2 \text{Ra}_3 \text{Ra}_4 \text{Ra}_5)^d \]  

Equation 6.7

where the Nusselt number of the selected pipe is as a function of its own Rayleigh number and the product of the Rayleigh numbers for the other surfaces in the bundle. The data were refitted and the constants ‘c’ and ‘d’ were calculated as shown in the Table 6.2.

<table>
<thead>
<tr>
<th>Pipe Surface</th>
<th>a</th>
<th>Standard deviation (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Product</td>
<td>2.563</td>
<td>6.4</td>
</tr>
<tr>
<td>Test</td>
<td>3.126</td>
<td>24.9</td>
</tr>
<tr>
<td>Heat Flow</td>
<td>2.760</td>
<td>10.8</td>
</tr>
<tr>
<td>Heat Return</td>
<td>4.238</td>
<td>44.0</td>
</tr>
<tr>
<td>Sleeve</td>
<td>3.979</td>
<td>15.1</td>
</tr>
</tbody>
</table>

Table 6.1: Constant ‘a’ and the standard deviations of each pipe calculated by correlating the CFD data with Equation 6.4

<table>
<thead>
<tr>
<th>Pipe Surface</th>
<th>c</th>
<th>d</th>
<th>Standard deviation (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Product</td>
<td>1.910</td>
<td>0.0045</td>
<td>4.8</td>
</tr>
<tr>
<td>Test</td>
<td>0.506</td>
<td>0.0267</td>
<td>4.0</td>
</tr>
<tr>
<td>Heat Flow</td>
<td>1.239</td>
<td>0.0117</td>
<td>4.8</td>
</tr>
<tr>
<td>Heat Return</td>
<td>1.435</td>
<td>0.0155</td>
<td>14.8</td>
</tr>
<tr>
<td>Sleeve</td>
<td>2.897</td>
<td>0.0050</td>
<td>11.0</td>
</tr>
</tbody>
</table>

Table 6.2: Constant ‘c’, ‘d’ and standard deviations of each pipe calculated by correlating the CFD data with Equation 6.6
The results from fitting Equation 6.7 to the data show a large reduction in the standard deviations for all the pipe surfaces. It can be argued that Equation 6.7 takes into account the complex thermal interactions of the pipe with the other pipes inside the bundle. However, it seems probable that the neural network approach described in Chapter 8 is a better way of correlating the data since it is of a generic form which would automatically take account of the two heat transfer processes, namely the convection and radiation heat transfer without assuming a correlation form appropriate only to one of them. It is also important to note that, better fitting would likely be acquired if the convective heat flows (Watt) can be extracted from the CFD calculated overall heat flows (Watt) which takes into account both convective and radiative heat flows.
### Table 6.3: Nusselt numbers and Rayleigh numbers for all the pipe surfaces of each CFD run

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>33.56</td>
<td>44.99</td>
<td>78.32</td>
<td>32.22</td>
<td>10.00</td>
<td>93.21</td>
<td>77.61</td>
<td>55.62</td>
<td>42.99</td>
<td>358.17</td>
<td>3.39E+07</td>
<td>1.52E+07</td>
<td>5.15E+06</td>
<td>6.60E+05</td>
<td>6.37E+08</td>
</tr>
<tr>
<td>2</td>
<td>43.70</td>
<td>26.28</td>
<td>60.76</td>
<td>37.51</td>
<td>10.00</td>
<td>133.29</td>
<td>46.07</td>
<td>51.01</td>
<td>48.56</td>
<td>368.53</td>
<td>3.12E+07</td>
<td>1.18E+07</td>
<td>5.44E+06</td>
<td>7.41E+05</td>
<td>6.22E+08</td>
</tr>
<tr>
<td>3</td>
<td>44.92</td>
<td>18.41</td>
<td>58.70</td>
<td>38.51</td>
<td>10.00</td>
<td>135.26</td>
<td>58.26</td>
<td>47.50</td>
<td>51.24</td>
<td>367.74</td>
<td>3.37E+07</td>
<td>4.16E+06</td>
<td>5.36E+06</td>
<td>1.02E+06</td>
<td>5.82E+08</td>
</tr>
<tr>
<td>4</td>
<td>45.41</td>
<td>16.68</td>
<td>57.50</td>
<td>37.58</td>
<td>10.00</td>
<td>138.39</td>
<td>61.95</td>
<td>48.86</td>
<td>53.61</td>
<td>388.72</td>
<td>3.50E+07</td>
<td>2.25E+06</td>
<td>5.34E+06</td>
<td>1.32E+06</td>
<td>5.41E+08</td>
</tr>
<tr>
<td>5</td>
<td>48.72</td>
<td>17.95</td>
<td>56.54</td>
<td>39.09</td>
<td>10.00</td>
<td>140.70</td>
<td>61.65</td>
<td>48.71</td>
<td>53.48</td>
<td>396.98</td>
<td>3.27E+07</td>
<td>4.43E+06</td>
<td>5.47E+06</td>
<td>1.43E+06</td>
<td>5.38E+08</td>
</tr>
<tr>
<td>6</td>
<td>51.84</td>
<td>19.52</td>
<td>55.60</td>
<td>40.82</td>
<td>10.00</td>
<td>142.96</td>
<td>61.98</td>
<td>48.65</td>
<td>53.71</td>
<td>403.43</td>
<td>3.13E+07</td>
<td>5.11E+06</td>
<td>5.72E+06</td>
<td>1.56E+06</td>
<td>5.42E+08</td>
</tr>
<tr>
<td>7</td>
<td>42.70</td>
<td>16.01</td>
<td>65.25</td>
<td>37.72</td>
<td>10.00</td>
<td>133.26</td>
<td>62.22</td>
<td>48.85</td>
<td>50.31</td>
<td>367.75</td>
<td>2.63E+07</td>
<td>5.41E+06</td>
<td>6.15E+06</td>
<td>1.76E+06</td>
<td>5.36E+08</td>
</tr>
<tr>
<td>8</td>
<td>47.46</td>
<td>15.81</td>
<td>69.13</td>
<td>39.41</td>
<td>10.00</td>
<td>141.03</td>
<td>71.38</td>
<td>51.42</td>
<td>56.28</td>
<td>393.70</td>
<td>1.58E+07</td>
<td>5.93E+06</td>
<td>6.59E+06</td>
<td>1.76E+06</td>
<td>5.27E+08</td>
</tr>
<tr>
<td>9</td>
<td>49.79</td>
<td>14.85</td>
<td>65.93</td>
<td>33.56</td>
<td>10.00</td>
<td>137.46</td>
<td>69.74</td>
<td>49.71</td>
<td>55.06</td>
<td>390.79</td>
<td>7.58E+06</td>
<td>6.18E+06</td>
<td>7.13E+06</td>
<td>1.60E+06</td>
<td>4.88E+08</td>
</tr>
<tr>
<td>10</td>
<td>54.18</td>
<td>15.92</td>
<td>69.44</td>
<td>36.97</td>
<td>10.00</td>
<td>144.20</td>
<td>77.77</td>
<td>53.14</td>
<td>54.19</td>
<td>367.53</td>
<td>3.53E+06</td>
<td>6.22E+06</td>
<td>7.55E+06</td>
<td>1.26E+06</td>
<td>4.52E+08</td>
</tr>
<tr>
<td>11</td>
<td>51.80</td>
<td>16.86</td>
<td>72.03</td>
<td>42.71</td>
<td>10.00</td>
<td>140.41</td>
<td>67.38</td>
<td>50.46</td>
<td>51.63</td>
<td>389.02</td>
<td>1.42E+07</td>
<td>5.74E+06</td>
<td>7.83E+06</td>
<td>8.81E+05</td>
<td>4.22E+08</td>
</tr>
<tr>
<td>12</td>
<td>45.22</td>
<td>18.59</td>
<td>81.67</td>
<td>32.58</td>
<td>10.00</td>
<td>133.65</td>
<td>64.41</td>
<td>54.12</td>
<td>49.29</td>
<td>383.36</td>
<td>2.01E+07</td>
<td>4.60E+06</td>
<td>8.19E+06</td>
<td>3.58E+05</td>
<td>3.76E+08</td>
</tr>
<tr>
<td>13</td>
<td>44.92</td>
<td>14.26</td>
<td>93.59</td>
<td>20.00</td>
<td>10.00</td>
<td>125.47</td>
<td>64.94</td>
<td>56.57</td>
<td>27.53</td>
<td>414.81</td>
<td>2.35E+07</td>
<td>4.27E+06</td>
<td>8.47E+06</td>
<td>1.86E+06</td>
<td>3.40E+08</td>
</tr>
<tr>
<td>14</td>
<td>42.15</td>
<td>13.67</td>
<td>97.22</td>
<td>19.74</td>
<td>10.00</td>
<td>125.81</td>
<td>66.21</td>
<td>58.05</td>
<td>30.49</td>
<td>399.12</td>
<td>2.62E+07</td>
<td>4.81E+06</td>
<td>8.53E+06</td>
<td>4.28E+05</td>
<td>3.35E+08</td>
</tr>
<tr>
<td>15</td>
<td>52.36</td>
<td>19.20</td>
<td>77.72</td>
<td>47.61</td>
<td>10.00</td>
<td>143.92</td>
<td>72.64</td>
<td>53.44</td>
<td>55.21</td>
<td>403.44</td>
<td>2.79E+07</td>
<td>5.41E+06</td>
<td>9.19E+06</td>
<td>4.68E+05</td>
<td>3.48E+08</td>
</tr>
<tr>
<td>16</td>
<td>55.43</td>
<td>18.78</td>
<td>74.15</td>
<td>45.94</td>
<td>10.00</td>
<td>144.48</td>
<td>67.13</td>
<td>52.18</td>
<td>54.13</td>
<td>409.69</td>
<td>2.39E+07</td>
<td>5.03E+06</td>
<td>9.03E+06</td>
<td>2.05E+05</td>
<td>3.49E+08</td>
</tr>
<tr>
<td>17</td>
<td>60.68</td>
<td>16.72</td>
<td>68.81</td>
<td>37.10</td>
<td>10.00</td>
<td>147.36</td>
<td>77.51</td>
<td>52.90</td>
<td>56.03</td>
<td>394.77</td>
<td>1.46E+07</td>
<td>5.40E+06</td>
<td>8.46E+06</td>
<td>2.80E+05</td>
<td>3.82E+08</td>
</tr>
<tr>
<td>18</td>
<td>60.48</td>
<td>19.15</td>
<td>73.31</td>
<td>43.97</td>
<td>10.00</td>
<td>149.83</td>
<td>58.32</td>
<td>51.89</td>
<td>51.19</td>
<td>381.87</td>
<td>9.18E+05</td>
<td>6.24E+06</td>
<td>7.85E+06</td>
<td>6.37E+05</td>
<td>4.24E+08</td>
</tr>
<tr>
<td>19</td>
<td>55.71</td>
<td>20.36</td>
<td>78.88</td>
<td>47.26</td>
<td>10.00</td>
<td>144.44</td>
<td>66.92</td>
<td>52.93</td>
<td>53.76</td>
<td>408.99</td>
<td>1.15E+07</td>
<td>6.64E+06</td>
<td>7.36E+06</td>
<td>8.79E+05</td>
<td>4.63E+08</td>
</tr>
<tr>
<td>20</td>
<td>52.31</td>
<td>19.89</td>
<td>86.64</td>
<td>33.69</td>
<td>10.00</td>
<td>138.55</td>
<td>65.56</td>
<td>53.83</td>
<td>49.61</td>
<td>404.61</td>
<td>1.73E+07</td>
<td>6.93E+06</td>
<td>6.72E+06</td>
<td>1.01E+06</td>
<td>5.29E+08</td>
</tr>
</tbody>
</table>
CHAPTER 7
CFD Inclined & Vertical Bundle Analysis
(3D Model)

7.0 SUMMARY OF CHAPTER
This chapter focuses primarily on the analysis of the results for the three dimensional (3D) Computational Fluid Dynamics (CFD) simulations that were carried out for the inclined and vertical bundles. The structure of this chapter is relatively similar to that for the previous chapter (Chapter 6) on the analysis of the CFD simulation for the horizontal bundle. Firstly, Section 7.1 gives a brief description of the background to the work. In section 7.2, a general overview is given of the work on the 3D calculations for inclined and vertical bundles. A total of 1053 3D simulations were carried out, covering a wide range of surface temperatures for inclination angles ranging from 0 degree to 90 degree. Section 7.3 discusses the analysis that was carried out on the results of the CFD simulations for the inclined and vertical bundles; several case studies were chosen for detailed analysis. Section 7.3 is divided into 2 Sub-Sections. The first Sub-Section (7.3.1) focuses on the irregular pattern in predicted temperature distribution occurred at the end walls of the bundle that was discovered during the course of the work. The associated impact on the overall heat transfer calculation in the bundle is examined and a solution to eliminate the irregularities on the temperature distribution is also presented. The chapter continues with the second Sub-Section (7.3.2) that presents the effect on the heat transfer as the bundle inclination angle increases from zero (horizontal) to 90° (vertical). Finally, some concluding remarks are given in Section 7.4.
7.1 BACKGROUND ON 3D MODEL FOR INCLINED AND VERTICAL BUNDLE

CFD simulations of heat transfer for inclined and vertical bundle is the primary original in this project. As explained previously in Chapter 3, the bundle is 3m long and has 4 internal pipes which are the product, test, heat flow (heater) and heat return pipe respectively. These pipes are inserted into a 30” sleeve pipe and the space between the pipes inside the bundle is typically filled up by an inert gas. In the simulations described in this Chapter, nitrogen was used as it has also been used widely in the actual bundle application in the field. However, during the validation activities (Chapter 5) against the experimental work which was conducted in a parallel PhD project by Mr. Myo Thant, air was used so that the CFD model replicates the system used in the experiments. It should be noted that the difference between air and nitrogen is small and one can be confident that the (positive) validation exercise described in Chapter 5 is appropriate in ensuring the accuracy and reliability of the CFD results reported here.

For the inclined and vertical bundle case, the flow of the gas inside the bundle is three dimensional in comparison with horizontal bundle where the flow is two dimensional and relatively less complicated. Therefore, a three dimensional (3D) CFD model was developed for the CFD simulations on inclined and vertical bundles. The 3D model that was developed for the bundle is shown in Figure 7.1 below. A hexagonal mesh was generated for the bundle which has in total of approximately 330,000 nodes. The detailed CFD set up which includes the mesh, boundary conditions, physical model, fluid propertise and the solver control for the 3D bundle model was presented in Chapter 3 of this thesis.
7.2 GENERAL OVERVIEW OF THE CFD RESULTS

As with the horizontal bundle, a large number of steady state simulations were undertaken for bundle in the inclined and vertical orientations. In total, 1053 3D simulations were performed covering the required range of angles and surface temperature combinations. The results were gathered and compiled as a heat transfer database for the selected bundle geometry. Each simulation was completed within 1.5 hours with 8 processors running in parallel to achieve the convergence criteria with less than 3% domain imbalances. Compared to 2D simulations, the 3D simulations require a longer time to be completed due to the complexity of the heat transfer interactions between the pipes inside the bundle. Also, the simulation of the 3D bundle model required a larger total number of nodes and iterations needed in order to achieve convergence.
Figure 7.2 below shows the typical Root Mean Square (RMS) residuals plot for the mass and momentum equations extracted from the CFX Solver for the inclined bundle simulation. Even though the values fluctuate, the residuals are still acceptably low, ranging between $1 \times 10^{-6}$ and $1 \times 10^{-2}$.

![Residuals Plot](image)

**Figure 7:2: Root Mean Square (RMS) residual plot for mass and momentum equations**

### 7.3 ANALYSIS OF THE 3D BUNDLE CFD SIMULATIONS

Figure 7.3 shows results calculated for the case of the bundle inclined at $10^\circ$ with a typical set of surface temperatures (product pipe $20^\circ$C, test pipe $20^\circ$C, heat flow pipe $60^\circ$C, heat return pipes $55^\circ$C and sleeve pipe $5^\circ$C). Figure 7.3 (a) show the gas (nitrogen) temperature distributions in a plane normal to the axis of the bundle at the mid plane (1.5m from each end) and Figure 7.3 (c) shows the temperature distributions in a plane through the axis in the vertical direction. Figure 7.3 (b) shows a plot of gas velocity vectors at the mid-plane of the bundle. The results shown in Figure 7.4 are for the bundle in the vertical position showing respectively the mid-plane temperature distribution (Figure 7.4 (a)), the vertical temperature distribution (Figure 7.4 (c)) and the mid-plane distribution of gas velocity vectors (Figure 7.4 (b)). For the case shown in Figure 7.4, the surface temperatures were the same as those
specified above for the 10° inclined bundle. Generally, a surface that is hotter than the average temperature of the gas will lose heat and this is indicated by the positive heat flow. On the other hand, a surface that is cooler than the average temperature of the gas will absorb heat and possesses negative heat flow. The average temperature of the gas is calculated simply by averaging the total nodal temperatures inside the domain.

![Figure 7:3](image)

**Figure 7:3**: Gas temperature distribution and velocity plot of an inclined bundle

From the Figures above, it is observed that highest gas velocity can be found at the highest temperature region in the bundle, particularly in the narrow spaces between the pipes such as between the product and the test pipe and also around the circumference of the heater pipe. The figures also show that the hot gas occupies mainly the top section of the bundle and the relatively colder gas accumulates at the bottom part of the bundle as indicated by the blue-green regions.
A significant difference in cross-sectional temperature distribution is observed for the vertical case. For horizontal bundles (as discussed in Chapter 6) and for an inclined bundle (as shown in Figure 7.3), the movement of the gas can be seen clearly, especially between the pipes that have high temperature gradient such as the heat flow (heater) pipe and product pipe. However, for vertical pipe the heat seems to accumulate around the circumference of each pipe inside the bundle as clearly shown in Figure 7.4 (a) above. Figure 7.4 (c) shows the temperature distribution along the length of the bundle and clearly demonstrates that the hot gas fills up the top section of the bundle and the relatively colder gas remains at the bottom section. Further discussion on the effect of inclination angle on temperature distribution will be presented later in Section 7.3.2.
7.3.1 Irregularities in Temperature Distribution

Some irregular patterns in the temperature distributions at the end walls of the bundle were discovered during the course of the work. It is suggested that the cause was the radiative heat fluxes generated at the end walls of the bundle. In the simulation, emissivity was set at 0.9 for all the wall boundary conditions (product, test, heat flow, heat return, sleeve and end walls) and radiation was solved at every 50\textsuperscript{th} iteration. For the radiation, a Monte Carlo model was specified with 5,000 histories. According to CFX Help manual, for Monte Carlo Radiation model:

“A photon is selected from a photon source and tracked through the system until its weight falls below some minimum at which point it ‘dies.’ Each time the photon experiences an ‘event,’ a surface intersection, scattering or absorption for example, the physical quantities of interest are updated. This process generates a complete ‘history’ of that photon in the system.”

Hence, a large number of histories have to be generated in order to acquire a good estimate on the radiation for a particular system especially in a complex system like a bundle pipeline. Furthermore, solving for radiation requires a massive computational power and time. This factor makes it impractical to increase the number of histories and also to set the solver to calculate radiation at every interval in the simulation. It is suggested that it is the limited number of histories that has caused the irregular (speckled) pattern of temperature distribution at the end walls of the bundle.

A further study was carried out later on in order to validate the assumption. It was done by simply turning off the radiation at the end walls to observe the effect on the irregular
temperature distribution. In the further investigation, the emissivity at the end walls was changed from 0.9 at 0. Figure 7.5 (b) below clearly shows that the speckled type of temperature distribution was successfully eliminated. This investigation has confirmed that the irregular pattern was caused by the radiation at the end walls.

![Figure 7:5: Irregular pattern in gas temperature distribution at the end walls of the bundle](image)

Eventhough the exclusion of radiation does solve the irregular pattern in temperature distribution at the end walls, it is also important to make sure that it does not influence the overall heat transfer calculation in the bundle system. The heat flows of each of the fluid boundary wall (product, test, heat flow, heat return and sleeve) and also the average temperature of the gas were extracted from the simulation files for both cases; with and without radiation at the end walls for comparison. The results given are tabulated as shown in the Table 7.1 below. It clearly shows that, there is no change in the average gas temperature and the differences in heat flows for all the pipes for both cases are very minimal, indeed, negligible. This indicates that by making the end walls radiatively inactive, it does not give any significant impact to the heat transfer calculation of the bundle. This is largely due to the fact that the end walls are adiabatic,
which in theory it does not allow any heat to flow in and out of its surfaces to interact with the other pipes in the bundle.

<table>
<thead>
<tr>
<th>Pipe</th>
<th>Heat Flows (Watt)</th>
<th>Figure (a)</th>
<th>Figure (b)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Product</td>
<td>1363</td>
<td>1364</td>
<td></td>
</tr>
<tr>
<td>Test</td>
<td>-159</td>
<td>-160</td>
<td></td>
</tr>
<tr>
<td>Heat Flow</td>
<td>27</td>
<td>27</td>
<td></td>
</tr>
<tr>
<td>Heat Return</td>
<td>-5</td>
<td>-4</td>
<td></td>
</tr>
<tr>
<td>Sleeve</td>
<td>-1227</td>
<td>-1224</td>
<td></td>
</tr>
<tr>
<td>Average Gas Temperature</td>
<td>28</td>
<td>28</td>
<td></td>
</tr>
</tbody>
</table>

Table 7.1: Comparison of the heat flows and gas average temperature for cases with and without radiation at bundle end walls

7.3.2 Effect of Inclination Angle on Bundle Heat Transfer

For the analysis and discussions purposes of this investigation, two set of temperature combinations were chosen as given in the Table 7.2 below. The two cases were selected because they represent typical temperature ranges of a bundle in deepwater operations where wax deposition and hydrates formation are likely to occur. The temperature of the production fluid can be low enough to be close to the wax appearance temperature (WAT) of the product fluid due to existence of a low sleeve temperature (5°C) reflecting a low ambient temperature, typically 4 °C. The temperature of the heater is relatively high in order to warm up the product pipe to keep the temperature of the production fluid sufficiently above the critical cloud point or wax appearance temperature (WAT).
Table 7.2: Selected temperature combinations for the study on effect of inclination angle on heat transfer in the bundle

<table>
<thead>
<tr>
<th>Pipes</th>
<th>CASE 1</th>
<th>CASE 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Product</td>
<td>20 °C</td>
<td>10 °C</td>
</tr>
<tr>
<td>Test</td>
<td>20 °C</td>
<td>10 °C</td>
</tr>
<tr>
<td>Heat Flow</td>
<td>60 °C</td>
<td>40 °C</td>
</tr>
<tr>
<td>Heat Return</td>
<td>55 °C</td>
<td>35 °C</td>
</tr>
<tr>
<td>Sleeve</td>
<td>5 °C</td>
<td>5 °C</td>
</tr>
</tbody>
</table>

The analysis was carried out for the angles of 10, 20, 40, 60, 80 and 90 degree. Cross sectional temperature distributions at the mid-plane (1.5m from each end of the bundle) for each case study at different inclination angles were extracted from the CFD simulation results as shown by Figure 7.6 and Figure 7.8. Both figures also provide the average temperature of the gas and also the total heat flow for each of the pipes inside the bundle. As explained previously, the pipes with higher temperature than the average gas temperature will give out heat and indicated by positive heat flows (Watt); those with lower temperature than the average gas temperature will absorb heat as represented by negative heat flows (Watt).

As shown in Figure 7.6 and Figure 7.8, and as is generally the case in buoyancy driven flows, the hotter fluid (gas) which is relatively less dense, occupies the top section of the bundle whereas the colder gas fills up the bottom section of the bundle (blue to green region). This can be observed clearly in the results for the first three inclination angles (10, 20 and 40 degrees). As the bundle inclination increases, the influence of the gravity becomes stronger and the heat transfer inside the bundle is affected. This is reflected in the changes in the temperature distribution as indicated by the cross-
sectional temperature profiles shown in both figures for higher inclination angles, namely 60, 80 and 90 degrees.

As the inclination angle increases, the heat seems to accumulate around the circumference of individual pipes inside the bundle. Therefore, the colder region is observed mainly at the outermost layer which is around the circumference of the sleeve pipe where the temperature is the lowest amongst all the pipes in the bundle at 5 °C. The temperature increases towards the centre of the pipe particularly around the circumference of the heat flow (heat supply) pipe and heat return pipe. This is because both pipes have a high temperature in comparison with the other pipes in the bundle.

Figure 7:6: Cross-sectional temperature distributions at the mid-plane (1.5) of the bundle for various inclination angles for Case 1
In Case 1 (Figure 7.6), the average temperature of the gas for all inclination angles is approximately between 17-18 °C which is slightly lower than the temperature of the product (20 °C) and the test (20 °C) pipes. Therefore both pipes are losing heat to the surroundings as shown by the positive heat flows in Figure 7.6 above (Case 1).

The general trend shows that as the bundle gets more inclined, the amount of heat loss from the product pipe increases as shown by the graph in Figure 7.7. The increase in heat loss from the product pipe is predicted due to the fact that as the bundle inclination angle increases, the influence of the gravity becomes stronger; therefore the heat from the heater (heat flow pipe) tends to accumulate around its circumference as can be clearly observed from Figure 7.6 for the 90 degree inclination angle. Hence, the heater becomes less efficient and the product pipe releases more heat to its surroundings. The additional heat loss from the product pipe goes mainly to the sleeve because the sleeve is in close proximity to the product pipe and has a much lower temperature (5 °C) than that of the product pipe (20°C), thus giving a large temperature difference to drive the heat flow.

![Graph showing increase in heat flow of product pipe as the inclination angle increases for Case 1](image-url)

Figure 7.7: Increase in heat flow of product pipe as the inclination angle increase for Case 1
As for Case 2 (Figure 7.8), the temperature of the product and test are at the lowest end of the range which is at 10 °C and the CFD calculation shows that the average temperature of the gas is approximately at 11 °C. Therefore it is expected that both the product and the test pipe should absorb heat as indicated by the negative heat flow in the Figure 7.8 below. Conversely, the hotter pipes (heat flow and heat return) are giving out heat to the surroundings in order to achieve thermal equilibrium.

A similar pattern of temperature distribution was also observed where, as the inclination angle increases, the influence of gravity forces is getting stronger. Hence, it makes the heater pipe less efficient. Therefore more heat from the heater pipe tends to gather around its circumference as shown by Figure 7.8 below. The average temperature of the gas is low (represented by greenish region), but a bright red ring region around the circumference of the heater and heat return pipes indicates that these regions are ones of high temperature.
Figure 7:8: Cross-sectional temperature distributions at the mid-plane (1.5) of the bundle for various inclination angles for Case 2

The graph plotted in Figure 7.9 below shows clearly that as the inclination angle of the bundle increases, less heat (less negative heat flow) is absorbed by the product pipe. In other words, the product pipe has to give out more heat particularly to the sleeve pipe as the bundle gets more inclined. In this case, the temperature difference between the product (10 °C) and the sleeve pipe (5 °C) is 5 °C which is higher than the temperature difference between the product (10 °C) and the average temperature of the gas (11 °C) (approximately 1 °C). Therefore as the bundle gets more inclined, the product has give out more heat in order to warm up the sleeve pipe until equilibrium is achieved.
Figure 7.9: Increase in heat flow of product pipe as the inclination angle increase for Case 2

Another aspect that was considered in this analysis is the effect of the inclination angles on the sleeve pipe. The sleeve is always absorbing heat in this study due to the fact that it has the lowest temperature amongst other pipes inside the bundle for all temperature combinations. It is important to know the amount of heat being absorbed by the sleeve pipe because it represents the total amount of heat loss from the bundle to the surroundings (sea water). The graphs in Figure 7.10 below show that as the inclination angle increases, the sleeve pipe absorbs more heat for both cases (mostly from heat contributed by the product pipe). In other words, there is more heat loss to the surroundings.
Figure 7.10: Heat flow (Watt) for sleeve pipe at various inclination angles for Case 1 (a) and Case 2 (b)

Figure 7.12 below gives better representation on the temperature distribution over the length of the bundle at different inclination angles. It shows clearly that in a steady state condition, as the inclination angle increases, more hot gas can be found at the top part of the bundle. This is due to the fact that in a buoyancy driven flow, as the temperature of the gas increases, it becomes less dense. Hence, the hot gas will move up to the top of the pipe while the colder gas will remain at the bottom section of the pipe.

The temperature distribution shown in Figure 7.11 (a) is the temperature for the gas that fills up the space between the pipes inside the bundle. However, within each 3m zone calculated, the surfaces are assumed to remain at constant temperature which implies that the temperature of the fluid that is flowing inside each of the pipes will remain nearly constant.
In the experiments, the bundle was inclined at several angles between the horizontal and the vertical. In Chapter 4, work is described whose aim was to investigate the effects of thermal stratification. It was established that, provided the water flow rate was increased significantly above the originally intended value, the effects of thermal stratification would be expected to be minimal. Though it is expected that thermal stratification in the experiments would be most serious for the horizontal bundle orientation, a series of calculations was carried out for the other extreme case – namely when the bundle is vertical. The surface temperatures for these calculations were chosen to be the same as those used in the studies reported in Chapter 4 as follows:

- Product : 65 °C
- Test : 20 °C
- Heat flow : 30 °C
- Heat return : 25 °C
- Sleeve : 17 °C

The tube-side flow rate chosen for these calculations was 40 litre/min (i.e. above the level at which thermal stratification would be expected to be significant even in a horizontal bundle). The results from these calculations are shown in Figure 7.11 where product side fluid temperatures are plotted over a large scale (Figure 7.13(a)) and over a smaller range (Figure 7.11(b)). As can be observed from Figure 7.11, the fluid temperature varies from 64.2 (minimum) to 65 °C (inlet temperature) while the calculated average temperature of the outlet is approximately at 64.4 °C. It will be seen, therefore, that the effects of tube side fluid temperature variation are, as expected, minimal and this further establishes the validity of the experiments.
This study also indicates that in general, the average temperature of the gas at the bottom pipe is relatively much lower than at the top of the bundle. Therefore the temperature gradient between the pipes especially the product pipe and the gas will be really high which will drive the heat to flow out from the pipe in order to achieve equilibrium as can be observed from Figure 7.11 (a) below. Hence, the temperature at the bottom part of the product pipe will drop quite significantly and might be going down to the wax appearance temperature (WAT) or cloud point of the product fluid which will induce the wax deposition and hydrates formation activity. Therefore it is important to ensure that the fluid temperature does not drop dramatically until reaching the WAT or the cloud point of the fluid at any part of the bundle along the overall length.

(a) ![Temperature variation](image1)

(b) ![Temperature variation](image2)

Figure 7.11: The temperature variation of product pipe is almost at a constant value over the length and around the perimeter.
7.3.3 Concluding Remarks

CFD calculations on heat transfer for inclined and vertical bundles show that the product tube loses more heat while the sleeve absorbs more heat (i.e, there is a greater heat loss to the surroundings) as the bundle inclination angle increases from horizontal to vertical. In other words, the heater pipe becomes less efficient as the bundle inclination increases. This does not imply that bundle active heating is incapable of maintaining the product temperature at a sufficiently high level to prevent solids formation and deposition. The critical consideration at the design stage is to ensure that the heater pipe temperature is high enough to warm up the product pipe and to keep the product fluid temperature above the WAT throughout the whole length of the bundle. This would include inclined and vertical sections of the bundle where the heat transfer from the heating pipe to the product pipe is less efficient. Subject to these considerations, it can be seen that bundle active heating is an effective method for thermal management of pipelines, including riser sections. Active heating is fundamentally preferable to insulation where the product fluid can cool without heat loss due to Joule-Thompson cooling (as discussed in Chapter 5).
Figure 7.12: Temperature profile along the 3m bundle length for different angle of inclinations for Case 1 (top) and Case 2 (bottom)
CHAPTER 8

Artificial Neural Network (ANN) Prediction of Bundle Heat Transfer

8.0 SUMMARY OF CHAPTER

In order to use the data sets generated by the CFD methodology in the design of bundle systems, a means is required for the accurate interpolation of the data so as to predict local heat transfer rates at each point along the system. This is a complex challenge and the best way of meeting this challenge seems to be the use of neural network analysis. The thermal behaviour of the bundle is complicated since there is conjugated natural convection and radiation in the bundle which makes it very difficult to fit the heat transfer data using conventional methods (for instance using the standard type of natural convection correlations). Employing artificial neural networks (ANN) seems to be the most viable approach due to its capability to adjust its parameters in order to be trained to understand, correlate and predict the outcomes based on given input-output datasets. ANN’s can empirically model any deterministic function with any number of dimensions given enough infrastructures (neurons and layers) and training examples.

The first section in this chapter (Section 8.1) gives a brief introduction on ANN. After that, in Section 8.2, previous work on neural network predictions for heat transfer is discussed. A general overview of neural networks covering the fundamental building block of a simple neuron and the commonly used transfer functions, learning methods and algorithms is given in Section 8.3.
Chapter 8: NEURAL NETWORK PREDICTION OF BUNDLE HEAT TRANSFER

Then in Section 8.4, the focus is narrowed down to the application of ANN’s in this research work to predict the bundle thermal performance parameters based on the heat transfer database that has been generated from the CFD model. A multi-layered feed-forward neural network with a back-propagation training algorithm was used for the bundle heat transfer analysis. The results from the ANN analysis for the bundle heat transfer were presented and analysed. One of the important aspects in this neural network analysis is incorporating the neural network function together with the weight and bias matrices into a simple neural network solver. The solver was developed in VBA code and can be run through an excel file. The excel solver has the capability to efficiently and quickly provide prediction of heat flows for all the surfaces in the bundle for any given surface temperature combination and bundle inclination angle. In addition, the solver also functions as a look-up table for the heat transfer database for the selected bundle configuration. Several screenshots from the neural network excel solver are presented in this chapter. Finally, in Section 8.5, the conclusions from the neural network analysis are presented.
8.1 INTRODUCTION TO NEURAL NETWORKS

Conventionally, it is quite common to represent a set of experimental data with correlating equations. Typically this can be done by implementing traditional regression analysis. However for a very complicated and multidimensional system, such as the conjugated natural convection/radiative heat transfer encountered in the present study, it is very difficult to fit the data by conventional means. Therefore it is beneficial to have a system that is capable of developing a model without necessarily requiring explicit formulation of the possible relationships that may exist between the parameters involved in the data. It is also important that the system is adaptive and can be trained in both its structure and the value of the fitting parameters. Artificial Neural Networks (ANN’s) are well suited for this purpose.

An artificial neural network (ANN) is a computational model which is inspired by the biological nervous system in the human body. In general, an artificial neural network consists of a group of interconnected simple processing elements which operate in parallel. Its capability to adapt with the changes in the external and internal information that flows through the network makes it capable of being trained to perform complex functions such as non-linear statistical data modelling. In other words, neural networks can be trained empirically so that a particular set of inputs will lead to a specific target output.

The main objective of undertaking the CFD simulations for the selected bundle was to establish a heat transfer database. The database consists of heat transfer parameters such as heat fluxes and heat transfer coefficients for different temperature combinations at different inclination angles. The database also includes some other information such as the average and minimum temperature of the gas (air in the experiments or nitrogen in the application) that fills up the space between the pipes in the bundle. In total 2,736 data sets for heat fluxes and heat transfer
coefficients, covering a wide range of surface temperature combinations and bundle inclination angles, were fitted by the ANN.

The neural network was trained so that a particular set of inputs leads to a specific target output. In this case, the input is the temperature combinations (°C) as well as the angle of inclination and the output would be the heat flows (W/m), heat transfer coefficient (W/m².K) and average gas bulk temperature (°C). Then the results from the neural network prediction were compared against the CFD generated data.

8.2 NEURAL NETWORK APPLICATIONS IN HEAT TRANSFER ANALYSIS
Thibault and Grandjean (1990) employed neural network analysis for three heat transfer case studies. They used a feed-forward network and compared the back-propagation method and the quasi-Newton learning algorithm. The first example studied was a lookup table for a thermocouple, the second case is a series of correlations between Nusslet and Rayleigh numbers for the free convection around horizontal smooth cylinders and the third case was on the problem of natural convection along slender vertical cylinders with variable surface heat flux.

Thibault and Grandjean (1990) emphasized that out of the two learning algorithms used in the investigation, the quasi-Newton method was by far the most reliable and the most accurate optimisation method. Its accuracy is more than two orders of magnitude greater than that of traditional back-propagation. In theory, back-propagation is also capable of producing similar results provided the appropriate set of initial random weights is given. Furthermore, back-propagation is the most common method employed because it can be used safely for systems with binary outputs. Its main disadvantage is the slow convergence rate. In the second example investigated by Thibault and Grandjean (1990), the correlation between Nusselt number and
Rayleigh numbers for a horizontal smooth cylinder was investigated. Morgan (1975) correlated such data using five traditional equations as follows:

\[
Nu = 0.675Ra^{0.68} \quad 10^{-10} \leq Ra \leq 10^{-2} \\
Nu = 1.02Ra^{0.148} \quad 10^{-2} \leq Ra \leq 10^2 \\
Nu = 0.850Ra^{0.188} \quad 10^2 \leq Ra \leq 10^4 \\
Nu = 0.480Ra^{0.250} \quad 10^4 \leq Ra \leq 10^7 \\
Nu = 0.125Ra^{0.333} \quad 10^7 \leq Ra \leq 10^{12} 
\]

Equation 8.1

The question addressed by Thibault and Grandjean (1990) was whether these 5 equations could be replaced by a single neural network. Their results are shown in Figure 8.1 below which shows a plot of Nu as a function of Ra, comparing the predictions of Equations 8.1 with a neural network containing five hidden neurons.

Figure 8.1: Plot of the Nusselt number as a function of Rayleigh number for the free convection around smooth horizontal cylinders (Thibault and Grandjean, 1990)
In this case, Thibault and Grandjean (1990) emphasized the importance of performing a logarithmic transformation of the input and output vectors to find an appropriate relationship between the two variables.

The third case reported by Thibault and Grandjean (1990) was concerned with the use of a neural network for correlating the average and the local Nusselt numbers as a function of the Prandtl number (Pr), the exponent of the power law variation of the surface heat flux and the surface curvature to describe the natural convection along slender vertical cylinders with variable heat fluxes. Thibault and Grandjean (1990) compared the results against the correlation equations proposed by Heckel et al (1989) as illustrated in the in Figure 8.2 below.

Figure 8.2: Plot of the average and the local Nusselt number prediction errors for the natural convection along slender vertical cylinders with variable surface heat flux (Thibault and Grandjean, 1990)
In the third case study, Thibault and Grandjean (1990), also showed that by increasing the number of hidden neurons, more accurate predictions were obtained as shown in Figure 8.2 above. Finally, Thibault and Grandjean (1990) concluded that, based on the results obtained, the neural network methodology can be used efficiently to model and correlate complex heat transfer data. Thibault and Grandjean (1990) also emphasized that the main advantage of using a neural network is to remove the burden of finding an appropriate model structure to fit experimental data or to find a useful regression equation.

Jambunathan et al (1995) reported the results of using ANN to model one-dimensional transient heat conduction for liquid crystal thermography (LCT). Typically, LCT combined with transient conduction analysis is used to deduce local values of convective heat transfer coefficient. A feed forward neural network model trained with a back-propagation algorithm was successfully trained to predict the convective heat transfer coefficient at a point in a duct which is heated by a flow of hot air. The ANN’s were trained on a broad range of initial temperatures and other parameters to cover reasonable ranges of real life transient experiments. Jambunathan et al reported that accuracies up to 2.7% have been obtained and it is also important to note that the speed of learning increases directly with the number of units in the first hidden layer.

Sreekanth et al (1999) used artificial neural networks to tackle the inverse heat conduction problem specifically for the determination of the surface heat transfer coefficient at a liquid-solid interface using temperature profile information from within the solid. The concept was tested with two geometric shapes, namely spherical and finite cylindrical solids. Sreekanth et al (1999) emphasized that the concept proposed can also be extended to other geometries. The investigation showed that the calculated heat transfer coefficients using an ANN model were in agreement (<3% error) with those calculated using conventional numerical/analytical techniques over a range of experimental conditions.
Ben-Nakhi et al (2007) described the use of neural networks as a tool for inter-model comparison with CFD results for cases in which CFD produces different solutions for different discretization schemes; such cases are referred to as *ill-posed* problems. A validated CFD code was used to generate a database that covered the range of Rayleigh number from $10^4$ to $4.7 \times 10^6$. A neural network was trained on a database generated by numerically-stable CFD analysis to predict flow variables for the ill-posed cases, thereby giving confidence in the use of steady-state CFD results for these cases. The robustness of the trained ANN’s was tested by applying them to a number of “production” data sets that the networks had never “seen” before. The ANN’s were also used to verify the CFD solutions in cases of higher Rayleigh number ranges for which CFD simulations produced different solutions for different discretization schemes. Three types of neural networks were evaluated and parametric studies were conducted to optimize network designs for the best predictions of the flow variables.

Ben-Nakhi et al (2007) emphasized the success of the ANN’s to accurately predict free convection in partitioned enclosures as significant because these ANN’s can be used for lending support to the authenticity of CFD solutions, especially when there are multiple solutions for a specific problem. Ben-Nakhi et al (2007) mentioned that the ANN’s, when designed and trained properly, encompass the associated physics of the problem since they are trained with CFD results that represent physical processes governing the heat and mass transfer.
8.3 GENERAL OVERVIEW OF NEURAL NETWORKS

8.3.1 Architecture of an Artificial Neural Network (ANN)

Basic Structure of an Artificial Neural Network

The most basic neural network component is a single output neuron or processing unit as shown in Figure.

This neuron maps its input \( p \) (a real-valued scalar) to an output value \( a \) (a real-valued scalar). Where \( w \) is a real-valued weight, \( w \cdot p \) is the product, and \( b \) is the 'bias', a constant term that does not depend on any input value. Intermediate value \( n \) is processed by transfer function \( f \) to produce the scalar output \( a \). If the transfer function is of the threshold type, the neuron acts as a linear classifier, whereas a continuous type transfer function (sigmoid) can result in a continuous output \( a \) for regression calculations.

A layer of Neurons

Fundamentally, two or more of the neurons described previously can be combined in a layer to map input vectors to output vectors for classification or regression. A single layered network with \( R \) input elements and \( S \) output neurons is illustrated below.
Chapter 8: NEURAL NETWORK PREDICTION OF BUNDLE HEAT TRANSFER

Figure 8.4: A schematic diagram showing the structure of a layer of neurons

The single neuron layer outputs form a vector of \( \mathbf{a} \) expressed as the function given at the bottom of the Figure 8.4 above. Single layered neural networks are only capable of learning linearly separable patterns.

**Multiple Layers of Neurons**

An ANN can have several layers, with the output from one layer feeding forward into the next layer; this is called a feedforward neural network. In this type of ANN, information only flows forwards from the input nodes, through the hidden nodes to the output nodes. Each layer has its own weight matrix \( \mathbf{W} \), bias vector \( \mathbf{b} \) and output vector \( \mathbf{a} \). This is shown in Figure 8.5 below.
Chapter 8: NEURAL NETWORK PREDICTION OF BUNDLE HEAT TRANSFER

Figure 8.5: A schematic diagram showing multiple layers of neurons (MATLAB Neural Network Manual)

The figure above can be simplified as follows.

Figure 8.6: Simplified three layer network (MATLAB Neural Network Manual)

Multiple-layered networks are a very powerful mechanism to obtain a functional relationship between complex variables. Generally, when the number of neurons and layers of a neural network increases, their collective complexity increases greatly. According to the Universal Approximation Theorem for neural networks, every continuous function that maps input intervals of real numbers to output intervals of real numbers can be approximated arbitrarily closely with a feedforward ANN with one hidden layer; containing a finite number of neurons. It is this ability of ANN’s to model functions of multivariate multidimensional data without analytical understanding of the underlying
principles that make them a very powerful mathematical tool for very complex situations such as heat transfer inside a bundle pipeline system.

8.3.2 Transfer Functions

As described previously, in most neural networks, an input vector $p$ is multiplied by a weight matrix $W$, and then added to a bias $b$. Then it will be manipulated by a transfer function $f$ to give the output $a$. This can be written as $a = f(Wp + b)$.

Four common transfer functions are shown in Figure 8.7 below.

![Transfer Functions Diagram](image)

Figure 8.7: Plots showing some transfer functions used in ANN: hardlim, purelin, tansig and radbas

The different transfer functions allow the neurons output to exhibit certain behaviours desired in the network. The *hardlim* transfer function is used when it is required that the neuron either recognises a pattern or does not, with no degree of recognition in between. The *purelin* transfer function is used when the output of the neuron is required to be in a
range that can be above 1 or below –1. The \textit{tansig} transfer function is used when the output of a neuron is required to be bounded between –1 and 1. The \textit{radbas} transfer function is typically used when input vectors are compared to a stored set of vectors with a perfect match resulting in a vector distance of zero, giving the highest output of one. There are other transfer functions that can be used in the design of ANNs; these are detailed in “Neural Network Toolbox 7 User’s Guide” (Beale et al., 2010).

8.3.3 Learning Methods

There are several learning paradigms that are always being employed in ANN’s. The common ones are supervised learning and unsupervised learning.

\textbf{Supervised Learning}

Supervised learning is an iterative process where the aim is to optimise the weights and biases of the ANN so that the outputs to inputs match prescribed target output values. Typically the training data consists of a set of many training examples covering a wide range of input vectors. In supervised learning, each example is a pair which consist of an input (typically a vector) and a desired output value. The supervised learning algorithm will analyse the training data and adjust the weight and bias matrices iteratively until the desired output is achieved. Generally, the more representative the training examples are of a particular data set, the better the ANN will perform. Figure 8.8 below illustrates the work flow of a supervised learning algorithm.
This learning method employed for the current work used a data set which has temperatures and angle of inclination as the input data with the respective heat transfer rates as the desired output.

**Unsupervised learning**

Unsupervised learning is a more independent form of learning than supervised learning, which does not require associative pairing between inputs and desired outputs. Basically it uses no external teacher and learning is based on correlating input data only. This form of learning is also referred to as *self-organisation*, in which the learning algorithm organises the input data into clusters and classifies new data into these learnt groups. The degree of fit of an input vector to established classification gives a measure of recognition and hence novelty detection.

### 8.4 NEURAL NETWORK ANALYSIS FOR BUNDLE HEAT TRANSFER

As discussed previously the heat transfer inside the bundle used for this work is very complex in nature. The heat transfer mechanisms are mainly natural convection and radiation. Detailed data and correlations for natural convective heat transfer with radiation in an enclosed circular pipe are very limited due to the highly complicated thermal interactions between the pipes inside the
bundle. An artificial neural network is a powerful analytical model that trainable in both structure and its parameter values. This ability makes ANN the best method to learn and correlate complex relationship of a bundle heat transfer problem.

Generally, the more the number of neurons and layers employed in an ANN, its potential complexity increases which makes it more able to model a set of empirical data. The heat transfer database for the bundle pipe used in the present work was generated using CFD. The database consist of temperature combinations and inclination angles with their respective heat flows.

Multi layer feedforward ANNs which are also known as multi-layered perceptrons are the most commonly used ANN. The general schematic of a feedforward ANN is shown in Figure below.

![Feedforward ANN](image)

Figure 8:9: A schematic diagram showing a feedforward ANN with one “hidden” layer

A multi-layered feedforward network is usually employed for the modelling of physical systems (Thibault and Grandjean, 1990). A multi-layered feed-forward network consisting of one or more layers of neurons (known as “hidden” layers) which, are followed by an output layer (see Figure 8.9 above) was used in the present work. The network modeled 5 outputs which are the heat transfer rate with respect to 6 inputs which are the 5 temperature combinations for all the pipes in the bundle and also the inclination angle. For the heat transfer bundle analysis, two
hidden layers of neurons were used with 10 and 12 “tansig” neurons respectively. The input to
the network is vector \( \mathbf{p} \), which was processed in the first hidden layer where it was be multiplied
by weight matrix \( \mathbf{IW} \) and a bias of \( \mathbf{b}_1 \) was added. This sum vector is then processed by a transfer
function of tansigmoid (or tansig) form to produce a vector \( \mathbf{a}_1 \) which will be passed to the
subsequent hidden layers of the output layer as shown in Figure 8.9 above. The output layer will
also multiply the results by a weight \( \mathbf{LW} \) and a bias, \( \mathbf{b}_2 \) is added before applying a transfer
function “purelin” for this case. “Purelin” provides a linear transfer function which is commonly
used in the final layer of multilayer networks.

The equations for the hidden layer output \( \mathbf{a}_1 \) and the output of the ANN \( y \) are given as follows.

\[
\begin{align*}
\mathbf{a}_1 &= \text{tansig} (\mathbf{IW}.\mathbf{p}_1 + \mathbf{b}_1) \\
y &= \text{purelin} (\mathbf{LW}.\mathbf{a}_1 + \mathbf{b}_2)
\end{align*}
\]

Equation 8.8.2

Equation 8.8.3

By adjusting the weight matrices \( \mathbf{IW} \) and \( \mathbf{LW} \) and the biases \( \mathbf{b}_1 \) and \( \mathbf{b}_2 \), the network can be
trained to model a set of input and output relationships from a database.

After the architecture of the feed-forward network has been set up, it can be trained for function
approximation or pattern classification. The learning process implies that the processing element
somehow changes its input/output behaviour in response to the environment. The learning
process thereby consists in determining the weight matrices \( \mathbf{IW} \) and \( \mathbf{LW} \) that produce the best fit
to the predicted outputs over the entire training dataset. In order to train a network with a
supervised learning method, a dataset of input and output vectors are required. The procedure for
batch training of the feedforward network is as follows (Rhys, 2005):

- The weights and biases of the network are first initialised to random values.
- The input dataset is processed once (this stage is called the epoch) and the resulting
  outputs are recorded.
• The outputs are compared to the target output dataset using a performance function, usually the mean squared error (mse).

• The weights and biases of the network are adjusted with the aim of improving its performance by minimising the network performance function mse.

• Steps 2 until 4 are repeated iteratively until a required mse is achieved or a maximum number of epochs has occurred without convergence.

There are a few learning algorithms available for ANN’s and back-propagation is the most versatile for a feed-forward multi-layered network. Back-propagation learning algorithms operate by redistributing or back-propagating the output errors of the network by appropriately modifying the weight matrices (Thibault and Grandjean, 1990). The back-propagation method was employed for the heat transfer bundle analysis. Back-propagation is based on the premise that the network has differentiable output functions of both the inputs and the weights and biases. If a performance function is chosen that is a differentiable function of the outputs, such as mse, then this performance function is itself a differentiable function of the weights and biases. The derivatives of the performance function with respect to the weights can then be evaluated to form an error surface, which can be used to find weight values that minimise the performance function. Either a gradient descent method or a more powerful optimisation technique can be used to adjust the weights to minimise the error of the network.

According to MATLAB Neural Network Toolbox, the Levenberg–Marquardt optimisation technique provides the fastest back-propagation training of a multi-layered feedforward network and is always recommended as the first choice of supervised learning algorithms even though it requires more memory.
The network was trained on a total of 2736 vectors from the thermal performance database generated from CFD simulations. As mentioned previously, the input vectors are the temperatures of all the surfaces inside the bundle and also the inclination angle which ranges from horizontal to inclined and vertical bundles. There are 5 outputs which are the respective heat flow rates for each of the surfaces in the bundle, namely the product, test, heat flow (heater) and the heat return pipes and also the sleeve pipe.

The Figure 8.10 below shows a screenshot taken from an on-going neural network analysis for heat transfer bundle using MATLAB.

![Figure 8:10: A typical screenshot of a neural network analysis running in MATLAB](image)

The following figure shows a screenshot after the training process for the heat transfer datasets for the bundle has completed. The simulation stopped after the maximum step (epoch) has been achieved. The results are reasonable with a small value of the final mean square error.
Figure 8:11: A typical screenshot after the network training process is completed and achieved convergence.

After the completion of the training process, clicking on “Performance” in the window shown in Figure 8.11 above, results in the provision of a plot of the training errors, validation errors and test errors as illustrated in Figure 8.12 below.

Figure 8:12: Performance plot after the network training process is completed for the bundle heat transfer datasets.

Figure 8.13 shows a comparison between the results obtained using CFD and the neural network generated results for heat flows at given pipe temperatures for all the 2,736 simulation datasets.
Chapter 8: NEURAL NETWORK PREDICTION OF BUNDLE HEAT TRANSFER

The top plot shows the heat flows for product pipe from the CFD calculations, the middle plots represents predictions from the output from the neural network and the bottom plot gives the error of the neural network results i.e the difference to the CFD results. From the error plot, the error between the neural network prediction and CFD calculation is approximately 4% which is very satisfactory.

![Figure 8:13: Plots showing a comparison between CFD results and neural network results](image)

Once the neural network training was completed, the Neural Network MATLAB Tool can be set to export the final weight and bias constants for the bundle heat transfer analysis in csv format (compatible with excel file). These values are then incorporated into a simple excel solver which also contains the neural network function. The neural network excel solver was jointly developed with Dr. Dan Rhys from University of Bristol who was acting as Research Assistant for this project.
Besides predicting the heat flows for each of the pipes in the bundle, for a given set of temperatures and inclinations using the neural network equations with the weights and biases, the simple excel solver can also work as a look-up table as illustrated in Figure 8.14 below. The red box in Figure 8.14 shows CFD results obtained using the solver as a look-up table. Also shown in Figure 8.14 are the values predicted by the solver from the neural network function incorporating weights and biases from the training process. In this example, the case number (simulation number) was keyed-in in the top row in the red box column. Once the case number is put in, the button “compare” has to be clicked and the neural network function will look-up in the heat transfer database that has been embedded in the solver to determine the temperature combination and associate inclination angle which are given in the “Inputs” column in the blue box. The “Outputs” column in the blue box provides the neural network predictions for the given temperature combinations and inclination angle. Below the blue box in Figure 8.14, the net heat flow per metre in of the bundle is shown in Watts. It is important to note that, for steady state condition, the net heat flow should be equal to, or as close as possible to zero.

Comparing the heat flows of each pipe between the neural network predictions (“Output” column in blue box section) and the column inside the red box section, the results are almost identical; the differences are very minimal.
Chapter 8: NEURAL NETWORK PREDICTION OF BUNDLE HEAT TRANSFER

Figure 8.14: Screenshot of the neural network excel solver which can function as a heat transfer look-up table.

The primary function of the neural network excel solver is to quickly predict the heat transfer rate for a given temperature combination and inclination angle. For this example, the focus is given to the “Input” column in the blue box section in the following figure. In the example presented in Figure 8.15 below, in order to test the robustness of the neural network prediction, a case that is new and never been “shown” to the neural network was put in. In this case, the 33 degree angle of inclination was chosen because it was never simulated in CFD. Hence, it does not exist in the database. After that, temperatures for each pipe inside the bundle including the sleeve pipe were defined. It is also important to note that, this solver provides a “live” predictions in the “Output” column as the user putting in the input values. In this example, the net heat flows is given at -1.17 Watts which is satisfactorily low; this proves the reliability of the neural network prediction.
Chapter 8: NEURAL NETWORK PREDICTION OF BUNDLE HEAT TRANSFER

8.5 CONCLUSION

Considering the complexity of the thermal relationship between the pipes inside the bundle in which the dominant heat transfer mechanisms are natural convection and radiation, it is very difficult to obtain an appropriate model structure to fit the data or to find an ideal regression equation. Based on the outcomes of the investigation carried out in this work, the artificial neural network (ANN) has proven its capability to efficiently correlate and predict a multivariate heat transfer data set generated by the CFD calculations. Another advantage of ANN’s is the speed that it produces the prediction is significantly faster than CFD simulations without compromising the accuracy and consistency of the outcomes. However, it is important to emphasize that the accuracy of an ANN very much depends on the accuracy of the data sets that are being fed to the ANN for training purposes. Therefore, the focus has to be on producing a correct and reliable database from the CFD model for the bundle pipe heat transfer analysis.
CHAPTER 9
Conclusions and Recommendations

9.0 SUMMARY OF CHAPTER

In what follows, Section 9.1 summarises the main conclusions from the studies and Section 9.2 presents recommendations for further work.
9.1 CONCLUSIONS

This study has focussed on a computational study of heat transfer in an actively heated tube bundle such as would be applied for flow assurance in a sub-sea pipeline. The objective of installation of such bundles is to ensure that the temperature of the production fluid is always above that leading to solids formation and deposition. Though the aim has been to develop a generally applicable methodology, a specific bundle geometry was selected as a development test bed. In a parallel project by Mr. Maung Maung Myo Thant, tests were conducted on the selected geometry and the results from these tests were used to validate the computational methods described in this thesis. The selected bundle consists of 4 internal pipes, namely the product, test, heat flow (heater) and heat return pipes. All the pipes are mounted in a 30 inch (762 mm) internal diameter sleeve pipe. In the present simulations and in the experiments, the space between the pipes inside the bundle is filled with air; in actual field applications, an inert gas such as nitrogen is always used. However, it was more convenient to use air rather than nitrogen; predictions for actual applications could be done taking account of the (small) difference between air and nitrogen.

Two CFD models have been developed for the bundle heat transfer analysis. The first one was a 2D model for the horizontal bundle and the second one was a 3D model for the inclined and vertical bundles. In a horizontal bundle the flow of the interstitial gas is two dimensional in a plane normal to the bundle axis and a 2D model is sufficient to carry out the heat transfer analysis. The use of a 2D model reduces the overall simulation time significantly. Both CFD models were validated against experimental work carried out in the parallel research project undertaken by Mr. Maung Maung Myo Thant.
Prior to the primary CFD simulation activities, a thermal stratification investigation was carried out. In the experimental work carried out by Mr. Myo Thant, the heat flows into or out of the surfaces of pipes in a simulated bundle were calculated. The primary objective of the computational thermal stratification investigation was to verify an important assumption used in the experimental work, namely that each of the surfaces in the bundle could be maintained at a nearly constant temperature by passing water streams of known temperature through the pipes and around the sleeve. There were two contradictory requirements in the experiments; thus, a high water flowrate is ideal to give good mixing and hence a nearly uniform surface temperature but, on the other hand, the water flow rate should be low enough to allow accurate measurement of the (small) difference in temperature between the inlet and the outlet of the water stream passing through each pipe and the sleeve. *Thermal stratification* can occur when relatively stagnant water zones exist on the inside of the pipes and these cool down to give lower surface temperatures (and hence lower heat transfer rates).

Thermal stratification can co-exist with small differences between inlet and outlet temperatures; however, these differences will be smaller than they would have been had the surface temperature been uniform. The investigation focused on the product pipe which was expected to have the greatest thermal stratification. It was, in fact, found that thermal stratification effects could occur at the water flowrates originally planned for the experiments and this necessitated an increase in these flow rates. At the higher flow rates, though the calculations showed that thermal stratification effects were minimal, the (already small) difference between the inlet and outlet temperature of the water streams was further reduced and improved instrumentation (namely platinum resistance thermometry) was necessary to achieve the required accuracy. Thus, the CFD investigation was important in establishing the validity and design of the experimental method.
Validation work was conducted to ensure the reliability and accuracy of the CFD predictions on bundle heat transfer. Several temperature combinations used in experimental work were selected for the validation activity. Exact temperatures and inclination angles were used to ensure the consistency between the experiment and CFD simulations. The surface heat flows were calculated and compared between the experiment and CFD simulations. The results show that there are some discrepancies both in sign and magnitude between the measured and predicted values; however, in general, the majority of the results show satisfactory agreement. Based on the results presented, it can be concluded that reasonable agreement was obtained between the CFD predictions and the experimental results. Therefore it can be concluded that the CFD model used in the bundle heat transfer analysis has been successfully validated.

In addition to the main validation activity between the experimental calculations and the CFD predictions, two other validation exercises were carried out. In the first of these, the 2D CFD model was validated against the 3D CFD model. As was stated above, the 2D model was used for the horizontal bundle due to the fact that the flow of the air in the the bundle is likely to be close to two dimensional. Generally, the results show that the bundle heat losses (W/m) for the 2D model show good agreement with the results for the 3D model, the difference being approximately 7 % on average. In conclusion, the validation study has proved that the 2D model has an acceptable level of accuracy for use in carrying out heat transfer simulations for the horizontal bundle. This allows a significant reduction in computational time to complete the simulations for the horizontal bundle case. The second validation activity was conducted by comparing the results from two different CFD software packages, namely ANSYS CFX (the main code used in this research project) and STAR-CCM+ by CD-Adapco. Similar bundle geometry, temperature combinations, boundary conditions, physical model and fluid properties were used for both. It can be concluded from the comparative study that both CFX and STAR-CCM+ are capable of being used to give reliable bundle heat transfer analyses. The agreement
between the two codes gives support to the view that the general CFD methodology is a viable one in addressing this problem.

The importance of radiation in the conjugate heat transfer that occurs inside a gas filled bundle pipe system was stressed in this work and radiation was always taken into account in calculating the net heat transfer rates. As an illustration of the importance of radiation, calculations were carried out by respectively including and excluding radiation from the model. The results show that (for given surface temperatures) the cross sectional distribution of gas temperature is essentially the same for both cases (with and without radiation). Since the gas is treated as being transparent, the radiation has no effect on the gas temperature which is essentially governed by the surface temperatures and the associated natural convection processes. However, the results also indicate that the radiation contributes strongly to the heat losses and gains from the various surfaces. It can be concluded from the study that radiation does play an important role in the heat transfer that occurs inside the bundle pipe. Eliminating the radiation from the CFD calculation, reduces the pipe heat flows which would cause a significant inaccuracy in bundle performance calculations.

A limited study was also undertaken to verify the effect of Joule-Thomson cooling in pipeline risers using OLGA, a multiphase simulation package. Fundamentally, the Joule-Thomson effect is associated with changes in temperature caused by pressure changes when the production fluid contains gas. This effect can cause significant temperature drops in the riser section of the flowlines; this may lead to the formation and deposition of solids which can restrict the flow of the production fluid. Two cases were prepared for the comparative study. One case is a bare pipe and the other case uses insulation. The study was carried out for various thickness of insulation to see the effect of the insulation on the Joule-Thomson cooling. From these limited OLGA simulations, it can be deduced that Joule-Thomson cooling is inevitable particularly at the riser
section of a pipeline system even with a perfect insulation. Therefore it is important to employ an alternative thermal management method such as active heating together with insulation to provide better thermal performance to keep the temperature of the pipeline outside the solids formation and deposition regions.

In total some 1,926 temperature combinations were simulated for the horizontal bundle case using the 2D model and 1,053 simulations were carried out with 3D model for the inclined and vertical bundle cases, covering a wide range of surface temperatures for inclination angles ranging from 0 degree to 90 degrees. The heat transfer parameters such as heat flows and heat transfer coefficients for each of the surfaces inside the bundle were obtained and gathered from both the 2D and 3D simulations as a heat transfer database. For the 3D bundle simulations, a study to observe the effect on the heat transfer as the bundle inclination angle increases from zero (horizontal) to 90° (vertical) was also conducted. The CFD calculations show that the product tube loses more heat while the sleeve absorbs more heat (i.e there is a greater heat loss to the surroundings) as the bundle inclination angle increases from horizontal to vertical. In order words, the heater pipe becomes less efficient as the bundle inclination increases.

In order to use the heat transfer data sets generated by the CFD methodology in the design of bundle systems, a means is required for the accurate interpolation of the data so as to predict local heat transfer rates at each point along the system. Since the thermal behaviour of the bundle is complicated due to conjugated natural convection and radiation in the bundle, it is very difficult to fit the heat transfer data using conventional methods. Therefore employing artificial neural networks (ANN) seems to be the most viable approach due to its capability to adjust its parameters in order to be trained to understand, correlate and predict the outcomes based on given input-output datasets. A neural network analysis was carried out on the heat transfer database. It was shown that the artificial neural network (ANN) is capable of efficiently
correlating and predicting the multivariate heat transfer data set generated by the CFD calculations. Another advantage of ANN is the speed that it produces the prediction is significantly faster than CFD simulations without compromising the accuracy and consistency of the outcomes. However, it is important to emphasize that the accuracy of an ANN very much dependent on the accuracy of the data sets that are being fed to ANN for training purposes.

9.2 RECOMMENDATIONS FOR FUTURE WORK

In the present CFD work, the end walls of the pipe bundle were not simulated. In CFD, the end walls were defined as adiabatic walls because in the actual pipe bundle used in the experimental work, there are spacers positioned at the end walls to hold the internal pipes (the product, test, heat supply and heat return) in their position. It is anticipated that the inclusion of end walls will influence the thermal interactions inside the bundle and this is an area where further investigation would be worthwhile.

The space between the pipes inside the bundle was filled with air. When the pipe surfaces were heated, radiation will occur during the heat transfer process. In the CFD simulations, a Monte Carlo model was used to calculate the radiative heat transfer between the pipe surfaces. It was specified that the emissivity of the metal surfaces at 0.9; the value of 0.9 was used based on the parametric study on emissivity that was carried out by Liu, et. al. (2006) in the previous TM3 project. The CFD calculations in the present work confirmed the significance of radiation. Therefore it is vital to obtain the correct emissivity in order to ensure the accuracy of the heat transfer calculation for the bundle. It is recommended that, in the future work, a radiometer be employed to accurately measure the emissivity of the metal surfaces in the bundle pipe.

One of the primary concerns in the methodology presented here is the large number of simulations required to produce a sufficiently extensive and reliable heat transfer database to be
fitted by the neural network procedure. With the computing capability used in the present work (8 processors) a 2D simulation (for the horizontal bundle) can be completed in approximately 30 minutes and a 3D simulation can be completed in around 2.5 hours. Thus, a very significant amount of time is required to complete the data sets. The time was restricted as far as possible by selecting temperature ranges which were typical of those where solids formation was a significant threat. However, ideally, a much larger temperature range should be covered in order to improve the fitting procedure. It is highly recommended that, for future work, the computational capability be extended; for instance, high performance computing (HPC) could be used to significantly speed up the simulation time and to allow the simulation of wider temperature ranges.
BIBLIOGRAPHY

A


B


D


E


F


G


P


R


S


T


W


Y


Z


APPENDICES

APPENDIX 1: BOUSSINESQ BUOYANCY MODEL

The present work is based on the application of the Boussinesq Buoyancy Model which is used within the ANSYS CFX Code. For completeness and future reference, the model is summarised here. A more complete description is given in the ANSYS CFX code Manual.

The Boussinesq buoyancy model assumes a constant density fluid, but accounts for buoyancy by applying a local gravitational body force throughout the fluid which is a linear function of fluid thermal expansivity $\beta$ and the local temperature difference with reference to a datum called the “Buoyancy Reference Temperature”. This reference temperature needs to be specified during setting up the bundle simulation (pre-solver). The reference temperature is an approximate value of the expected average temperature of the domain. For buoyancy calculations, a general source term is included in the momentum equation as follows,

$$S_{M,\text{buoy}} = (\rho - \rho_{\text{ref}}) g$$

(A1)

In the Boussinesq buoyancy model, the density difference is driven only by temperature variations. A constant reference density, $\rho_{\text{ref}}$, is used and the density difference is given by the following equation,

$$\rho - \rho_{\text{ref}} = -\rho_{\text{ref}} \beta (T - T_{\text{ref}})$$

(A2)

where $\beta$ is the thermal expansivity and $T_{\text{ref}}$ is the buoyancy reference temperature. $\beta$ is defined by the equation:

$$\beta = \left. -\frac{1}{\rho} \frac{\partial \rho}{\partial T} \right|_{\rho}$$

(A3)
APPENDIX 2: SHEAR STRESS TRANSPORT (SST) K-OMEGA TURBULENCE MODEL

An important class of turbulence models is that of “two-equation” models. Two-equation turbulence models invoke two additional transport equations to represent the turbulent properties of the flow. Use of such models allows account to be taken of history effects like convection and diffusion of turbulent energy. One of the transported variables is the kinetic energy, $k$ and the second transported variable varies depending on the type of the two-equation model used. Common choices for the second transported variable are the turbulent dissipation rate $\varepsilon$ and the specific dissipation $\omega$. These two variables are the variables that determine the scale of the turbulence (length-scale or time-scale) whereas the kinetic energy, $k$ determines the turbulence energy.

According to Menter (1994), the Shear Stress Transport (SST) turbulence model is a two-equation eddy-viscosity model which combines two most common turbulence models namely, $k-\varepsilon$ and $k-\omega$. Menter (1994) states that SST model is actually a modification to the definition of the eddy-viscosity in the baseline (BSL) model which takes into account the effect of the transport of the principal turbulent shear stress. In general, the baseline (BSL) model utilizes the original $k-\omega$ model of Wilcox (1988) in the inner region of the boundary layer and switches to the standard $k-\varepsilon$ model in the outer region and in free shear flows. It has a performance similar to the Wilcox model but avoids that model’s strong freestream sensitivity.

Fundamentally, the SST model is based on BSL model. Hence, it is important to also derive the BSL model in this section. The main aim of the model is to retain the robust and accurate formulation of the Wilcox $k-\omega$ model in the near wall region and to take advantage of the freestream independence of the $k-\varepsilon$ model in the outer part of the boundary layer. The
difference between the new formulation and the original $k - \omega$ model is that an additional cross-diffusion term appears in the $\omega$ equation and that the modelling constants are different as given in equations below. The left-hand side of the Lagrangian derivative is given in the following equations:

\[
\frac{D}{Dt} = \partial_t + u_i \partial / \partial x_i
\]  \hspace{1cm} (A4)

Original $k - \omega$ model:

\[
\frac{D\rho k}{Dt} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_{k1} \mu_i) \frac{\partial k}{\partial x_j} \right]
\]  \hspace{1cm} (A5)

\[
\frac{D\rho \omega}{Dt} = \gamma_1 \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta_1 \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_{\omega1} \mu_i) \frac{\partial \omega}{\partial x_j} \right]
\]  \hspace{1cm} (A6)

Transformed $k - \varepsilon$ model:

\[
\frac{D\rho k}{Dt} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_{k2} \mu_i) \frac{\partial k}{\partial x_j} \right]
\]  \hspace{1cm} (A7)

\[
\frac{D\rho \omega}{Dt} = \gamma_2 \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta_2 \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_{\omega2} \mu_i) \frac{\partial \omega}{\partial x_j} \right]
+ 2\rho \sigma_{\omega2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
\]  \hspace{1cm} (A8)

Now, Equation (A5) and Equation (A6) are multiplied by $F_i$ and Equation (A7) and Equation (A8) are multiplied by $(1-F_i)$ and the corresponding equations of each set are added together to give the new BSL model:

\[
\frac{D\rho k}{Dt} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_k \mu_i) \frac{\partial k}{\partial x_j} \right]
\]  \hspace{1cm} (A9)
\[
\frac{D \rho \omega}{Dt} = \frac{\gamma}{\nu_t} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \mu + \sigma_{\omega} \mu_t \right) \frac{\partial \omega}{\partial x_j} \right] \\
+ 2 \rho(1 - F_1) \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
\]

(A10)

where

\[
\tau_{ij} = \mu_t \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} - 2 \frac{\partial u_k}{\partial x_i} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij}
\]

(A11)

Let \( \varphi_1 \) represents any constant in the original model \((\sigma_{k1}, ...)\), \( \varphi_2 \) any constant in the transformed \( k - \varepsilon \) model \((\sigma_{k2}, ...)\) and \( \varphi \) the corresponding constant of the new model \((\sigma_k, ...)\), then the relation between them is:

\[
\varphi = F_1 \varphi_1 + (1 - F_1) \varphi_2
\]

(A12)

The constants of set 1 \((\sigma_1)\) are (Wilcox):

\[
\sigma_{k1} = 0.5, \quad \sigma_{\omega1} = 0.5, \quad \beta_1 = 0.0750
\]

\[
\beta^* = 0.09, \quad \kappa = 0.41, \quad \gamma_1 = \beta_1 / \beta^* - \sigma_{\omega1} \kappa^2 / \sqrt{\beta^*}
\]

(A13)

The constants of set 2 \((\sigma_2)\) are (standard \( k - \varepsilon \)):

\[
\sigma_{k2} = 1.0, \quad \sigma_{\omega2} = 0.856, \quad \beta_2 = 0.0828
\]

\[
\beta^* = 0.09, \quad \kappa = 0.41, \quad \gamma_2 = \beta_2 / \beta^* - \sigma_{\omega2} \kappa^2 / \sqrt{\beta^*}
\]

(A14)

With the following definitions:

\[
\nu_t = \frac{k}{\omega}
\]

(A15)

The function \( F_1 \) is designed to be unity in the near wall region (activating the original model) and zero away from the surface. The blending will take place in the wake region of the boundary layer.

\[
F_1 = \tanh (\arg_1^4)
\]

(A16)
where \( y \) is the distance to the next surface and \( CD_{k\omega} \) is the positive portion of the cross-diffusion term of Equation (A10).

\[
CD_{k\omega} = \max\left(2\rho\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, \frac{4\rho\sigma_{\omega 2} k}{CD_{k\omega} y^2}, 10^{-20}\right)
\]  

(A18)

The term \( \text{arg}_1 \) obviously goes to zero far enough away from solid surfaces because of the \( 1/y \) or \( 1/y^2 \) dependency in all three terms in Equation (A18).

The SST model is identical to the preceding formulation, except that the constants, \( \emptyset_1 \), have to be changed to:

Set 1 (SST inner):

\[
\sigma_{k1} = 0.85, \quad \sigma_{\omega 1} = 0.5, \quad \beta_1 = 0.0750, \quad a_1 = 0.31
\]

\[
\beta^* = 0.09, \quad \kappa = 0.41, \quad \gamma_1 = \beta_1/\beta^* - \sigma_{\omega 1} \kappa^2/\sqrt{\beta^*}
\]

(A19)

and the eddy viscosity is defined as:

\[
\nu_t = \frac{a_1 k}{\max(a_1 \omega; \Omega F_2)}
\]

(A20)

where \( \Omega \) is the absolute value of the vorticity. \( F_2 \) is given by:

\[
F_2 = \tanh (\text{arg}_2^2)
\]  

(A21)

\[
\text{arg}_2 = \max\left(2 \frac{\sqrt{k}}{0.09 \omega y}, \frac{500\nu}{y^2 \omega}\right)
\]  

(A22)

Menter (1994) stresses that in applying this model, it is important to note of the following ambiguity in the formulation of the production term of \( \omega \) for the SST model. The definition of the production term of \( \omega \) is sometimes written as:
APPENDICES

\[ P_\omega = \gamma \frac{\omega}{k} \tau_{ij} \frac{\partial u_i}{\partial x_j} \]  
(A23)

which introduces the nondimensional group \( \nu_t (\omega/k) \) in front of the strain rate tensor. In the original and in the BSL model, this group is equal to one and the two formulations for \( P_\omega \) are therefore identical. This is not the case for the SST model because of Equation (A20). The SST model has been calibrated with respect to Equation (A10) and Equation (A23) should therefore not be used.
APPENDIX 3: TYPICAL K-VALUE FOR VARIOUS INHIBITORS (BAI AND BAI (2005))

<table>
<thead>
<tr>
<th>Inhibitor</th>
<th>K value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Methanol</td>
<td>2335</td>
</tr>
<tr>
<td>Ethanol</td>
<td>2335</td>
</tr>
<tr>
<td>Ethylene glycol (MEG)</td>
<td>2700</td>
</tr>
<tr>
<td>Diethylene glycol (DEG)</td>
<td>4000</td>
</tr>
<tr>
<td>Triethylene glycol (TEG)</td>
<td>5400</td>
</tr>
</tbody>
</table>
APPENDICES

APPENDIX 4: THE EFFECT OF THERMODYNAMIC INHIBITORS ON HYDRATES FORMATION (BAI AND BAI, (2005))

![Graph showing the effect of thermodynamic inhibitors on hydrates formation. The graph plots pressure (psi) against temperature (°F) for various inhibitors including No inhibitors, TEG, MEG, MeOH, and NaCl. The graph illustrates the transition temperature and pressure for hydrate formation.](image-url)
APPENDIX 5: EXAMPLE OF PROJECTS USING ELECTRICAL HEATING AS MEANS OF MANAGING SOLIDS FORMATION AND DEPOSITION (COCHRAN (2003))

<table>
<thead>
<tr>
<th>Operator &amp; Project</th>
<th>Line Diameter &amp; Length</th>
<th>Water Depth (m)</th>
<th>Year of Installation</th>
<th>Electrical Heating Method</th>
<th>Mode of Operation</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shell Nakika</td>
<td>10-in. x 16-in. PIP</td>
<td>1900</td>
<td>2003</td>
<td>PIP Direct</td>
<td>Remediation</td>
<td>Engineering &amp; qualification testing</td>
</tr>
<tr>
<td>Shell Serrano / Oregano</td>
<td>6-in. x 10-in. PIP two lines 10 and 12 km</td>
<td>1000</td>
<td>2001</td>
<td>PIP Direct</td>
<td>Temperature maintenance for shutdown</td>
<td>In operation</td>
</tr>
<tr>
<td>StatOil Huldra</td>
<td>8-in. single pipe 15 km</td>
<td>300 to 400</td>
<td>2001</td>
<td>Earthed Direct</td>
<td>Temperature maintenance of 25°C during shutdown</td>
<td>In operation</td>
</tr>
<tr>
<td>StatOil Asgard</td>
<td>8-in. single pipe 6 lines 43 km total</td>
<td>300 to 400</td>
<td>2000</td>
<td>Earthed Direct</td>
<td>Temperature maintenance of 27°C during shutdown</td>
<td>In operation</td>
</tr>
<tr>
<td>StatOil Steppner</td>
<td>20-in. single pipe 12.6 km</td>
<td>-</td>
<td>1996</td>
<td>Induction</td>
<td>Temperature maintenance</td>
<td>In operation</td>
</tr>
</tbody>
</table>
APPENDIX 6: THERMAL CONDUCTIVITIES OF TYPICAL SOIL SURROUNDING PIPELINE FOR BURIAL METHOD (BAI AND BAI (2005))

<table>
<thead>
<tr>
<th>Material</th>
<th>Thermal conductivity, $k_{eff}$</th>
<th>BTU/(ft⋅hr⋅°F)</th>
<th>W/(m⋅K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Peat (dry)</td>
<td></td>
<td>0.10</td>
<td>0.17</td>
</tr>
<tr>
<td>Peat (wet)</td>
<td></td>
<td>0.31</td>
<td>0.54</td>
</tr>
<tr>
<td>Peat (icy)</td>
<td></td>
<td>1.09</td>
<td>1.89</td>
</tr>
<tr>
<td>Sand soil (dry)</td>
<td></td>
<td>0.25 – 0.40</td>
<td>0.43 – 0.69</td>
</tr>
<tr>
<td>Sandy soil (moist)</td>
<td></td>
<td>0.50 – 0.60</td>
<td>0.87 – 1.04</td>
</tr>
<tr>
<td>Sandy soil (soaked)</td>
<td></td>
<td>1.10 – 1.40</td>
<td>1.90 – 2.42</td>
</tr>
<tr>
<td>Clay soil (dry)</td>
<td></td>
<td>0.20 – 0.30</td>
<td>0.35 – 0.52</td>
</tr>
<tr>
<td>Clay soil (moist)</td>
<td></td>
<td>0.40 – 0.50</td>
<td>0.69 – 0.87</td>
</tr>
<tr>
<td>Clay soil (wet)</td>
<td></td>
<td>0.60 – 0.90</td>
<td>1.04 – 1.56</td>
</tr>
<tr>
<td>Clay soil (frozen)</td>
<td></td>
<td>1.45</td>
<td>2.51</td>
</tr>
<tr>
<td>Gravel</td>
<td></td>
<td>0.55 – 0.72</td>
<td>0.9 – 1.25</td>
</tr>
<tr>
<td>Gravel (sandy)</td>
<td></td>
<td>1.45</td>
<td>2.51</td>
</tr>
<tr>
<td>Limestone</td>
<td></td>
<td>0.75</td>
<td>1.30</td>
</tr>
<tr>
<td>Sandstone</td>
<td></td>
<td>0.94 – 1.20</td>
<td>1.63 – 2.08</td>
</tr>
</tbody>
</table>
### APPENDIX 7: PROPERTIES OF TYPICAL EXTERNAL INSULATION MATERIALS (BAI AND BAI (2005))

<table>
<thead>
<tr>
<th>Name of Coating System</th>
<th>Abbreviation</th>
<th>Releable (yes/no)</th>
<th>Max. Temp. [°C]</th>
<th>Water Absorption</th>
<th>Conductivity increase at end of life [%]</th>
<th>Compression at end of life [%]</th>
<th>Max. Thickness if &lt;100mm</th>
<th>Density [kg/m³]</th>
<th>Max. Depth [m]</th>
<th>Conductivity [W/m²K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polypropylene – Solid</td>
<td>PP</td>
<td>Y</td>
<td>145</td>
<td>&lt;0.5%</td>
<td>&lt;1%</td>
<td>&lt;2%</td>
<td>60</td>
<td>900</td>
<td>3000</td>
<td>0.22</td>
</tr>
<tr>
<td>Polypropylene – Syntactic PPF (e.g. Carcice)</td>
<td>SPP</td>
<td>Y</td>
<td>115</td>
<td>&lt;5.0%</td>
<td>&lt;1%</td>
<td>&lt;5%</td>
<td>-</td>
<td>600-800</td>
<td>-</td>
<td>0.13 – 0.22</td>
</tr>
<tr>
<td>Polypropylene – Reinforced Form Combination</td>
<td>RPPF</td>
<td>Y</td>
<td>115-140</td>
<td>&lt;0.5%</td>
<td>&lt;1%</td>
<td>&lt;0.5%</td>
<td>multi-layer</td>
<td>600-800</td>
<td>600–3000</td>
<td>0.16 – 0.18</td>
</tr>
<tr>
<td>Polyurethane – Solid</td>
<td>PU</td>
<td>Y</td>
<td>115</td>
<td>&lt;5.0%</td>
<td>&lt;10%</td>
<td>&lt;2%</td>
<td>1150</td>
<td></td>
<td></td>
<td>0.19–0.20</td>
</tr>
<tr>
<td>Polyurethane – Syntactic (plastic beads)</td>
<td>SPU</td>
<td>Y</td>
<td>70-115</td>
<td>&lt;5.0%</td>
<td>&lt;10%</td>
<td>&lt;2%</td>
<td>multi-layer</td>
<td>750–780</td>
<td>100@115C 300@90C</td>
<td>0.12 – 0.15</td>
</tr>
<tr>
<td>Polyurethane – Glass Syntactic</td>
<td>GSPU</td>
<td>Y</td>
<td>55-90</td>
<td>&lt;2.2%</td>
<td>Variable on thickness</td>
<td>&lt;5%</td>
<td>multi-layer</td>
<td>610-830</td>
<td>2000–3000</td>
<td>0.12 – 0.17</td>
</tr>
<tr>
<td>Polyurethane – Reinforced Form Combination</td>
<td>RPUF</td>
<td>N.A</td>
<td>n.a.</td>
<td>n.a.</td>
<td>n.a.</td>
<td>n.a.</td>
<td>multi-layer</td>
<td>448</td>
<td>-</td>
<td>0.080</td>
</tr>
<tr>
<td>Phenolic Syntactic</td>
<td>PhS</td>
<td>Y</td>
<td>290</td>
<td>&lt;5%</td>
<td>&lt;1%</td>
<td></td>
<td></td>
<td>500</td>
<td>-</td>
<td>0.080</td>
</tr>
<tr>
<td>Epoxy Syntactic</td>
<td>SEP</td>
<td>N</td>
<td>75-100</td>
<td>&lt;5.0%</td>
<td>&lt;10%</td>
<td>&lt;2%</td>
<td>75</td>
<td>590–720</td>
<td>2000–3000</td>
<td>0.10–0.135</td>
</tr>
<tr>
<td>Epoxy Syntactic with Mini-Spheres</td>
<td>MSEP</td>
<td>N</td>
<td>75</td>
<td>&lt;5.0%</td>
<td>&lt;10%</td>
<td>&lt;4%</td>
<td>540</td>
<td></td>
<td></td>
<td>0.12</td>
</tr>
</tbody>
</table>

**Note 1:** For the multi-layer coatings the properties quoted are for the insulation layer.

**Note 2:** Value measured at mean product temperature of 50 °C;

**Note 3:** Values are approximate only. All values used in design must be provided by the manufacturer.
## APPENDIX 8: PROPERTIES OF TYPICAL INSULATION MATERIALS FOR PIP SYSTEM (BAI AND BAI (2005))

<table>
<thead>
<tr>
<th>Insulation material</th>
<th>Density [kg/m³]</th>
<th>Conductivity [W/mK]</th>
<th>Thickness [mm]</th>
<th>Annulus Gap (if any)</th>
<th>Max. temp [°C]</th>
<th>U-value [W/m²K] 16&quot; inner pipe</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mineral wool</td>
<td>140</td>
<td>0.037</td>
<td>100</td>
<td>clearance</td>
<td>700</td>
<td>1.6 (40 mm)</td>
<td>Rockwool or Glava, usually in combination with Mylar reflective film.</td>
</tr>
<tr>
<td>Alumina silicate microspheres</td>
<td>390-420</td>
<td>0.1</td>
<td>no limit</td>
<td>none</td>
<td>1000+</td>
<td>3.9 (100 mm)</td>
<td>Commonly referred to as Fly-ash, injected to fill the annulus.</td>
</tr>
<tr>
<td>Thermal cement</td>
<td>900-1200</td>
<td>0.26</td>
<td>100</td>
<td>none</td>
<td>200</td>
<td>-</td>
<td>Currently being investigated under a JIP to provide collapse resistance with reduced carrier pipe wall thickness.</td>
</tr>
<tr>
<td>LD PU foam</td>
<td>60</td>
<td>0.027</td>
<td>125</td>
<td>none</td>
<td>147</td>
<td>0.76 (100 mm)</td>
<td>Pre-assembled as single or double jointed system, used on the Erskine Replacement.</td>
</tr>
<tr>
<td>HD PU foam</td>
<td>150</td>
<td>0.035</td>
<td>125</td>
<td>none</td>
<td>147</td>
<td>1.2 (100 mm)</td>
<td></td>
</tr>
<tr>
<td>Microporous silica blanket</td>
<td>200-400</td>
<td>0.022</td>
<td>24</td>
<td>clearance</td>
<td>900</td>
<td>0.4 (100 mm)</td>
<td>Cotton blanket, calcium based powder, glass and titanium fibers.</td>
</tr>
<tr>
<td>Vacuum insulation panels</td>
<td>60-145</td>
<td>0.006-0.008</td>
<td>10</td>
<td>clearance</td>
<td>160</td>
<td>0.26 (100 mm)</td>
<td>Foam shells formed under vacuum with aluminium foil and uses gas absorbing 'getter' pills to absorb any free gas thereafter.</td>
</tr>
</tbody>
</table>
## APPENDIX 9: PHYSICAL PROPERTIES OF PIPES AND BUNDLE CONFIGURATION

<table>
<thead>
<tr>
<th>No.</th>
<th>Pipe</th>
<th>Area (m²)</th>
<th>Diameter (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Product</td>
<td>0.0011169</td>
<td>0.35501</td>
</tr>
<tr>
<td>2</td>
<td>Test</td>
<td>0.0006382</td>
<td>0.20294</td>
</tr>
<tr>
<td>3</td>
<td>HeatFlow</td>
<td>0.0003190</td>
<td>0.10145</td>
</tr>
<tr>
<td>4</td>
<td>HeatReturn</td>
<td>0.0003190</td>
<td>0.10145</td>
</tr>
<tr>
<td>5</td>
<td>Sleeve</td>
<td>0.0023936</td>
<td>0.76183</td>
</tr>
</tbody>
</table>
# APPENDIX 10: THERMAL PROPERTIES OF GAS (NITROGEN)

Source: [http://www.engineeringtoolbox.com/air-properties-d_156.html](http://www.engineeringtoolbox.com/air-properties-d_156.html)

<table>
<thead>
<tr>
<th>Temp (°C)</th>
<th>Pr</th>
<th>Thermal Conductivity (W/m K)</th>
<th>Kinematic Viscosity (m²/s) x10⁶</th>
<th>Thermal Expansion (K⁻¹) x10⁻³</th>
</tr>
</thead>
<tbody>
<tr>
<td>-150</td>
<td>0.76</td>
<td>0.0116</td>
<td>3.08</td>
<td>8.21</td>
</tr>
<tr>
<td>-100</td>
<td>0.74</td>
<td>0.016</td>
<td>5.95</td>
<td>5.82</td>
</tr>
<tr>
<td>-50</td>
<td>0.725</td>
<td>0.0204</td>
<td>9.55</td>
<td>4.51</td>
</tr>
<tr>
<td>0</td>
<td>0.715</td>
<td>0.0243</td>
<td>13.3</td>
<td>3.67</td>
</tr>
<tr>
<td>20</td>
<td>0.713</td>
<td>0.0257</td>
<td>15.11</td>
<td>3.43</td>
</tr>
<tr>
<td>40</td>
<td>0.711</td>
<td>0.0271</td>
<td>16.97</td>
<td>3.2</td>
</tr>
<tr>
<td>60</td>
<td>0.709</td>
<td>0.0285</td>
<td>18.9</td>
<td>3</td>
</tr>
<tr>
<td>80</td>
<td>0.708</td>
<td>0.0299</td>
<td>20.94</td>
<td>2.83</td>
</tr>
<tr>
<td>100</td>
<td>0.703</td>
<td>0.0314</td>
<td>23.06</td>
<td>2.68</td>
</tr>
<tr>
<td>120</td>
<td>0.7</td>
<td>0.0328</td>
<td>25.23</td>
<td>2.55</td>
</tr>
<tr>
<td>140</td>
<td>0.695</td>
<td>0.0343</td>
<td>27.55</td>
<td>2.43</td>
</tr>
<tr>
<td>160</td>
<td>0.69</td>
<td>0.0358</td>
<td>29.85</td>
<td>2.32</td>
</tr>
<tr>
<td>180</td>
<td>0.69</td>
<td>0.0372</td>
<td>32.29</td>
<td>2.21</td>
</tr>
<tr>
<td>200</td>
<td>0.685</td>
<td>0.0386</td>
<td>34.63</td>
<td>2.11</td>
</tr>
<tr>
<td>250</td>
<td>0.68</td>
<td>0.0421</td>
<td>41.17</td>
<td>1.91</td>
</tr>
<tr>
<td>300</td>
<td>0.68</td>
<td>0.0454</td>
<td>47.85</td>
<td>1.75</td>
</tr>
<tr>
<td>350</td>
<td>0.68</td>
<td>0.0485</td>
<td>55.05</td>
<td>1.61</td>
</tr>
<tr>
<td>400</td>
<td>0.68</td>
<td>0.0515</td>
<td>62.53</td>
<td>1.49</td>
</tr>
</tbody>
</table>


<table>
<thead>
<tr>
<th>Temp (K)</th>
<th>Temp (°C)</th>
<th>Thermal Diffusivity (m²/s) x10⁶</th>
</tr>
</thead>
<tbody>
<tr>
<td>250</td>
<td>-23.15</td>
<td>15.9</td>
</tr>
<tr>
<td>300</td>
<td>26.85</td>
<td>22.5</td>
</tr>
<tr>
<td>350</td>
<td>76.85</td>
<td>29.9</td>
</tr>
<tr>
<td>400</td>
<td>126.85</td>
<td>38.3</td>
</tr>
<tr>
<td>450</td>
<td>176.85</td>
<td>47.2</td>
</tr>
</tbody>
</table>
Prandtl

\[ y = -0.000x + 0.715 \]
\[ R^2 = 0.961 \]

Kinematic Viscosity, \( \nu \) (m\(^2\)/s) \( \times 10^{-6} \)

\[ y = 0.099x + 13.10 \]
\[ R^2 = 0.998 \]

Thermal Conductivity, \( \lambda \) (W/m K)

\[ y = 7E-05x + 0.024 \]
\[ R^2 = 0.999 \]

Thermal Expansion, \( \beta \) (K\(^{-1}\)) \( \times 10^{-3} \)

\[ y = -0.009x + 3.611 \]
\[ R^2 = 0.987 \]

Thermal Diffusivity, \( \kappa \) (m\(^2\)/s) \( \times 10^{-6} \)

\[ y = 0.156x + 18.71 \]
\[ R^2 = 0.996 \]
## APPENDIX 11: SELECTED PIPE TEMPERATURES (OC) AND CFD CALCULATED HEAT FLOWS (W) OF HORIZONTAL BUNDLE

<table>
<thead>
<tr>
<th>Run</th>
<th>Temperatures (°C)</th>
<th>Heat Flows (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>33.56 44.99 78.32 32.22 10.00</td>
<td>0.1062 -0.0232 0.2270 0.0422 -0.3504</td>
</tr>
<tr>
<td>2</td>
<td>43.70 26.28 60.76 37.51 10.00</td>
<td>0.2355 0.1699 0.1061 0.0307 -0.5358</td>
</tr>
<tr>
<td>3</td>
<td>44.92 18.41 58.70 38.51 10.00</td>
<td>0.1514 -0.0294 0.1768 0.0545 -0.3524</td>
</tr>
<tr>
<td>4</td>
<td>45.41 16.68 57.50 37.58 10.00</td>
<td>0.3401 -0.0187 0.1179 0.0791 -0.5117</td>
</tr>
<tr>
<td>5</td>
<td>48.72 17.95 56.54 39.09 10.00</td>
<td>0.3519 0.0095 0.1070 0.0628 -0.5310</td>
</tr>
<tr>
<td>6</td>
<td>51.84 19.52 55.60 40.82 10.00</td>
<td>0.3404 0.0901 0.0925 0.0498 -0.5731</td>
</tr>
<tr>
<td>7</td>
<td>42.70 16.01 65.25 37.72 10.00</td>
<td>0.1863 -0.0469 0.2311 0.0453 -0.4155</td>
</tr>
<tr>
<td>8</td>
<td>47.46 15.81 69.13 39.41 10.00</td>
<td>0.3568 -0.0655 0.1361 0.0344 -0.4661</td>
</tr>
<tr>
<td>9</td>
<td>49.79 14.85 65.93 33.56 10.00</td>
<td>0.3794 -0.0626 0.1199 0.0477 -0.4850</td>
</tr>
<tr>
<td>10</td>
<td>51.84 15.92 69.44 36.97 10.00</td>
<td>0.2222 -0.0503 0.2404 0.0789 -0.4921</td>
</tr>
<tr>
<td>11</td>
<td>51.80 16.86 72.03 42.71 10.00</td>
<td>0.2125 -0.0375 0.2563 0.0660 -0.4988</td>
</tr>
<tr>
<td>12</td>
<td>45.22 18.59 81.67 32.58 10.00</td>
<td>0.2146 -0.0517 0.3216 -0.0090 -0.4739</td>
</tr>
<tr>
<td>13</td>
<td>44.92 14.26 93.59 20.00 10.00</td>
<td>0.0446 -0.0619 0.4101 0.0261 -0.4103</td>
</tr>
<tr>
<td>14</td>
<td>42.15 13.67 97.22 19.74 10.00</td>
<td>-0.0511 -0.0532 0.4340 0.0402 -0.3517</td>
</tr>
<tr>
<td>15</td>
<td>52.36 19.20 77.72 47.61 10.00</td>
<td>0.4061 -0.0673 0.1812 0.0455 -0.5609</td>
</tr>
<tr>
<td>16</td>
<td>55.43 18.78 74.15 45.94 10.00</td>
<td>0.4552 -0.0767 0.1582 0.0350 -0.5704</td>
</tr>
<tr>
<td>17</td>
<td>60.68 16.72 68.81 37.10 10.00</td>
<td>0.5148 -0.0798 0.1151 0.0342 -0.5847</td>
</tr>
<tr>
<td>18</td>
<td>60.48 19.15 73.31 43.97 10.00</td>
<td>0.3035 -0.0429 0.2421 0.0816 -0.5866</td>
</tr>
<tr>
<td>19</td>
<td>55.71 20.36 78.88 47.26 10.00</td>
<td>0.2735 -0.0311 0.2673 0.0398 -0.5526</td>
</tr>
<tr>
<td>20</td>
<td>52.31 19.89 86.64 33.69 10.00</td>
<td>0.3444 -0.0443 0.2446 0.0551 -0.5968</td>
</tr>
<tr>
<td>21</td>
<td>60.49 20.94 78.45 44.71 10.00</td>
<td>0.4759 -0.0383 0.1675 0.0700 -0.6700</td>
</tr>
<tr>
<td>22</td>
<td>62.34 20.75 75.20 43.44 10.00</td>
<td>0.5005 -0.0626 0.1574 0.0621 -0.6509</td>
</tr>
<tr>
<td>23</td>
<td>64.44 20.49 71.62 42.44 10.00</td>
<td>0.5438 -0.0512 0.1388 0.0676 -0.6909</td>
</tr>
<tr>
<td>24</td>
<td>66.38 20.34 68.21 42.70 10.00</td>
<td>0.5838 -0.0672 0.1209 0.0606 -0.6950</td>
</tr>
<tr>
<td>25</td>
<td>60.69 22.69 83.98 40.59 10.00</td>
<td>0.3215 -0.0330 0.2857 0.0317 -0.6110</td>
</tr>
<tr>
<td>26</td>
<td>54.97 18.11 89.14 34.98 10.00</td>
<td>0.1873 -0.0477 0.3084 0.0591 -0.5061</td>
</tr>
<tr>
<td>27</td>
<td>53.94 15.58 90.64 38.19 10.00</td>
<td>0.1315 -0.0335 0.3310 0.0623 -0.4920</td>
</tr>
<tr>
<td>28</td>
<td>56.49 13.96 93.52 40.15 10.00</td>
<td>0.3481 -0.0310 0.2794 0.0244 -0.6189</td>
</tr>
<tr>
<td>29</td>
<td>63.94 25.29 77.76 42.28 10.00</td>
<td>0.5031 -0.0344 0.1624 0.0654 -0.6879</td>
</tr>
<tr>
<td>30</td>
<td>58.51 16.18 99.55 39.43 10.00</td>
<td>0.1987 -0.0577 0.3653 0.0729 -0.5770</td>
</tr>
<tr>
<td>31</td>
<td>57.60 13.88 102.22 42.72 10.00</td>
<td>0.1050 -0.0435 0.3871 0.1128 -0.5619</td>
</tr>
<tr>
<td>32</td>
<td>53.94 15.58 102.84 44.41 10.00</td>
<td>0.0608 -0.0466 0.4017 0.1236 -0.5414</td>
</tr>
<tr>
<td>33</td>
<td>69.89 24.99 71.84 47.55 10.00</td>
<td>0.5917 -0.0391 0.1401 0.0771 -0.7681</td>
</tr>
<tr>
<td>34</td>
<td>63.22 22.93 102.91 39.79 10.00</td>
<td>0.2661 -0.0680 0.3600 0.0987 -0.6542</td>
</tr>
<tr>
<td>35</td>
<td>58.76 20.43 109.03 48.49 10.00</td>
<td>0.1335 -0.0419 0.3498 0.1371 -0.5791</td>
</tr>
<tr>
<td>36</td>
<td>83.06 27.17 82.92 51.21 10.00</td>
<td>0.7459 -0.0432 0.1707 0.0968 -0.9694</td>
</tr>
<tr>
<td>Run</td>
<td>Heat Transfer Coefficient (W/m².K)</td>
<td>Prandtl Number</td>
</tr>
<tr>
<td>-----</td>
<td>-----------------------------------</td>
<td>----------------</td>
</tr>
<tr>
<td>1</td>
<td>11.88</td>
<td>13.17</td>
</tr>
<tr>
<td>2</td>
<td>10.53</td>
<td>9.97</td>
</tr>
<tr>
<td>3</td>
<td>9.97</td>
<td>8.37</td>
</tr>
<tr>
<td>4</td>
<td>10.59</td>
<td>10.08</td>
</tr>
<tr>
<td>5</td>
<td>10.85</td>
<td>4.18</td>
</tr>
<tr>
<td>6</td>
<td>11.07</td>
<td>9.28</td>
</tr>
<tr>
<td>7</td>
<td>11.60</td>
<td>12.02</td>
</tr>
<tr>
<td>8</td>
<td>10.87</td>
<td>9.93</td>
</tr>
<tr>
<td>9</td>
<td>11.09</td>
<td>9.99</td>
</tr>
<tr>
<td>10</td>
<td>10.12</td>
<td>9.86</td>
</tr>
<tr>
<td>12</td>
<td>9.18</td>
<td>8.31</td>
</tr>
<tr>
<td>13</td>
<td>7.88</td>
<td>8.84</td>
</tr>
<tr>
<td>14</td>
<td>187.68</td>
<td>8.62</td>
</tr>
<tr>
<td>15</td>
<td>10.88</td>
<td>10.01</td>
</tr>
<tr>
<td>16</td>
<td>11.48</td>
<td>10.30</td>
</tr>
<tr>
<td>17</td>
<td>11.84</td>
<td>10.27</td>
</tr>
<tr>
<td>20</td>
<td>10.65</td>
<td>10.67</td>
</tr>
<tr>
<td>21</td>
<td>11.31</td>
<td>7.18</td>
</tr>
<tr>
<td>22</td>
<td>11.42</td>
<td>10.70</td>
</tr>
<tr>
<td>23</td>
<td>11.73</td>
<td>8.14</td>
</tr>
<tr>
<td>24</td>
<td>11.93</td>
<td>9.62</td>
</tr>
<tr>
<td>25</td>
<td>10.26</td>
<td>8.31</td>
</tr>
<tr>
<td>26</td>
<td>10.70</td>
<td>10.23</td>
</tr>
<tr>
<td>28</td>
<td>10.15</td>
<td>9.23</td>
</tr>
<tr>
<td>29</td>
<td>11.54</td>
<td>10.56</td>
</tr>
<tr>
<td>30</td>
<td>10.80</td>
<td>10.53</td>
</tr>
<tr>
<td>31</td>
<td>9.67</td>
<td>10.74</td>
</tr>
<tr>
<td>32</td>
<td>7.99</td>
<td>10.75</td>
</tr>
<tr>
<td>33</td>
<td>12.29</td>
<td>12.05</td>
</tr>
<tr>
<td>34</td>
<td>11.27</td>
<td>11.52</td>
</tr>
<tr>
<td>35</td>
<td>10.36</td>
<td>11.15</td>
</tr>
<tr>
<td>36</td>
<td>12.61</td>
<td>14.86</td>
</tr>
</tbody>
</table>
APPENDIX 13: VALUES OF CONSTANT ‘A’ (EQUATION 6.3) FOR ALL PIPES

\[ Nu = a Ra^{1/3} \left[ f(Pr) \right]^{1/3} \]

<table>
<thead>
<tr>
<th>Run</th>
<th>a</th>
<th>Product</th>
<th>Test</th>
<th>HeatFlow</th>
<th>HeatReturn</th>
<th>Sleeve</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.505</td>
<td>2.723</td>
<td>2.808</td>
<td>4.293</td>
<td>3.614</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2.655</td>
<td>3.354</td>
<td>2.888</td>
<td>3.841</td>
<td>3.803</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2.616</td>
<td>3.126</td>
<td>2.723</td>
<td>3.908</td>
<td>3.826</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>2.624</td>
<td>3.105</td>
<td>2.808</td>
<td>4.106</td>
<td>4.091</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>2.579</td>
<td>3.207</td>
<td>2.854</td>
<td>4.034</td>
<td>4.125</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2.554</td>
<td>3.402</td>
<td>2.914</td>
<td>3.986</td>
<td>4.134</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>2.653</td>
<td>2.985</td>
<td>2.634</td>
<td>3.870</td>
<td>3.850</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>2.674</td>
<td>3.249</td>
<td>2.736</td>
<td>4.314</td>
<td>4.029</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>2.499</td>
<td>3.114</td>
<td>2.691</td>
<td>4.880</td>
<td>4.037</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>2.591</td>
<td>3.364</td>
<td>2.890</td>
<td>4.705</td>
<td>3.649</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>2.578</td>
<td>3.048</td>
<td>2.676</td>
<td>3.832</td>
<td>3.890</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>2.646</td>
<td>3.262</td>
<td>2.665</td>
<td>4.734</td>
<td>3.896</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>2.442</td>
<td>2.847</td>
<td>2.589</td>
<td>3.249</td>
<td>4.292</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>2.577</td>
<td>2.833</td>
<td>2.617</td>
<td>3.480</td>
<td>4.116</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>2.675</td>
<td>3.474</td>
<td>2.762</td>
<td>3.848</td>
<td>3.970</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>2.585</td>
<td>3.155</td>
<td>2.763</td>
<td>3.881</td>
<td>4.032</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>2.499</td>
<td>3.369</td>
<td>2.921</td>
<td>4.989</td>
<td>3.876</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>2.614</td>
<td>2.619</td>
<td>2.830</td>
<td>3.999</td>
<td>3.666</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>2.609</td>
<td>3.271</td>
<td>2.738</td>
<td>3.831</td>
<td>3.984</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>2.556</td>
<td>3.295</td>
<td>2.624</td>
<td>4.918</td>
<td>4.008</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>2.534</td>
<td>2.875</td>
<td>2.765</td>
<td>4.344</td>
<td>4.144</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>2.512</td>
<td>3.212</td>
<td>2.801</td>
<td>4.402</td>
<td>4.096</td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>2.510</td>
<td>2.720</td>
<td>2.842</td>
<td>4.756</td>
<td>4.148</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>2.506</td>
<td>2.620</td>
<td>2.899</td>
<td>4.821</td>
<td>4.175</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>2.575</td>
<td>2.928</td>
<td>2.803</td>
<td>3.908</td>
<td>4.053</td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>2.492</td>
<td>3.323</td>
<td>2.685</td>
<td>4.744</td>
<td>4.078</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>2.514</td>
<td>3.027</td>
<td>2.701</td>
<td>4.206</td>
<td>4.033</td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>2.591</td>
<td>3.075</td>
<td>2.609</td>
<td>4.056</td>
<td>3.891</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>2.473</td>
<td>3.387</td>
<td>2.829</td>
<td>4.804</td>
<td>3.958</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>2.515</td>
<td>3.365</td>
<td>2.684</td>
<td>3.861</td>
<td>4.029</td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>2.563</td>
<td>2.954</td>
<td>2.675</td>
<td>4.066</td>
<td>4.043</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>2.632</td>
<td>2.836</td>
<td>2.637</td>
<td>4.091</td>
<td>3.947</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>2.489</td>
<td>3.492</td>
<td>2.951</td>
<td>4.325</td>
<td>3.963</td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>2.449</td>
<td>3.461</td>
<td>2.668</td>
<td>4.692</td>
<td>4.057</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>2.628</td>
<td>3.114</td>
<td>2.681</td>
<td>3.996</td>
<td>3.917</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>2.555</td>
<td>3.338</td>
<td>2.995</td>
<td>4.780</td>
<td>3.813</td>
<td></td>
</tr>
</tbody>
</table>
\[ \text{y} = -0.002x + 2.603 \quad R^2 = 0.127 \]

\[ \text{y} = 0.001x + 3.095 \quad R^2 = 0.004 \]

\[ \text{y} = 0.001x + 2.771 \quad R^2 = 0.003 \]

\[ \text{y} = 0.010x + 4.047 \quad R^2 = 0.060 \]

\[ \text{y} = 0.002x + 3.932 \quad R^2 = 0.030 \]
APPENDICES

APPENDIX 14: VALUES FOR CONSTANT ‘C’ AND ‘D’ (EQUATION 6.7) FOR ALL THE PIPES

\[ Nu = cRa_1^{1/3} \left[ f(Pr) \right]^{1/3} \left( Ra_2 Ra_3 Ra_4 Ra_5 \right)^d \]

<table>
<thead>
<tr>
<th>Run</th>
<th>Product</th>
<th>Test</th>
<th>HeatFlow</th>
<th>HeatReturn</th>
<th>Sleeve</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.864</td>
<td>0.462</td>
<td>1.274</td>
<td>1.459</td>
<td>2.641</td>
</tr>
<tr>
<td>2</td>
<td>1.993</td>
<td>0.551</td>
<td>1.320</td>
<td>1.341</td>
<td>2.792</td>
</tr>
<tr>
<td>3</td>
<td>1.955</td>
<td>0.512</td>
<td>1.229</td>
<td>1.344</td>
<td>2.794</td>
</tr>
<tr>
<td>4</td>
<td>1.959</td>
<td>0.508</td>
<td>1.265</td>
<td>1.408</td>
<td>2.984</td>
</tr>
<tr>
<td>5</td>
<td>1.927</td>
<td>0.523</td>
<td>1.284</td>
<td>1.384</td>
<td>3.009</td>
</tr>
<tr>
<td>6</td>
<td>1.909</td>
<td>0.553</td>
<td>1.311</td>
<td>1.370</td>
<td>3.017</td>
</tr>
<tr>
<td>7</td>
<td>1.979</td>
<td>0.488</td>
<td>1.186</td>
<td>1.323</td>
<td>2.806</td>
</tr>
<tr>
<td>8</td>
<td>1.992</td>
<td>0.527</td>
<td>1.227</td>
<td>1.465</td>
<td>2.931</td>
</tr>
<tr>
<td>9</td>
<td>1.865</td>
<td>0.511</td>
<td>1.211</td>
<td>1.655</td>
<td>2.941</td>
</tr>
<tr>
<td>10</td>
<td>1.932</td>
<td>0.549</td>
<td>1.295</td>
<td>1.590</td>
<td>2.656</td>
</tr>
<tr>
<td>11</td>
<td>1.918</td>
<td>0.491</td>
<td>1.196</td>
<td>1.297</td>
<td>2.826</td>
</tr>
<tr>
<td>12</td>
<td>1.977</td>
<td>0.537</td>
<td>1.210</td>
<td>1.613</td>
<td>2.847</td>
</tr>
<tr>
<td>13</td>
<td>1.825</td>
<td>0.474</td>
<td>1.178</td>
<td>1.096</td>
<td>3.135</td>
</tr>
<tr>
<td>14</td>
<td>1.924</td>
<td>0.472</td>
<td>1.191</td>
<td>1.174</td>
<td>3.006</td>
</tr>
<tr>
<td>15</td>
<td>1.990</td>
<td>0.556</td>
<td>1.234</td>
<td>1.305</td>
<td>2.884</td>
</tr>
<tr>
<td>16</td>
<td>1.924</td>
<td>0.505</td>
<td>1.233</td>
<td>1.314</td>
<td>2.929</td>
</tr>
<tr>
<td>17</td>
<td>1.864</td>
<td>0.548</td>
<td>1.307</td>
<td>1.682</td>
<td>2.821</td>
</tr>
<tr>
<td>18</td>
<td>1.946</td>
<td>0.421</td>
<td>1.262</td>
<td>1.350</td>
<td>2.665</td>
</tr>
<tr>
<td>19</td>
<td>1.941</td>
<td>0.522</td>
<td>1.223</td>
<td>1.298</td>
<td>2.895</td>
</tr>
<tr>
<td>20</td>
<td>1.909</td>
<td>0.539</td>
<td>1.189</td>
<td>1.667</td>
<td>2.926</td>
</tr>
<tr>
<td>21</td>
<td>1.887</td>
<td>0.460</td>
<td>1.236</td>
<td>1.469</td>
<td>3.013</td>
</tr>
<tr>
<td>22</td>
<td>1.872</td>
<td>0.515</td>
<td>1.252</td>
<td>1.488</td>
<td>2.979</td>
</tr>
<tr>
<td>23</td>
<td>1.871</td>
<td>0.437</td>
<td>1.270</td>
<td>1.608</td>
<td>3.018</td>
</tr>
<tr>
<td>24</td>
<td>1.868</td>
<td>0.422</td>
<td>1.294</td>
<td>1.630</td>
<td>3.037</td>
</tr>
<tr>
<td>25</td>
<td>1.921</td>
<td>0.472</td>
<td>1.261</td>
<td>1.324</td>
<td>2.953</td>
</tr>
<tr>
<td>26</td>
<td>1.858</td>
<td>0.541</td>
<td>1.210</td>
<td>1.598</td>
<td>2.970</td>
</tr>
<tr>
<td>27</td>
<td>1.870</td>
<td>0.488</td>
<td>1.209</td>
<td>1.411</td>
<td>2.929</td>
</tr>
<tr>
<td>28</td>
<td>1.925</td>
<td>0.494</td>
<td>1.163</td>
<td>1.355</td>
<td>2.821</td>
</tr>
<tr>
<td>29</td>
<td>1.849</td>
<td>0.545</td>
<td>1.276</td>
<td>1.639</td>
<td>2.889</td>
</tr>
<tr>
<td>30</td>
<td>1.869</td>
<td>0.540</td>
<td>1.199</td>
<td>1.290</td>
<td>2.923</td>
</tr>
<tr>
<td>31</td>
<td>1.900</td>
<td>0.471</td>
<td>1.190</td>
<td>1.355</td>
<td>2.927</td>
</tr>
<tr>
<td>32</td>
<td>1.952</td>
<td>0.452</td>
<td>1.175</td>
<td>1.368</td>
<td>2.860</td>
</tr>
<tr>
<td>33</td>
<td>1.858</td>
<td>0.558</td>
<td>1.320</td>
<td>1.469</td>
<td>2.886</td>
</tr>
<tr>
<td>34</td>
<td>1.825</td>
<td>0.556</td>
<td>1.200</td>
<td>1.578</td>
<td>2.953</td>
</tr>
<tr>
<td>35</td>
<td>1.949</td>
<td>0.493</td>
<td>1.195</td>
<td>1.339</td>
<td>2.839</td>
</tr>
<tr>
<td>36</td>
<td>1.904</td>
<td>0.531</td>
<td>1.333</td>
<td>1.608</td>
<td>2.772</td>
</tr>
</tbody>
</table>
APPENDIX 15: EXAMPLE OF TYPICAL CFX CCL FILE (PHYSICS, BOUNDARY CONDITIONS & SIMULATION SET-UP)

+--------------------------------------------------------------------+  |  |
| | CFX Command Language for Run |
| |  |
| +--------------------------------------------------------------------+  |

LIBRARY:

CEL:

EXPRESSIONS:

\[
\begin{align*}
\text{AveRadFlux HeatFlow} & = \text{areaInt}(\text{Wall Radiative Heat Flux})@\text{HeatF} \\
\text{AveRadFlux HeatReturn} & = \text{areaInt}(\text{Wall Radiative Heat Flux})@\text{HeatR} \\
\text{AveRadFlux Product} & = \text{areaInt}(\text{Wall Radiative Heat Flux})@\text{Product} \\
\text{AveRadFlux Sleeve} & = \text{areaInt}(\text{Wall Radiative Heat Flux})@\text{Sleeve} \\
\text{AveRadFlux Test} & = \text{areaInt}(\text{Wall Radiative Heat Flux})@\text{Test} \\
\text{AveTemp Gas} & = \text{volumeAve}(\text{Temperature})@\text{Default Domain} \\
\text{Heat Flux Net} & = \text{areaInt}(\text{Wall Heat Flux})@\text{Sleeve}+\text{areaInt}(\text{Wall Heat Flux})@\text{Product}+\text{areaInt}(\text{Wall Heat Flux})@\text{HeatF}+\text{areaInt}(\text{Wall Heat Flux})@\text{HeatR}+\text{areaInt}(\text{Wall Heat Flux})@\text{Test} \\
\text{Heat Flux HeatFlow} & = \text{areaAve}(\text{Wall Heat Flux})@\text{HeatF} \\
\text{Heat Flux HeatReturn} & = \text{areaAve}(\text{Wall Heat Flux})@\text{HeatR} \\
\text{Heat Flux Product} & = \text{areaAve}(\text{Wall Heat Flux})@\text{Product} \\
\text{Heat Flux Sleeve} & = \text{areaAve}(\text{Wall Heat Flux})@\text{Sleeve} \\
\text{Heat Flux Test} & = \text{areaAve}(\text{Wall Heat Flux})@\text{Test} \\
\text{MinTemp Gas} & = \text{minVal}(\text{Temperature})@\text{Default Domain} \\
\text{Radiation HF Net} & = \text{areaInt}(\text{Wall Radiative Heat Flux})@\text{Sleeve}+\text{areaInt}(\text{Wall Radiative Heat Flux})@\text{Product}+\text{areaInt}(\text{Wall Radiative Heat Flux})@\text{HeatF}+\text{areaInt}(\text{Wall Radiative Heat Flux})@\text{HeatR}+\text{areaInt}(\text{Wall Radiative Heat Flux})@\text{Test} \\
\text{Temp HeatFlow} & = 040 \ [\text{C}] \\
\text{Temp HeatReturn} & = 035 \ [\text{C}] \\
\text{Temp Product} & = 010 \ [\text{C}] \\
\text{Temp Sleeve} & = 005 \ [\text{C}] \\
\text{Temp Test} & = 010 \ [\text{C}] \\
\text{YInclineAngle} & = -g\cdot\cos(\text{InclineAngle}) \\
2\\text{InclineAngle} & = g\cdot\sin(\text{InclineAngle}) \\
\text{InclineAngle} & = 005 \ [\text{deg}] \\
\end{align*}
\]

END

MATERIAL: N2 at STP

Material Description = Nitrogen N2 at STP (0 C and 1 atm)
Material Group = Constant Property Gases
Option = Pure Substance
Thermodynamic State = Gas

PROPERTIES:

\[
\begin{align*}
\text{Density} & = 1.250 \ [\text{kg m}^{-3}] \\
\text{Molar Mass} & = 28.01 \ [\text{kg kmol}^{-1}] \\
\text{Option} & = \text{Value} \\
\end{align*}
\]

END

REFERENCE STATE:

\[
\begin{align*}
\text{Option} & = \text{Specified Point} \\
\text{Reference Pressure} & = 1 \ [\text{atm}] \\
\text{Reference Specific Enthalpy} & = -2.5896365E+04 \ [\text{J/kg}] \\
\text{Reference Specific Entropy} & = 6.7454037E+03 \ [\text{J/kg/K}] \\
\text{Reference Temperature} & = 0 \ [\text{C}] \\
\end{align*}
\]

END

DYNAMIC VISCOSITY:

\[
\begin{align*}
\text{Dynamic Viscosity} & = 17.7E-06 \ [\text{kg m}^{-1} \text{s}^{-1}] \\
\text{Option} & = \text{Value} \\
\end{align*}
\]

END

THERMAL CONDUCTIVITY:

\[
\begin{align*}
\text{Option} & = \text{Value} \\
\text{Thermal Conductivity} & = 259E-04 \ [\text{W m}^{-1} \text{K}^{-1}] \\
\end{align*}
\]

END

ABSORPTION COEFFICIENT:

\[
\begin{align*}
\text{Absorption Coefficient} & = 1.0 \ [\text{m}^{-1}] \\
\end{align*}
\]
Option = Value
END
SCATTERING COEFFICIENT:
  Option = Value
  Scattering Coefficient = 0.0 [m^-1]
END
REFRACTIVE INDEX:
  Option = Value
  Refractive Index = 1.0 [m m^-1]
END
THERMAL EXPANSIVITY:
  Option = Value
  Thermal Expansivity = 0.00366 [K^-1]
END
END
END
END
FLOW: Flow Analysis 1
SOLUTION UNITS:
  Angle Units = [rad]
  Length Units = [m]
  Mass Units = [kg]
  Solid Angle Units = [sr]
  Temperature Units = [K]
  Time Units = [s]
END
ANALYSIS TYPE:
  Option = Steady State
EXTERNAL SOLVER COUPLING:
  Option = None
END
END
DOMAIN: Default Domain
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = GAS
BOUNDARY: Default Domain Default
  Boundary Type = WALL
  Location = Primitive 2D, Primitive 2D A
BOUNDARY CONDITIONS:
  HEAT TRANSFER:
    Option = Adiabatic
  END
  MASS AND MOMENTUM:
    Option = No Slip Wall
  END
  THERMAL RADIATION:
    Diffuse Fraction = 1.
    Emissivity = 1.
    Option = Opaque
  END
  WALL ROUGHNESS:
    Option = Smooth Wall
  END
END
BOUNDARY: HeatF
  Boundary Type = WALL
  Location = HEATFLOW
BOUNDARY CONDITIONS:
  HEAT TRANSFER:
    Fixed Temperature = Temp HeatFlow
    Option = Fixed Temperature
  END
  MASS AND MOMENTUM:
    Option = No Slip Wall
  END
  THERMAL RADIATION:
    Diffuse Fraction = 1.
    Emissivity = 0.9
    Option = Opaque
  END
  WALL ROUGHNESS:
    Option = Smooth Wall
  END
END
BOUNDARY: HeatR
    Boundary Type = WALL
    Location = HEATRETURN
    BOUNDARY CONDITIONS:
        HEAT TRANSFER:
            Fixed Temperature = Temp HeatReturn
            Option = Fixed Temperature
        END
    MASS AND MOMENTUM:
        Option = No Slip Wall
    END
    THERMAL RADIATION:
        Diffuse Fraction = 1.
        Emissivity = 0.9
        Option = Opaque
    END
    WALL ROUGHNESS:
        Option = Smooth Wall
    END
    END

BOUNDARY: Product
    Boundary Type = WALL
    Location = PRODUCT
    BOUNDARY CONDITIONS:
        HEAT TRANSFER:
            Fixed Temperature = Temp Product
            Option = Fixed Temperature
        END
    MASS AND MOMENTUM:
        Option = No Slip Wall
    END
    THERMAL RADIATION:
        Diffuse Fraction = 1.
        Emissivity = 0.9
        Option = Opaque
    END
    WALL ROUGHNESS:
        Option = Smooth Wall
    END
    END

BOUNDARY: Sleeve
    Boundary Type = WALL
    Location = SLEEVE
    BOUNDARY CONDITIONS:
        HEAT TRANSFER:
            Fixed Temperature = Temp Sleeve
            Option = Fixed Temperature
        END
    MASS AND MOMENTUM:
        Option = No Slip Wall
    END
    THERMAL RADIATION:
        Diffuse Fraction = 1.
        Emissivity = 0.9
        Option = Opaque
    END
    WALL ROUGHNESS:
        Option = Smooth Wall
    END
    END

BOUNDARY: Test
    Boundary Type = WALL
    Location = TEST
    BOUNDARY CONDITIONS:
        HEAT TRANSFER:
            Fixed Temperature = Temp Test
            Option = Fixed Temperature
        END
    MASS AND MOMENTUM:
        Option = No Slip Wall
    END
    THERMAL RADIATION:
        Diffuse Fraction = 1.
        Emissivity = 0.9
Option = Opaque
END
WALL ROUGHNESS:
  Option = Smooth Wall
END
END
END
DOMAIN MODELS:
BUOYANCY MODEL:
  Buoyancy Reference Temperature = 15 [°C]
  Gravity X Component = 0 [m s\(^{-2}\)]
  Gravity Y Component = YInclineAngle
  Gravity Z Component = ZInclineAngle
  Option = Buoyant
BUOYANCY REFERENCE LOCATION:
  Option = Automatic
END
END
DOMAIN MOTION:
  Option = Stationary
END
MESH DEFORMATION:
  Option = None
END
REFERENCE PRESSURE:
  Reference Pressure = 1 [atm]
END
END
FLUID DEFINITION: Fluid 1
Material = N\(_2\) at STP
Option = Material Library
MORPHOLOGY:
  Option = Continuous Fluid
END
END
FLUID MODELS:
COMBUSTION MODEL:
  Option = None
END
HEAT TRANSFER MODEL:
  Option = Thermal Energy
END
THERMAL RADIATION MODEL:
  Number of Histories = 5000
  Option = Monte Carlo
  Radiation Transfer Mode = Surface to Surface
SCATTERING MODEL:
  Option = None
END
SPECTRAL MODEL:
  Option = Gray
END
END
TURBULENCE MODEL:
  Option = SST
BUOYANCY TURBULENCE:
  Option = Production
END
TRANSITIONAL TURBULENCE:
  Option = Gamma Model
  Transition Onset Reynolds Number = 260.0
END
END
TURBULENT WALL FUNCTIONS:
  Option = Automatic
END
END
END
OUTPUT CONTROL:
MONITOR OBJECTS:
MONITOR BALANCES:
  Option = Full
END
MONITOR FORCES:
  Option = Full
END
MONITOR PARTICLES:
  Option = Full
END
MONITOR POINT: AveTemp Gas
  Expression Value = volumeAve(Temperature)@Default Domain
  Option = Expression
END
MONITOR POINT: HF HeatFlow
  Expression Value = areaInt(Wall Heat Flux)@HeatF
  Option = Expression
END
MONITOR POINT: HF HeatReturn
  Expression Value = areaInt(Wall Heat Flux)@HeatR
  Option = Expression
END
MONITOR POINT: HF Net
  Expression Value = areaInt(Wall Heat Flux)@Sleeve+areaInt(Wall Heat Flux)@Product+areaInt(Wall Heat Flux)@HeatF+areaInt(Wall Heat Flux)@HeatR+areaInt(Wall Heat Flux)@Test
  Option = Expression
END
MONITOR POINT: HF Product
  Expression Value = areaInt(Wall Heat Flux)@Product
  Option = Expression
END
MONITOR POINT: HF Sleeve
  Expression Value = areaInt(Wall Heat Flux)@Sleeve
  Option = Expression
END
MONITOR POINT: HF Test
  Expression Value = areaInt(Wall Heat Flux)@Test
  Option = Expression
END
MONITOR POINT: Imbalance Percentage
  Expression Value = (abs(areaInt(Wall Heat Flux)@Sleeve+areaInt(Wall Heat Flux)@Product+areaInt(Wall Heat Flux)@HeatF+areaInt(Wall Heat Flux)@HeatR+areaInt(Wall Heat Flux)@Test)/(abs(areaInt(Wall Heat Flux)@Sleeve)+0.0001 [W]))
  Option = Expression
END
MONITOR POINT: MinTemperature Gas
  Expression Value = minVal(Temperature)@Default Domain
  Option = Expression
END
MONITOR POINT: Temp HeatFlow
  Expression Value = ave(Temperature)@HeatF
  Option = Expression
END
MONITOR POINT: Temp HeatReturn
  Expression Value = ave(Temperature)@HeatR
  Option = Expression
END
MONITOR POINT: Temp Product
  Expression Value = ave(Temperature)@Product
  Option = Expression
END
MONITOR POINT: Temp Sleeve
  Expression Value = ave(Temperature)@Sleeve
  Option = Expression
END
MONITOR POINT: Temp Test
  Expression Value = ave(Temperature)@Test
  Option = Expression
END
MONITOR RESIDUALS:
  Option = Full
END
MONITOR TOTALS:
  Option = Full
END
RESULTS:
  File Compression Level = Default
  Option = Standard
END
SOLVER CONTROL:
  Turbulence Numerics = High Resolution
APPENDICES

ADVECTION SCHEME:
  Option = High Resolution
END

CONVERGENCE CONTROL:
  Length Scale Option = Conservative
  Maximum Number of Iterations = 750
  Minimum Number of Iterations = 1
  Timescale Control = Auto Timescale
  Timescale Factor = 1.0
END

CONVERGENCE CRITERIA:
  Residual Target = 1.E-4
  Residual Type = RMS
END

DYNAMIC MODEL CONTROL:
  Global Dynamic Model Control = On
END

THERMAL RADIATION CONTROL:
  Iteration Interval = 50
END

END

COMMAND FILE:
  Version = 12.0.1
  Results Version = 12.0
END

SIMULATION CONTROL:

EXECUTION CONTROL:

EXECUTABLE SELECTION:
  Double Precision = Yes
END

PARALLEL HOST LIBRARY:
  HOST DEFINITION: cegfh07.ce.ic.ac.uk
    Remote Host Name = ce-gfh07.ce.ic.ac.uk
    Installation Root = /usr/ansys_inc/v%v/CFX
    Host Architecture String = linux-amd64
  END

PARTITIONER STEP CONTROL:
  PARTITIONING TYPE:
    Option = MeTiS
    MeTiS Type = k-way
    Partition Size Rule = Automatic
    Partition Weight Factors = 0.125, 0.125, 0.125, 0.125, 0.125, 0.125, \
    0.125, 0.125
END

RUN DEFINITION:
  Solver Input File = \
    /home/mmohdsal/Taufik/PETBundle/CFX/3d/Inclined/ccl5-15/PETBundle.def
  Run Mode = Full

INITIAL VALUES SPECIFICATION:

INITIAL VALUES CONTROL:
  Use Mesh From = Solver Input File
  Continue History From = Initial Values 1
END

INITIAL VALUES: Initial Values 1
  Option = Results File
  File Name = PETBundle.res
END

SOLVER STEP CONTROL:

PARALLEL ENVIRONMENT:
  Start Method = HP MPI Local Parallel
  Number of Processes = 8
  Parallel Host List = cegfh07.ce.ic.ac.uk*8
END

END

END
APPENDIX 16: ENLARGED FIGURE OF THE VELOCITY PROFILE FOR THE COMPARISON STUDY BETWEEN (A) ANSYS CFX AND (B) STAR-CCM+

(a) Velocity profile plot from ANSYS CFX
(b) Velocity profile plot from STAR-CCM+