CFD Investigation for Turbidity Spikes in Drinking Water Distribution Networks

Alamgir Hossain

A thesis submitted to the
Faculty of Engineering and Industrial Sciences,
Swinburne University of Technology, Australia,
in fulfillment of the requirements for the degree of
Doctor of Philosophy.

August 2005
To my sister Late Rehena Khatun
Abstract

Drinking water distribution networks such as South East Water Ltd. (SEWL), Melbourne Water, Sydney Water, etc. are supposed to transport only dissolved matter rather than a few visible particles. However, it is almost impossible to make the drinking water free from suspended solid particles. The ability to determine the origins of these particles varies between different water supply systems, with possible sources being from catchment, treatment processes, biofilm growth within the water supply pipes, and corrosion products. Improvement of our understanding of the complex hydrodynamic behavior of suspended and/or deposited particles involved in these distribution pipe networks requires mathematical and physical models. Computational Fluid Dynamics (CFD) along with analytical turbulent model is one of the most popular mathematical techniques, which has the ability to predict the behavior of complex flows for such multiphase flow applications.

This study has been completed mainly in two steps. A CFD investigation was carried out to predict the hydrodynamic behavior of turbid particle flowing through a horizontal pipe networks including loop consist of bends and straight pipes. Furthermore, an extended analytical model was re-developed for the liquid-solid system to look at the similar behavior of the solid particles flowing in a turbulent field. These two parallel studies will provide better understandings about the turbidity spikes movements in the distribution networks.
A comprehensive CFD investigation was carried out for particle deposition in a horizontal pipe loop consisting of four 90° bends in a turbulent flow field. A satisfactory agreement was established with the experimental data as validation. This was a steady state multi-particle problem, which helped to understand the deposition characteristics for different particle sizes and densities at upstream and downstream sides of the bends as well as its circumference. Particle concentration was seen high at the bottom wall in the pipe flow before entering the bends, but for the downstream of bend the deposition was not seen high at the bottom as seen in upstream of bend rather inner side of the bend wall (60° skewed from bottom). The larger particles clearly showed deposition near the bottom of the wall except downstream. As expected, the smaller particles showed less tendency of deposition and this was more pronounced at higher velocity. Due to the high streamline curvature and associated centrifugal force acting on the fluid at different depths the particles became well mixed and resulted in homogeneous distribution near the bend regions.

The hydrodynamic behavior of particles flowing in a turbulent unsteady state flowing through a horizontal pipe was also studied to compare with the drinking water distribution networks data. In this numerical simulation six different flow-profiles and particle-load profiles were used to compute particles deposition and re-entrainment into the systems and to identify the conditions of the deposition and suspension mechanisms. Results showed that after a certain length of pipe and period of time after downward velocity gradient, when the velocity was constants over time, the shear stress was sufficiently high enough to cause the particle deposition on and roll along the bottom wall of pipe wall and created a secondary group of particle peak (called kink).
Finally, an extended analytical Turbulent Diffusion Model for liquid-solid phase was developed following an existing gas-liquid turbulence model. This turbulent diffusion model was then compared with the results of the CFD investigation making use of the same boundary conditions. The comparison established good agreement between these two models. The influence of velocity on the particle size distribution was dominant over the influence of the superficial liquid velocity, which was also explained by using the new parameter, velocity ratio. This velocity ratio was defined as the ratio between the free flight and gravitational velocity. Due to some inevitable assumptions used in the analytical model, the results showed typically less deposition as compared with the CFD investigation.
Acknowledgements

The author would like to acknowledge the supportive efforts of the many people and organizations that contributed to this research.

Dr. Jamal Naser, my supervisor, whose knowledge, patient teaching, and subtle direction guided me through this project and provided me with a great introduction into the field of research. I would like to thank him for such constant supervision in on and off campus.

I would like to acknowledge the South East Water Ltd., who provided the partial financial support to undertake this project. I would like to thank CSIRO for supporting and making the experimental data available to me, without that I could not authenticate my work. Clive Grainger, team leader of the experiment, CSIRO, is also deeply acknowledged for his important advices and support.

My beloved Mum and Dad, you were always encouraging and guiding me while I was away from you, which drove me forward. My sisters, brothers, and sister-in-laws, your inspirations helped me to endure the long journey.

All my friends inside and outside of the university, who I don’t know they helped and inspired me, but they did.

Alamgir Hossain
Declaration of Originality

This thesis contains no material, which has been accepted for the award of any other degree in any university or institute of education; nor, to my best knowledge, does it contain material previously published by any other person, except where due reference is made in the body of the thesis.

Signed ______________

[Alamgir Hossain]
## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abstract</td>
<td>iii</td>
</tr>
<tr>
<td>Acknowledgements</td>
<td>vi</td>
</tr>
<tr>
<td>Declaration of Originality</td>
<td>vii</td>
</tr>
<tr>
<td>Contents</td>
<td>viii</td>
</tr>
<tr>
<td>List of Figures</td>
<td>xiii</td>
</tr>
<tr>
<td>List of Tables</td>
<td>xx</td>
</tr>
</tbody>
</table>

### Chapter 1: Introduction

1.1 Background .................................................................................. 1
1.2 Numerical Investigation ................................................................ 2
1.3 Analytical Model .......................................................................... 2
1.4 The Objectives Of This Research ............................................... 3
1.5 Thesis Outline ............................................................................. 4

### Chapter 1: Literature Review

2.1 Introduction .................................................................................. 6
2.1.1 Methods of Investigation ..................................................... 7
   Experimental investigation ......................................................... 8
   Theoretical Calculation ............................................................. 8
   Advantages of a Theoretical Calculation ..................................... 9
   Disadvantages of a Theoretical Calculation ................................ 10
   Choice of Investigation in this Research .................................... 11
2.2 Particle Deposition in a Pipe Loop ............................................ 12
2.3 Turbidity Spikes Movement ......................................................... 14
2.4 Particle Dispersion and Deposition Model .................................... 17
2.5 Numerical Simulation .................................................................... 20
2.6 Conclusion .................................................................................. 22
## Chapter 3: Governing Equations and Numerical Method

### 3.1 Introduction

### 3.2 Continuity and Momentum Equations

- 3.2.1 Definition of a Differential Equation
- 3.2.2 The General Differential Equation
- 3.2.3 The Mass Conservation Equation
- 3.2.4 Momentum Conservation Equations

### 3.3 Modelling Multiphase Flows

- 3.3.1 Multiphase Flow Regimes
  - Gas-liquid or liquid-liquid flows
  - Gas-solid flows
  - Liquid-solid flows
- 3.3.2 Examples of Multiphase Systems
- 3.3.3 Approaches to Multiphase Modelling
  - The Euler-Lagrange Approach
  - The Euler-Euler Approach
  - The VOF Model
  - The Mixture Model
  - The Eulerian Model
- 3.3.4 Choosing a Multiphase Model
  - Detailed Guidelines
  - The Effect of Particulate Loading
  - The Significance of the Stokes Number

### 3.4 The Multiphase Mixture Model

- 3.4.1 Overview of the Mixture Model
- 3.4.2 Limitations of the Mixture Model
- 3.4.3 Continuity Equation for the Mixture
- 3.4.4 Momentum Equation for the Mixture
- 3.4.5 Relative (Slip) Velocity and the Drift Velocity
- 3.4.6 Volume Fraction Equation for the Secondary Phases

### 3.5 Turbulent Modelling

- 3.5.1 Turbulence Model Selection for this Research
  - Reynolds-Averaged Approach vs. LES
Reynolds (Ensemble) Averaging ............................................. 48
Boussinesq Approach vs. Reynolds Stress Transport Models ....... 49
3.5.2 The Spalart-Allmaras Model ........................................ 49
3.5.3 The Standard k-ε Model ............................................. 50
3.5.4 The Standard k-ω Model ............................................. 51
3.5.5 The Reynolds Stress Model (RSM) ................................. 51
3.5.6 Computational Effort: CPU Time and Solution Behavior .... 52

3.6 The Spalart-Allmaras Model ............................................ 53
3.6.1 Transport Equation for the Spalart-Allmaras Model .......... 53
3.6.2 Modelling the Turbulent Viscosity ................................. 54
3.6.3 Modelling the Turbulent Production ............................... 54
3.6.4 Modelling the Turbulent Destruction .............................. 56
3.6.5 Model Constants ...................................................... 56
3.6.6 Wall Boundary Conditions ........................................... 57

3.7 Turbulent Boundary Conditions ...................................... 57
3.7.1 Flow Inlets and Exits ................................................ 57
Using Flow Boundary Conditions ........................................ 57
Determining Turbulence Parameters ................................... 59
3.7.2 Uniform Specification of Turbulence Quantities ............... 60
Turbulence Intensity ......................................................... 61
Turbulence Length Scale and Hydraulic Diameter .................. 61
Turbulent Viscosity Ratio .................................................. 62
3.7.3 Relationships for Deriving Turbulence Quantities ............. 63
Estimating Modified Turbulent Viscosity from Turbulence
Intensity and Length Scale ............................................... 63

3.8 Numerical Simulation and Geometry Construction ........... 63
3.8.1 Commercial Software ............................................... 63
3.8.2 The Grid Topology ................................................... 64
Choosing the Appropriate Grid Type ................................... 65
Setup Time ......................................................................... 65
Computational Expense .................................................... 66
Numerical Diffusion .......................................................... 66
3.8.3 Grid Requirements and Considerations .......................... 67
Geometry or Grid Requirements .......................................... 68
Mesh Quality ................................................................. 68
Node Density and Clustering ........................................... 68
Smoothness .................................................................... 70
Cell Shape ....................................................................... 70
Flow-Field Dependency .................................................. 70

3.9 Conclusion .................................................................. 71

Chapter 4: Particle Deposition in a Pipe Loop ............ 72-93
4.1 Introduction .............................................................. 72
4.2 Numerical Technique ................................................. 74
4.3 Result and Discussion ................................................. 75
   4.3.1 Model Validation .................................................... 75
   4.3.2 Particle Concentration at Different Depths across the
       Pipe ........................................................................... 78
   4.3.3 Circumferential Particle Deposition at Bends ............. 83
   4.3.4 Circumferential deposition at 3rd bend .................... 87
4.4 Conclusion .................................................................. 93

Chapter 5: Turbidity Spikes Movement ....................... 94-118
5.1 Introduction .............................................................. 94
5.2 Background .............................................................. 95
   5.2.1 Definition of Turbidity ............................................. 96
   5.2.2 Formation of a Turbidity Spike ............................... 96
5.3 Numerical Simulation of Turbidity Spikes ................ 99
   5.3.1 Online Monitoring Data ......................................... 99
   5.3.2 Velocity and Particle-Load Profiles ......................... 101
   5.3.3 Geometry and Boundary Conditions ....................... 104
5.4 Results and Discussion .............................................. 105
   5.4.1 Average Particle Distribution along the Pipe .......... 105
   5.4.2 Average Particle Deposition at Different Cross-sections .... 111
   5.4.3 Gravity Deposition and Kink Formation ................ 115
      Kink Formation ......................................................... 116
5.5 Conclusion .................................................................. 117

Chapter 6: Turbulent Diffusion and Dispersion Model ... 119-196
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1</td>
<td>Introduction</td>
<td>119</td>
</tr>
<tr>
<td>6.2</td>
<td>Analytical Turbulent Diffusion Model</td>
<td>122</td>
</tr>
<tr>
<td>6.2.1</td>
<td>Definition of the turbulent diffusion problem</td>
<td>127</td>
</tr>
<tr>
<td>6.2.2</td>
<td>Mathematical formulation of the model</td>
<td>131</td>
</tr>
<tr>
<td>6.3</td>
<td>Analytical Solution of the Problem</td>
<td>140</td>
</tr>
<tr>
<td>6.3.1</td>
<td>One Dimensional Problem</td>
<td>140</td>
</tr>
<tr>
<td>6.3.2</td>
<td>Extension to a two-dimensional deposition flux comparison with a semi empirical correlation</td>
<td>146</td>
</tr>
<tr>
<td>6.4</td>
<td>Cfd Investigation and Results Comparison</td>
<td>150</td>
</tr>
<tr>
<td>6.4.1</td>
<td>Relative Concentration along Height</td>
<td>150</td>
</tr>
<tr>
<td>6.4.2</td>
<td>Circumferential Deposition for the CFD Simulation</td>
<td>154</td>
</tr>
<tr>
<td>6.5</td>
<td>Conclusion</td>
<td>157</td>
</tr>
</tbody>
</table>

**Chapter 7: Conclusion and Recommendations**

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.1</td>
<td>Conclusion</td>
<td>160</td>
</tr>
<tr>
<td>7.2</td>
<td>Recommendations</td>
<td>162</td>
</tr>
</tbody>
</table>

**Chapter 8: References**

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Chapter 8: References</td>
<td>162-170</td>
</tr>
</tbody>
</table>

**List of Personal Publications**

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>List of Personal Publications</td>
<td>171</td>
</tr>
</tbody>
</table>
# List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 3-1</td>
<td>Flux balance over a control volume.</td>
<td>26</td>
</tr>
<tr>
<td>Figure 3-2</td>
<td>Multiphase flow regimes.</td>
<td>30</td>
</tr>
<tr>
<td>Figure 4-1c</td>
<td>Schematic diagram of the pipe loop with four 90° bends.</td>
<td>73</td>
</tr>
<tr>
<td>Figure 4-2</td>
<td>Comparison of CFD results and experimental data (Grainger et al. 2003) for the velocity 0.4 ms⁻¹ at center of the pipe (D is the diameter of pipe).</td>
<td>75</td>
</tr>
<tr>
<td>Figure 4-3</td>
<td>Comparison of CFD results and experimental data (Grainger et al. 2003) for different velocities at center of the pipe.</td>
<td>77</td>
</tr>
<tr>
<td>Figure 4-4</td>
<td>Contour of volume fraction of total particles at different heights (a, b, c, d, e, and f are for at 1, 5, 10, 20, 30, and 50 µm measured from bottom, respectively) for the velocity of 0.05 ms⁻¹.</td>
<td>79</td>
</tr>
<tr>
<td>Figure 4-5</td>
<td>Velocity vector on a plane at different heights (a, b, c, and d are at 5, 10, 30, and 50 µm measured from bottom, respectively) at velocity 0.05 ms⁻¹.</td>
<td>80</td>
</tr>
<tr>
<td>Figure 4-6</td>
<td>Velocity vector on a plane at different heights (a, b, c, and d are at 5, 10, 30, and 50 µm measured from bottom, respectively) at velocity 0.4 ms⁻¹.</td>
<td>80-81</td>
</tr>
<tr>
<td>Figure 4-7</td>
<td>CFD investigation of particle concentration as function of velocity at different heights across the pipe at the measuring plane.</td>
<td>81</td>
</tr>
<tr>
<td>Figure 4-8a</td>
<td>Relative concentration of particles for different depths along the pipe at 0.05 ms⁻¹.</td>
<td>82</td>
</tr>
<tr>
<td>Figure 4-8b</td>
<td>Relative concentration of particles for different depths along the pipe at 0.4 ms⁻¹.</td>
<td>83</td>
</tr>
</tbody>
</table>
Figure 4-9: Vertical cross-section of a pipe.

Figure 4-10a: Cumulative particle deposition as a function of circumferential pipe angles at three different up and down stream of bends at 0.05 ms\(^{-1}\).

Figure 4-10b: Cumulative particle deposition as a function of circumferential pipe angles at three different up and down stream of bends at 0.1 ms\(^{-1}\).

Figure 4-10c: Cumulative particle deposition as a function of circumferential pipe angles at three different up and down stream of bends at 0.2 ms\(^{-1}\).

Figure 4-10d: Cumulative particle deposition as a function of circumferential pipe angles at three different up and down stream of bends at 0.3 ms\(^{-1}\).

Figure 4-10e: Cumulative particle deposition (CFD) as a function of circumferential pipe angles at three different up and down stream of bends at 0.4 ms\(^{-1}\).

Figure 4-11: Schematic diagram of a bend.

Figure 4-12: Contour plot of particle volume fraction on three different cross-sectional planes for the velocity of 0.05, 0.1, 0.2, 0.3, and 0.4 ms\(^{-1}\) respectively.

Figure 4-13a: Particle deposition as a function of circumferential pipe angles \(\theta\) at different planes of 3\(^{rd}\) bend at 0.05 ms\(^{-1}\).

Figure 4-13b: Particle deposition as a function of circumferential pipe angles \(\theta\) at different planes of 3\(^{rd}\) bend at 0.1 ms\(^{-1}\).

Figure 4-13c: Particle deposition as a function of circumferential pipe angles \(\theta\) at different planes of 3\(^{rd}\) bend at 0.2 ms\(^{-1}\).

Figure 4-13d: Particle deposition as a function of circumferential pipe angles \(\theta\) at different planes of 3\(^{rd}\) bend at 0.3 ms\(^{-1}\).

Figure 4-13e: Particle deposition as a function of circumferential pipe angles \(\theta\) at different planes of 3\(^{rd}\) bend at 0.4 ms\(^{-1}\).
Figure 4-14a: Particle deposition (CFD) as a function of circumferential pipe angles at the up and down stream of 3rd bend for different particles at 0.05 ms\(^{-1}\). .................................................. 91

Figure 4-14b: Particle deposition (CFD) as a function of circumferential pipe angles at the up and down stream of 3rd bend for different particles at 0.1 ms\(^{-1}\). .................................................. 92

Figure 4-14c: Particle deposition (CFD) as a function of circumferential pipe angles at the up and down stream of 3rd bend for different particles at 0.2 ms\(^{-1}\). .................................................. 92

Figure 4-14d: Particle deposition (CFD) as a function of circumferential pipe angles at the up and down stream of 3rd bend for different particles at 0.3 ms\(^{-1}\). .................................................. 92

Figure 4-14e: Particle deposition (CFD) as a function of circumferential pipe angles at the up and down stream of 3rd bend for different particles at 0.4 ms\(^{-1}\). .................................................. 92

Figure 5-1: Flow rate and turbidity spikes in particular days for two monitoring stations (MS1 & MS2) (Prince et al. 2001).................... 93

Figure 5-2: Different arbitrary velocity and particle load profiles for the spike................................................................. 100

Figure 5-3: Schematic diagram of the horizontal straight pipe. ........ 102-3

Figure 5-4a: Average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2a. ......................... 106

Figure 5-4b: Average particle distribution along the pipe at different times (60-150 minutes) for the profile of 5-2a.......................... 106

Figure 5-5a: Average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2b. ......................... 107

Figure 5-5b: Average particle distribution along the pipe at different times (60-150 minutes) for the profile of 5-2b......................... 107

Figure 5-6a: Average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2c. ......................... 108
Figure 5-6b: Average particle distribution along the pipe at different times (60-150 minutes) for the profile of 5-2c. ........................................... 108

Figure 5-7a: Average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2d. ........................................... 109

Figure 5-7b: Average particle distribution along the pipe at different times (60-150 minutes) for the profile of 5-2d. ........................................... 109

Figure 5-8a: Average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2e. ........................................... 110

Figure 5-8b: Average particle distribution along the pipe at different times (60-150 minutes) for the profile of 5-2e. ........................................... 110

Figure 5-9a: Average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2f. ........................................... 111

Figure 5-9b: Average particle distribution along the pipe at different times (60-150 minutes) for the profile of 5-2f. ........................................... 111

Figure 5-10: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 5-2a. ........................................... 112

Figure 5-11: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 5-2b. ........................................... 112

Figure 5-12: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 5-2c. ........................................... 113

Figure 5-13: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 5-2d. ........................................... 113

Figure 5-14: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 5-2e. ........................................... 113

Figure 5-15: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 2f. ........................................... 114
Figure 5-16: Particle concentration at different depths along the pipe length at different time for the profile of 5-2e (a, b, and c for bottom, center, and top respectively). ......................................................... 115

Figure 6-1: The entrainment/deposition mechanism in a horizontal pipe. .......................................................................................... 120

Figure 6-2: The deposition flux in a tube as a function of the circumferential angle in the pipe used by Laurinat (1985). .................... 124

Figure 6-3: Vertical cross-section of a pipe. .................................. 125

Figure 6-4: The diffusion/free-flight problem between two infinite horizontal plates. ................................................................. 127

Figure 6-5a: Ratio between pipe diameter and particle mean free path as a function of particle diameter at 0.1 and 0.5 ms\(^{-1}\) fluid velocity \((D = 4.310^{-1}\text{ m})\). ................................................................. 133

Figure 6-5b: Ratio between pipe diameter and particle mean free path as a function of particle density at 0.1 and 0.5 ms\(^{-1}\) fluid velocity \((D = 4.72310^{-1}\text{ m})\). ................................................................. 133

Figure 6-6a: The inertial and crossing trajectories coefficients as a function of particle diameter for the velocity 0.1 and 0.5 ms\(^{-1}\). ......... 137

Figure 6-6b: The inertial and crossing trajectories coefficients as a function of particle specific gravity for the velocity 0.1 and 0.5 ms\(^{-1}\). ... 137

Figure 6-7a: Relative concentration of particles for different height as a function of particle diameter for the velocity 0.1 ms\(^{-1}\). ............... 143

Figure 6-7b: Relative concentration of particles for different height as a function of specific gravity for the particle of 10 \(\mu\text{m}\) at velocity 0.1 ms\(^{-1}\). ................................................................................................................ 144

Figure 6-8a: Relative concentration of particles for different height as a function of particle diameter for the velocity 0.5 ms\(^{-1}\). .............. 144

Figure 6-8b: Relative concentration of particles for different height as a function of specific gravity for the particle of 10 \(\mu\text{m}\) at velocity 0.5 ms\(^{-1}\). ................................................................................................................ 144
Figure 6-9a: Comparison of the relative concentration of particles between top and bottom of the pipe as a function of the particle diameter at 0.1 and 0.5 ms\(^{-1}\). ................................................................. 145
Figure 6-9b: Comparison of the relative concentration of particles between top and bottom of the pipe as a function of the particle density at 0.1 and 0.5 ms\(^{-1}\). ................................................................. 146
Figure 6-10a: Deposition flux normalized by \(C_E(\tau_p)\) vs circumferential pipe angle for different particle sizes (5, 10, and 20 µm) for the velocity 0.1 and 0.5 ms\(^{-1}\). ................................................................. 149
Figure 6-10b: Deposition flux normalized by \(C_E(\tau_p)\) vs circumferential pipe angle for different particle sizes (50, 100, and 200 µm) at 0.1 and 0.5 ms\(^{-1}\). ........................................................................ 149
Figure 6-10c: Deposition flux normalized by \(C_E(\tau_p)\) vs circumferential pipe angle for five different particle densities (sg 1.5, 2.5, 4, 5, and 6) at 0.1 and 0.5 ms\(^{-1}\). ................................................................. 149
Figure 6-11a: Comparison of analytical and simulated relative concentration as a function of particle diameter at the top of pipe at 0.1 and 0.5 ms\(^{-1}\). ........................................................................ 151
Figure 6-11b: Comparison of analytical and simulated relative concentration as a function of particle diameter at \(y = 0.75D\) at 0.1 and 0.5 ms\(^{-1}\). ........................................................................ 151
Figure 6-11c: Comparison of analytical and simulated relative concentration as a function of particle diameter at the center of pipe at 0.1 and 0.5 ms\(^{-1}\). ........................................................................ 151
Figure 6-11d: Comparison of analytical and simulated relative concentration as a function of particle diameter at \(y = 0.25D\) at 0.1 and 0.5 ms\(^{-1}\). ........................................................................ 152
Figure 6-12a: Comparison of analytical and simulated relative concentration as a function of particle weight at the top of the pipe for velocity of 0.1 and 0.5 ms\(^{-1}\). ................................................................. 152
Figure 6-12b: Comparison of analytical and simulated relative concentration as a function of particle density at y = 0.75D at 0.1 and 0.5 ms\(^{-1}\). ................................................................. 153

Figure 6-12c: Comparison of analytical and simulated relative concentration as a function of particle density weight at the center of the pipe at 0.1 and 0.5 ms\(^{-1}\). ................................................................. 153

Figure 6-12d: Comparison of analytical and simulated relative concentration as a function of particle density at y = 0.25D at 0.1 and 0.5 ms\(^{-1}\). ................................................................. 153

Figure 6-13a: Circumferential deposition as a function of circumferential pipe angles for five different particle sizes for the velocity 0.1 ms\(^{-1}\). ................................................................. 155

Figure 6-13b: Circumferential deposition as a function of circumferential pipe angles for five different particle sizes for the velocity 0.5 ms\(^{-1}\). ................................................................. 155

Figure 6-13c: Circumferential deposition as a function of circumferential pipe angles for five different particle densities for the velocity 0.1 ms\(^{-1}\). ................................................................. 155

Figure 6-13d: Circumferential deposition as a function of circumferential pipe angles for five different particle densities for the velocity 0.5 ms\(^{-1}\). ................................................................. 156
List of Tables

Table 3-1: General guidelines for flow types................................................. 36
Table 4-1: Physical and hydraulic characteristics of the system used for CFD simulation.............................................................. 74
Table 5-1: Physical and hydraulic characteristics of the system used for CFD simulation.............................................................. 105
Table 6-1: Properties of the different size particles with specific gravity 3.0 that are used in the Turbulent Diffusion Model. ............... 141
Table 6-2: Properties of the different weighted particles with diameter 10 \( \mu \text{m} \) that are used in the Turbulent Diffusion Model. ....... 141
Table 6-3: Properties of the fluid that are used in the Turbulent Diffusion Model................................................................. 141
Table 6-4: Gravitational settling velocity, \( v_g \) and its ratio to free-flight velocity, \( v \). ................................................................. 143
Table 6-5: Physical and hydraulic characteristics of the system used for CFD simulation............................................................. 150
Chapter 1
Introduction

1.1 BACKGROUND
Drinking water distribution networks are supposed to transport only dissolved matter rather than suspended solid particles. However, it is not possible to make the drinking water free from any suspended solid particles. The ability to determine the origins of these particles varies between different water supply systems, with possible sources being from catchment, treatment processes, biofilm growth within the water supply pipes, and corrosion products (Jayaratne et al. 2004).

Turbidity, which is, in general, the measure of particles content in water, is the main cause of discolouration of water. The distribution systems in Australia receive less than 2 NTU of turbidity, which is far less than international standard (5 NTU). However, when the level of turbidity exceeds 5 NTU, the discolouration of the water is visible to naked eyes (National Water Quality Management Strategy 1996). According to NWQMS (National Water Quality Management Strategy) the water becomes discoloured just after the average background datum 1.4 NTU, but a major discoloured water event was defined as having a maximum turbidity exceeding 2.6 NTU under normal circumstances. This very small amount of particles can accumulate and settle down at the bottom wall or any pockets like dead ends, T-junctions, bends of the pipe networks, or a combination of these. During any abnormal flow (e.g. during wash out, pipe burst, etc.), these particles re-entrain into the system at a
time under high turbulence, which create turbidity. There are also some other causes of turbidity spikes, which have not yet been identified. Complains come often when these distribution networks experienced turbidity spikes at anywhere of the systems due to numerous causes. These spikes travel along the pipe networks and into households resulting in dirty water. To drink such discoloured water caused by the turbidity spikes is not safe.

1.2 NUMERICAL INVESTIGATION
Computational Fluid Dynamics (CFD) is a well established technique for numerical simulation by which any flow behavior can be described. The advantage of high-speed and large-memory computers has enabled CFD to obtain solutions for many flow problems including compressible or incompressible, laminar or turbulent, reacting or non-reacting, single phase or multiphase, and steady or transient flows. Multiple combinations of the above flows can also be solved with the help of CFD.

By using CFD simulation technique the turbid solid particles, which are admitted into distribution systems, can be tracked. This technique is also helpful to identify the location of the most likely place of accumulation and/or deposition of such particles and time of re-entrainment of them into the system. The CFD investigation is a fast, easy to scale, easily accessible and relatively low cost technique, whereas experiments are time consuming, expensive, sometimes non-accessible, and not free from instrumental or human errors. However, to establish numerical results, experimental data are required for model validation.

1.3 ANALYTICAL MODEL
Analytical model is an alternative to CFD investigation, which can provide detail hydrodynamics of particles in a flow field or any physical problems. It is also a versatile, flexible, and cost effective
technique. Researchers have been using this technique proficiently since early days of mathematics.

The combination of CFD analysis and analytical model along with experiment data can confirm a detailed, complete, and accurate physical parameters for any systems like drinking water distribution networks.

1.4 THE OBJECTIVES OF THIS RESEARCH

Turbidity in drinking water is mainly due to the presence of suspended solids such as clay particles or silt. The suspended particles that are introduced into the water distribution networks, are transported and modified by a number of physical and chemical forces. In order to develop an accurate model, it is of prime importance to understand and accurately describe the various phenomena affecting the development of turbidity spikes. The proposed models include a comprehensive description of hydrodynamic phenomena of flow regimes in a pipeline system and the physical phenomena of particle settling, deposition and re-suspension for straight, bend, and/or joints pipes. Since colloidal particles can possess a surface charge, water chemistry (pH, ionic concentration) can affect the stability of suspended particles. For example, the settling rates of colloidal particles may be enhanced through the formation of aggregates as a result of chlorination. Biofilm growth and charge particles may also affect the deposition rates and their re-entrainment near pipe wall or any dead ends.

Detailed experimental investigation in a real-size water distribution network to track the transition of the turbidity spike is expensive and time consuming and in some cases impossible. Therefore, the CFD (Computational Fluid Dynamics) investigation is the most widely known, effective, and appropriate technique that was used in this research work.
This research consisted of developing an extended analytical turbulent diffusion model, CFD investigation for turbid particle using same parameters for both steady and unsteady flow domain, and compared the results with experimental data and analytical solutions.

The entire body of the research work was designed to carry out the following major tasks:

i. A CFD investigation of particle deposition along length as well as at periphery wall flowing though a horizontal pipe loop consisting of four 90° bends in a turbulent flow field and a comparison of the results with the experimental data.

ii. A CFD investigation of particle deposition in a straight horizontal pipe for transient flow using the arbitrary inlet velocity and particle load profiles.

iii. Re-development of an extended analytical turbulent diffusion model in liquid-solid phases emanated from a gas-solid turbulent model of Mols and Oliemans (1998). A comparison was then established between the results of analytical turbulent diffusion model and the results of CFD investigation using the same physical geometry and boundary conditions.

iv. Furthermore, the author has also intension to model the biofilm growth with detail chemistry and the charge particle in fluid flow, which play significant role causing turbidity spikes in drinking water distribution networks.

1.5 THESIS OUTLINE
In chapter 1 a short introduction, covering the background, reasons for the research program, and the objectives of this thesis has been given. In chapter 2 a short overview of previous works on analytical
and numerical modelling techniques applied to this field has been presented.

Chapter 3 presents the governing equations that are used for the numerical simulation. Navier Stokes for continuity and momentum along with turbulence transport equations are presented here.

In chapter 4 a comprehensive CFD investigation of particle deposition in a horizontal pipe loop consisting of four 90° bends in a turbulent and steady flow field is covered. This study deals with different particle sizes and densities, which is then validated with experimental data in this chapter.

Chapter 5 represents another CFD investigation of particle deposition in a straight horizontal pipe flow using different time dependent velocity and particle-load profiles collecting from current online monitoring data of SEWL’s distribution networks.

An extended analytical turbulent diffusion model is presented in Chapter 6, which has been developed for liquid-solid phases. This model has emanated from a gas-solid turbulent model developed by Mols and Oliemans (1998). The comparison between analytical model results and the that of CFD investigation making use of same parameters has been presented in this chapter.

A brief conclusion along with future recommendations has been presented in Chapter 7. The references followed by a list of publications that come up with this research are given at the end of the thesis.
Chapter 2

Literature Review

2.1 INTRODUCTION

This thesis was primarily aimed at CFD (Computational Fluid Dynamics) investigation of solid particles in the drinking water distribution networks. This problem was associated with mass transfer in a turbulent field through horizontal pipe for both steady and transient flows. As it was revealed, among the different methods of prediction, the numerical and analytical solutions offer great promise. In this thesis, a multiphase numerical investigation and analytical solutions had been constructed for predicting the hydrodynamic behavior of the particle.

This ambitious task could not, of course, be accomplished in a modest-sized thesis without leaving out a number of important topics. Therefore, the mathematical formulation of the equations that govern the hydrodynamic behavior of the particle was discussed only briefly in this thesis. For the complete derivation of the required equations, the reader must turn to standard textbooks on the subject (Patankar 1980; Stephen 2000; Markatos 1986). The mathematical models for complex processes like turbulence were assumed to be known or available to the reader. Even in the subject of numerical solutions, all available methods were not surveyed and discussed. However, the author has used, developed, or contributed to the subject of numerical solutions in a particular family of methods. References to other methods were made only when this served to highlight a certain issue. While a general formulation was
attempted, special attention was given to multiphase flow and analytical solutions only, which described the physical problem of suspended solid particles causing turbidity spikes in drinking water distribution networks effectively. Different boundary conditions and geometries have been used in CFD investigations and extended analytical turbulent diffusion model in this thesis to understand the hydrodynamics of particles flowing through horizontal pipe networks.

An important characteristic of the CFD investigation to be used in this thesis was that they were strongly based on the physical considerations, not just on mathematical manipulations. Indeed, nothing was more sophisticated than simple algebra and elementary calculus used in the CFD investigation. A significant advantage of the strategy was that the reader, while learning about the numerical investigation, developed a deeper analytical understanding of, and insight into, the underlying physical processes. This appreciation for physical significance was very helpful in analysing and interpreting computed results. But, even if the reader never performs numerical computation, this study of the numerical investigation and analytical solution will provide – it is interesting to note – a greater feel for the physical aspects of the fluid flow. Furthermore, the physical approach will equip the reader with general criteria with which to judge other existing and future numerical investigation and analytical solutions in this field.

2.1.1 Methods of Investigation

Investigation of fluid-flow processes can be obtained by two main methods: the experimental investigation and theoretical calculation. In this thesis CFD investigation and theoretical calculation for multiphase flow were carried out and then compared them with the available experimental results. The results of CFD investigation were also compared to that of analytical turbulent diffusion model.


**Experimental investigation**

The most reliable information about a physical process is often given by the actual measurement. An experimental investigation involving full-scale equipment can be used to predict how identical copies of the equipment would perform under the same conditions. Such full-scale tests are in most cases, prohibitively expensive and often impossible. The alternative then is to perform experiments on small-scale models. The resulting information, however, must be extrapolated to full scale, and general rules for doing this are often unavailable. Further, the small-scale models do not always simulate all the features of the full-scale equipment; frequently, important features such as turbulence flow field of water distribution networks, especially underground pipeline for fluid flow, are omitted from the model tests. This further reduces the usefulness of the test results. Finally, it must be remembered that there are serious difficulties of measurement in many situations, and that the measuring instruments are not free from errors.

**Theoretical Calculation**

A theoretical prediction works out the consequences of a mathematical model, rather than those of an actual physical model. For the physical processes of interest here, the mathematical model mainly consists of a set of differential equations. If the methods of classical mathematics were to be used for solving these equations, there would be little hope of predicting many phenomena of practical interest. A look at a classical text on fluid mechanics leads to the conclusion that only a tiny fraction of the range of practical problems can be solved in closed form. Further, these solutions often contain infinite series, special functions, transcendental equations for eigenvalues, etc., so that their numerical evaluation may present a formidable task.
Advantages of a Theoretical Calculation

The advantages that a theoretical calculation (CFD investigation and analytical solution) offers over a corresponding experimental investigation are listed as follows:

*Low cost:* The most important advantage of a computational prediction is its low cost. In most applications, the cost of a computer run is many orders of magnitude lower than the cost of a corresponding experimental investigation. This factor assumes increasing importance as the physical situation to be studied becomes larger and more complicated. Further, whereas the prices of most items are increasing, computing costs are likely to be even lower in the future.

*Speed:* A computational investigation can be performed with remarkable speed. A designer can study the implications of hundreds of different configurations in less than a day and choose the optimum design. On the other hand, a corresponding experimental investigation, it is easy to imagine, would take a very long time.

*Complete information:* A computer solution of a problem gives detailed and complete information. It can provide the values of all the relevant variables (such as velocity, pressure, temperature, concentration, turbulence intensity) throughout the domain of interest. Unlike the situation in an experiment, there are few inaccessible locations in a computation, and there is no counterpart to the flow disturbance caused by the probes. Obviously, no experimental study can be expected to measure the distributions of all variables over the entire domain. For this reason, even when an experiment is performed, CFD solutions have a great value to supplement the experimental information.
Ability to simulate realistic conditions: In a theoretical calculation, realistic conditions can be easily simulated. There is no need to resort to small-scale or cold-flow models. For a computer program, there is little difficulty in having very large or very small dimensions, in treating very low or very high temperatures, in handling toxic or flammable substances, or in following very fast or very slow processes.

Ability to simulate ideal conditions: A prediction method is sometimes used to study a basic phenomenon, rather than a complex engineering application. In the study of a phenomenon, one wants to focus attention on a few essential parameters and eliminate all irrelevant features. Thus, many idealizations are desirable— for example, two-dimensionality, constant density, an adiabatic surface, or infinite reaction rate. In a computation, such conditions can be easily and exactly set up. On the other hand, even a very careful experiment can barely approximate the idealization.

Disadvantages of a Theoretical Calculation

The foregoing advantages are sufficiently impressive to stimulate enthusiasm about computer analysis. A blind enthusiasm for any cause is, however, undesirable. It is useful to be aware of the drawbacks and limitations.

As mentioned earlier, a computer analysis works out the implications of a mathematical model. The experimental investigation, by contrast, observes the reality itself. The validity of the mathematical model, therefore, limits the usefulness of a computation. In this thesis, we shall be concerned only with computational investigation and analytical solutions of the physical problems and not with mathematical models. Yet, we must note that the user of the computer analysis receives an end product that
depends on both the mathematical model and the numerical investigation. A perfectly satisfactory numerical technique can produce worthless results if an inadequate mathematical model is employed.

**Choice of Investigation in this Research**

This research work consists of an extended analytical turbulent diffusion model development, CFD investigations using different boundary conditions and geometries for both steady and unsteady flow domain. These results obtained from both studies have been compared to each other and validated with the experimental data. Thus, CFD investigations have played a key role in this research. A striking example of this role can be found in the recent development of turbulence diffusion models. The popular and widely used turbulence diffusion models are primarily based on the work of Laurinat *et al.* (1985) and Anderson and Russell (1970b). The extended analytical turbulent model of this research work was also based on the work of the above authors.

Furthermore, the CFD investigation was carried out making use of widely used commercial software Fluent 6.1 (2001). Among them water is considered as the primary phase and solid sperical particles as secondary phases. Five different types of solid particles were chosen to represent the secondary phases; this was deemed sufficient to accurately represent the turbidity condition of SEWL’s distribution networks.

The entire body of the research work was divided into three major components, which are delineated in the following manner:

i. A comprehensive CFD investigation of particle deposition in a horizontal pipe loop consisting of four $90^0$ bends in a turbulent flow field and the results were then validated with the experimental data,
ii. A comprehensive CFD investigation of particle deposition in a straight horizontal pipe for transient flow using the arbitrary inlet velocity and particle load profiles, and finally

iii. Development of an extended analytical turbulent diffusion model for liquid-solid phases emanated from a gas-solid turbulent model of Mols and Oliemans (1998) and then the results were compared with that of CFD investigation.

2.2 PARTICLE DEPOSITION IN A PIPE LOOP

Deposition of particles from flowing suspensions is an important process in various fields of engineering as well as in nature. Analysing of the deposition of small particles suspended in fluid streams has attracted considerable attention in the past three decades (Anderson and Russell 1970b; Anderson and Russell 1970a; David et al. 1987; Swailes and Reeks 1994; Thomson 2003; Laurinat et al. 1985; Abuzeid et al. 1991; Grainger et al. 2003; Hossain et al. 2003; Hossain et al. peer review). This is because particle deposition plays a major role in a number of industrial processes such as filtration, separation, particle transport, combustion, air and water pollution, and many others.

Computational models for simulating the hydraulic behavior of water-distribution systems have been available for many years (Hossain et al. 2003; Hossain et al. peer review; Mols and Oliemans 1998). More recently these models have been extended to analyze water quality (Hossain et al. 2003; Hossain et al. peer review). Computational models have suffered from some serious limitations. The number of particles which can be modelled is one of them. However, in recent past, multiphase modelling technique has been extended where more than two phases can be introduced. When the number of particle, which can be modelled is fairly small the effect of particle sizes becomes important, especially at low particle concentrations. In this study we have introduced water as a primary
phase and 5 different solid spherical particles as secondary phases. The driving force behind this trend is the timely challenge to comply with increasingly stringent governmental regulations and customer-oriented expectations. Modern management of water-distribution systems like South East Water Ltd, Melbourne Water, Sydney Water, need simulation models that are able to accurately predict the hydrodynamics of particle behavior (cause of dirty water) in the water distribution networks.

Deposition of small particles on surfaces flowing domain i.e. pipe wall or floor in turbulent flows has attracted the interest of many researchers. Using the stopping distance of a particle near a wall, Friendlander and Johnstone (1957) developed the free-flight model for particle deposition process. Davies (1966) among others offered an improved theoretical model for particle deposition rate. Liu and Agarwal (1974) analyzed the deposition of aerosol particles in turbulent pipe flows. Simplified simulation procedures for deposition of particles in turbulent flows were described by Abuzeid (1991), and Li and Ahmadi (1993).

A CFD simulation was carried out by Hossain et al. (2003), where the author represented the circumferential particle deposition in a straight pipe for turbulent flow. Mols and Oliemans (1998) developed a mathematical model for particle deposition and dispersion, in which the researchers investigated the circumferential deposition in a straight pipe for gas-liquid flows. But that model was not applicable for circumferential deposition around upstream and downstream sides of bends. In the following sections the circumferential deposition of particle has been represented for both straight and bend sections of the pipe loop.

Particle deposition in bends of circular cross section is important to the sampling and transport of particles in high-purity fluid streams (Tsai and Pui 1990; David et al. 1987). In the experimental study of
Pui (1987), it was revealed that discrepancies still existed between the experimental data and the available theories. These discrepancies are believed to be mainly caused by various flow field assumptions made by different investigators. Even though the problem has been studied both theoretically and experimentally by a number of investigators, it is still unclear at present whether the most applicable theory for the different flow regimes can provide exact solutions of the problems. The complexity of the flow field (three dimensional with strong secondary motion) makes it very difficult to calculate the particle trajectories and deposition in the bends. Unfortunately, detail experiments have not been performed so far for liquid-solid flow to validate this study. Hossain and Naser (2004) have also worked on particles deposition near bend. This study was partially validated with the experimental results of Granger et al. (2003).

2.3 TURBIDITY SPIKES MOVEMENT

The most important method of addressing turbid particles in water has been to counteract the accumulation of sediment in drinking water distribution networks through cleaning activities, treatment processes, biofilm growth within the supply pipes, corrosion products, and/or other unidentified factors (Jayaratne et al. 2004). These cleaning activities include routine planned flushing from service reservoirs to the farthest point of the system, plus isolated flushing following aesthetic related incidents. The cost of these activities is in the order of hundreds of thousands of dollars.

The other reason of turbid particles in water distribution networks is to attribute to the accumulation and subsequent re-suspension of colloidal material (Prince et al. 2003). The colloidal material that is accumulated has been resulted in the turbid water entering from the unfiltered source water (0.7–2.3 NTU), as the system has few if any unlined mains. Introductory indication suggests that this
material is predominately clay and silt (Prince et al. 2000) with no health issues due to the protected catchments from where the water originates. The water is dosed with chlorine for disinfection, lime for pH correction, and fluoride for public health considerations.

Main sources of particles causing turbidity in water distribution networks are from a number of locations as follows:

- Sources of water (Yarra Valley Water 1999; Gauthier et al. 1999) and treatment (Gauthier et al. 2001),
- Generated within the water distribution system itself from pipe and fitting corrosion, erosion (Stephenson 1989),
- Biological growth (Stephenson 1989; Clark et al. 1993; Gauthier et al. 1996; Brunone et al. 2000; Gauthier et al. 2001),
- External contamination such as during pipe repairs (Gauthier et al. 1996), and
- Chemical reactions from the formation of iron or manganese oxides (Gauthier et al. 1996; Stephenson 1989).

Prince (2004) indicated that the majority of particles in Melbourne’s and SEWL’s water distribution networks arose from source water. These distribution networks receive unfiltered water of turbidity typically less than 2 NTU, which is only likely to cause an aesthetic issue if concentration of particles occurs (most likely due to accumulation of particles in the distribution networks).

Any particles that are deposited at the pipe wall are resuspended into the bulk flow when a critical minimum shear stress is exceeded (Prince et al. 2003; Boxall et al. 2001). The frequency of these events is related to the frequency at which this minimum shear stress is exceeded, and the severity is due to the amount of material that has accumulated i.e. the time between critical shear stress exceedences. Thus the worst discoloured water events will
occur when conditions are favourable for accumulation of particles and there is a significant time interval between critical shear stress exceedences. In such situation operational events such as flushing procedures, bursts, and fire fighting may result in a sudden increase in demand greater than normal daily flow, and could cause turbidity spikes event. Other papers that mention operational events causing turbidity spikes events include Hoven & Vreeburg (1992) – hydrant operations, Gray (1994) – repair and maintenance work.

The most important process in various fields of engineering and in nature is the deposition of particles from flowing suspensions. Analysing deposition of small particles suspended in fluid streams has attracted considerable attention in the past three decades (Anderson and Russell 1970b; Anderson and Russell 1970a; David et al. 1987; Swailes and Reeks 1994; Thomson 2003; Laurinat et al. 1985; Abuzeid et al. 1991; Grainger et al. 2003; Hossain et al. 2003; Hossain et al. peer review).

A detailed steady state numerical simulation of particle deposition in distribution networks has been described in Chapter 4. Different flow velocities were considered to explain the hydraulic behavior of particles in distribution networks. For many years the computational models for simulating the hydraulic behavior of water distribution systems have been available (Hossain et al. 2003; Hossain et al. peer review). These CFD models have been extended to analyze water quality as well (Hossain et al. 2003; Hossain et al. peer review). Deposition of small particles in turbulent flows had attracted the interest of many researchers. Davies (1966) among others offered an improved theoretical model for particle deposition rate. Liu and Agarwal (1974) analysed the deposition of aerosol particles in turbulent pipe flows. Simplified simulation procedures for deposition of particles in turbulent flows were described by Abuzeid (1991), and Li and Ahmadi (1993). These all are steady
state models, which are almost unsuccessful to provide enough evidence for the turbidity spikes movement. In general, a transient flow can describe the turbidity spikes movement properly.

Turbidity spikes, which are created from particles deposition and re-suspension through the pipe flow, are the most important problem in water industry, especially for drinking water where there is public health concern. A numerical simulation has been carried out to understand the hydrodynamic behavior of the turbidity spikes in the distribution networks. Depending upon the consumers demand the flow is not constant for all the time, which is the main cause of particles re-suspension. Therefore, in this study several unsteady flow and particle load profiles have been considered (shown in Chapter 5). The particle load profile represents the volume fraction of particle as function of time. The driving force behind this study is the timely challenge to comply with increasingly stringent governmental regulations and customer-oriented expectations. Modern management of water-distribution systems like South East Water Ltd, Melbourne Water, Sydney Water, need simulation models that are able to accurately predict the hydrodynamics of turbidity spikes movement and their deposition criterion, which is the main cause of turbidity in the water distribution networks.

2.4 PARTICLE DISPERSION AND DEPOSITION MODEL

In Australian drinking water distribution networks the largest proportion of water quality customers’ complaints are related to discoloured or ‘dirty water’. One of the main causes of this discoloured water is the particles that are introduced into the systems from various sources. However, little is known about the origins and movement of such particulates. The importance of particle dispersion and deposition has been recognized in the drinking water distribution networks. This study will help to locate the most risky place where these particles settle and then re-
entrainment into the system at high turbulence. The quality of drinking water mostly depends on particulate material from the reticulation system appearing at the customer tap. In Australia, little is understood of the origins, transport and fate of such material, yet it may cause up to 60 per cent of water quality related customer complaints to a water authority (Prince et al. 2000). Therefore, particle dispersion and deposition is to be clearly understood to identify the cause of discolouration of drinking water at the distribution networks. However, the ability to determine the origins of the particulate material varies between different water supply systems, with possible sources being from catchment, treatment processes, biofilm growth within the water supply pipes, corrosion products or other unidentified factors (Jayaratne et al. 2004).

Except for the very large particles (>100 μm), for which the motion is totally dominated by gravity and the particle’s initial entrainment velocity (Anderson and Russell 1970b; Anderson and Russell 1970a; James et al. 1987), there is at present no theoretical analysis of this deposition flux in a two-dimensional geometry. The deposition flux of particles at a certain circumferential angle in the pipe has to be known (Fukano and Ousaka 1989; Laurinat et al. 1985).

Anderson and Russell (1970a) developed a semi-empirical expression to correlate deposition and entrainment fluxes, but only for the top half in the pipe. This model was used to derive this expression assumed that droplet deposition is caused by deterministic drop trajectories intersecting the liquid film. The work of (James et al. 1987) is an extension of the work of (Anderson and Russell 1970b). In both approaches no effect of turbulence was taken into account because only very large particles were considered. Laurinat et al. (1985) proposed an empirical fit to a representative deposition flux profile measured by Anderson and
Russell (1970a). Taylor (1921) and then Friendlander and Johnstone (1957) started at earlier days of Turbulent Diffusion Model. Taylor (1921) introduced the concept of Turbulent Diffusion in a study of the spread of scalar properties like smoke, heat and soluble matter. Friendlander and Johnstone (1957) used this concept for modelling a two-phase flow with particles. They also introduced the “diffusion/free-flight” concept (explained later) for particles deposition on the wall. In order to improve agreement with experimental data, different modifications of this concept were proposed in the course of time: varying free-flight distance from the wall; modifying free-flight velocity; particle diffusivity unequal to eddy diffusivity; changing concentration boundary condition at the free-flight distance (Kallio and Reeks 1989).

The work of Binder and Hanratty (1992) is considered the most recent contribution in the field of particle deposition. They considered the dispersion and deposition of particles in a two-dimensional horizontal rectangular channel by a convection/diffusion model. The diffusion part of this model represents the influence of turbulence and the convection part represents the influence of gravity on the particles. Particles are emitted from an instantaneous point source at the bottom of the channel with some initial entrainment velocity and can deposit at either of the perfectly absorbing boundaries. The particle diffusivity and the particle fall velocity are taken to be functions of the time that a particle has been in the flow field. The resulting convection/diffusion equation and the equation for the time-dependent deterministic velocity of the particles are solved numerically.

The main differences between the method used in this study and the approach of Binder and Hanratty (1992) are two-fold. First, we assumed the particle diffusion coefficient to be constant instead of time-dependent and the particles to be settled due to gravitational
settling velocity. This assumption has the great advantage that the one-dimensional problem can then be solved analytically, so that we find a general expression for the deposition flux independent of the exact quantitative modelling of the particle diffusion coefficient and the gravitational settling velocity. Furthermore, it has the advantage that an analytical two-dimensional deposition flux in a pipe can be calculated containing the relevant physical parameters of the problem. Of course, it has the disadvantage of not taking into account the fact that the particle deterministic velocity is generally time-dependent and that the particle diffusion coefficient is initially also time-dependent. Second, we explicitly included the inertial and crossing trajectories effects in the particle diffusion coefficient. Binder and Hanratty (1992) assumed the particle diffusion coefficient to be equal to the fluid diffusivity. They did not consider at all the crossing trajectories effect.

Mols and Oliemans (1998) developed a similar turbulent model as Binder and Hanratty (1992) in a horizontal annular dispersed gas-liquid flow. However, this model is neither qualitatively nor physically applicable for drinking water distribution networks. Therefore, following the work of Mols and Oliemans (1998) a turbulent diffusion model has been re-developed for liquid-solid flow, which represents the water distribution networks accurately. No such experimental data are available yet to date to validate with this Turbulent Diffusion Model results. The other option, with which this model could be validated and/or compared, is the comprehensively used numerical simulation.

2.5 NUMERICAL SIMULATION

Computational Fluid Dynamics (CFD) is the analysis of systems involving fluid flow, heat transfer and associated phenomena, by means of computer-base simulation. The advent of high-speed and large memory computers has enabled CFD to solve many flow
problems (including those that are compressible or incompressible, laminar or turbulent, chemically reacting or non-reaching, and multiphase) in a reasonable time. Today’s CFD (computational fluid dynamics) codes cover the numerical solution of the Navier-Stokes equations in three dimensions and offer several possibilities for their extension. Usually, scalar equations like the concentration equations may easily be added to the set, and closure of the Reynolds-averaged equations are possible with different turbulence models. These equations form a system of coupled non-linear partial differential equations (PDEs). CFD replaces these differential equations with a set of algebraic equations – a process called discretization. The methods of discretization in CFD are the Finite Difference Method (FDM), Finite Volume Method (FVM), Finite Element Method (FEM), and the Boundary Element Method (BEM).

CFD is now routinely used to get results in different industrial applications. There are many advantages to use CFD including low cost, quick solutions, scale up, comprehensive solutions, and safe options. One of the main advantages of CFD is its ability to test a large number of variables at a very lower cost compare to do so with experimental models. CFD also offers software interfaces, which allow the incorporation of additional terms describing physical effects, not present in usual application problems. Furthermore, several algorithms exist for spatial and temporal discretization. Depending on the numerical scheme behind the solver, the flexibility of the grid generation may be very high. In the present case, the work was performed with the commercial flow solver FLUENT 6.1 (2001). The standard equations of continuity, momentum, and energy conservation are solved on a non-staggered grid by a finite volume approach. The Spalart-Allmaras model provides closure for turbulence. One advantage of this solver is the great number of software interfaces, which allow
incorporating almost any physical behavior. Such an interface was used to implement the algorithm to take into account the settling of the particles. The key obstacle in the implementation of the sink velocity term lies in the definition of the concentration gradient. Since the latter is a spatial difference operator, one of the known schemes must apply. A straightforward solution would be central differencing, which proved to be unstable as expected in convection dominated equation. Therefore an upwind scheme was applied.

To create different geometries, in this study a widely known commercial software GAMBIT (version 2.1), which is supported by Fluent, was used.

2.6 CONCLUSION

Depending on the previous literature, the entire research was mainly worked out in three major steps. Firstly, a comprehensive 3D numerical investigation was carried out to study the hydrodynamics of particles in a horizontal pipe loop consists of four 90° bends in a steady state turbulence flow field. The particle deposition was also investigated at circumferential pipe wall and upstream and downstream sides of the bends. This deposition was also investigated along pipe length at different heights. Secondly, transient flows involving different time dependent arbitrary inlet velocity and particle-load profiles were numerically solved to identify the problems of the drinking water distribution networks. Thousands of such profiles are seen in the distribution system; only a few of them were considered to justify this study with real conditions. Finally, an extended analytical turbulent diffusion model was re-developed for liquid-solid flow emanated from a turbulent model of gas-liquid system. The solution data of the analytical model was then compared with the CFD results.
Chapter 3
Governing Equations and Numerical Method

3.1 INTRODUCTION
The numerical prediction of fluid flow and other related processes can begin when the laws governing these processes have been expressed in mathematical form, generally in terms of differential equation. For a detailed and complete derivation of these equations, the reader should turn to numerous standard textbook (Patankar 1989; Stephen 2000; Markatos 1986). The purpose of this chapter is to develop familiarity with the form and the meaning of these equations. A comprehensive 3D numerical model that is capable of solving a wide range of incompressible and compressible, laminar and turbulent fluid flow problems as well as steady-state or transient analyses has been presented in this chapter. A broad range of mathematical models for transport phenomena (like heat transfer and chemical reactions) is combined with the ability to model any complex geometry. For example, the applications include laminar non-Newtonian flows in process equipment; conjugate heat transfer in turbo-machinery and automotive engine components; pulverized coal combustion in utility boilers; external aerodynamics;
flow through compressors, pumps, and fans; and multiphase flows in bubble columns and fluidized beds.

There are various useful features provided in CFD (Computational Fluid Dynamics) to allow modelling of fluid flow and related transport phenomena in industrial equipment and processes. These include porous media, lumped parameter models (fans and heat exchangers), streamwise-periodic flow and heat transfer, swirl, and moving reference frame models. A time-accurate sliding mesh method, useful for modelling for multiple stages in turbo-machinery applications, is also provided, along with the mixing plane model for computing time-averaged flow fields.

The set of free surface and multiphase flow models is another very useful group of models in CFD. These can be used for analysis of gas-liquid, gas-solid, liquid-solid, and gas-liquid-solid flows. For these types of problems, CFD provides the volume-of-fluid (VOF), mixture, and Eulerian models, as well as the discrete phase model (DPM). The DPM performs Lagrangian trajectory calculations for dispersed phases (particles, droplets, or bubbles), including coupling with the continuous phase. There are many examples of multiphase flows, which include channel flows, sprays, sedimentation, separation, and cavitation. Robust and accurate turbulence models are a vital component that suite of models. The turbulence models provided have a broad range of applicability, and they include the effects of other physical phenomena, such as buoyancy and compressibility. Particular care has been taken to addressing issues of near-wall accuracy via the use of extended wall functions and zonal models.

The basic mathematical model consisting of continuity and momentum equations followed by multiphase mixture model that has been studied in the research will be presented in this chapter.
The multiphase mixture model for numerical prediction will include comprehensive descriptions of hydrodynamic phenomena and flow regimes in a pipeline system as well as physical phenomena such as the rates of particle settling, deposition and re-suspension for straight, bend, and/or joints pipes. The turbulent transport equations are also solved, which will be presented after the multiphase mixture. Detailed experimental investigation in a real-size water distribution network to track the transition of the turbidity spike is expensive and time consuming and in some cases impossible. Therefore, the CFD investigation for this research using the following mathematical models provided by a very well known commercial software FLUENT (2001) is the most renown, effective, and appropriate technique.

3.2 CONTINUITY AND MOMENTUM EQUATIONS

3.2.1 Definition of a Differential Equation

The individual differential equations that are encountered in this mathematical model express a certain conservation principle. Each equation employs a certain physical quantity as its dependent variable and implies that there must be a balance among the various factors that influence the variable. The dependent variables of the differential equations are usually specific properties, i.e., quantities expressed on a unit-mass basis. Examples are mass fraction, velocity (i.e., momentum per unit mass), and specific enthalpy. For all flows, Navier Stokes equation solves conservation equations for mass and momentum. An additional equation for energy conservation is solved for flows involving heat transfer or compressibility and for flows involving species mixing or reactions, a species conservation equation is solved or, if the non-premixed combustion model is used, conservation equations for the mixture fraction and its variance are solved. Additional transport equations are also solved when the flow is turbulent. In this section, the
conservation equations for laminar flow (in an inertial (non-accelerating) reference frame) are presented. The conservation equations relevant to turbulence modelling will be discussed in the chapters where those models are described.

The terms in a differential equation denote influences on a unit-volume basis. An example will make this clear. Suppose $J$ denotes a flux influencing a typical dependent variable $\phi$. Let us consider the control volume of dimensions $dx$, $dy$, and $dz$ shown in Figure 3-1.

Figure 3-1: Flux balance over a control volume.

The $J_x$ (which is the $x$-direction component of $J$) is shown entering on face of area $dx \, dz$, while the flux leaving the opposite face is shown as $J_x + (\partial J_x/\partial x)dx$. Thus, the net efflux is $(\partial J_x/\partial x)dx \, dy \, dz$ over the area of the face. Considering the contributions of the $y$ and $z$ directions as well and noting that $dx \, dy \, dz$ is the volume of the region considered, we have

\[
\text{Net efflux per unit volume} = \frac{\partial J_x}{\partial x} + \frac{\partial J_y}{\partial y} + \frac{\partial J_z}{\partial z} = \nabla \cdot J \quad [2-1]
\]

This interpretation of $\nabla \cdot J$ will be particularly useful because of the numerical method (explained later) will be constructed by performing a balance over a control volume.

Another example of a term expressed on a unit-volume basis is the rate-of-change term $\partial(\rho \phi)/\partial t$. If $\phi$ is a specific property and $\rho$ in the density, then $\rho \phi$ denotes the amount of the corresponding extensive property contained in a unit volume.
A differential equation is a compilation of such terms, each representing an influence on a unit-volume basis, and all the terms together implying a balance or conservation.

**3.2.2 The General Differential Equation**

The governing transport differential equations have indicated that all the dependent variables of interest here seem to obey a generalized conservation principle. If the dependent variable is denoted by $\phi$, the general differential equation is

$$\frac{\partial}{\partial t}(\rho \phi) + \nabla \cdot (\rho \phi \hat{V}) = \nabla \cdot (\Gamma \text{grad} \phi) + S$$  \[2-2\]

where $\Gamma$ is the diffusion coefficient, and $S$ is the source term. The quantities $\Gamma$ and $S$ are specific to a particular meaning of $\phi$.

The four terms in the general differential equation are the unsteady term, the convection term, the diffusion term, and the source term. The dependent variable $\phi$ can be a variety of different quantities, such as the mass fraction of a chemical species, the enthalpy or the temperature, a velocity component, the turbulence kinetic energy, or a turbulence length scale. Accordingly, for each of these variables, an appropriate meaning will have to be given to the diffusion coefficient $\Gamma$ and the source term $S$.

Not all diffusion fluxes are governed by the gradient of the relevant variable. The use of $\nabla \cdot (\Gamma \text{grad} \phi)$ as the diffusion term does not, however, limit the general $\phi$ equation to gradient-driven diffusion processes. Whatever cannot be fitted into the nominal diffusion term can be expressed as a part of the source term; in fact, the diffusion coefficient $\Gamma$ can even be set equal to zero if desired. A gradient-diffusion term has been explicitly included in the general $\phi$ equation because most dependent variables do require a prominent diffusion term of this nature.
3.2.3 The Mass Conservation Equation

The dependent variable $\phi$ and the diffusion coefficient $\Gamma$ are considered to be 1 and zero, respectively and therefore, the equation for conservation of mass, or the continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m$$  \[2-3\]

Equation [2-3] is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows. The source $S_m$ is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources.

3.2.4 Momentum Conservation Equations

The differential equation governing the conservation of momentum in a given direction for a Newtonian fluid can be written along similar lines; however, the complication is greater because the Stokes viscosity law is more complicated than Fick’s law or Fourier’s law (Patankar 1980). With $\vec{V}$ denoting the $x$-direction velocity, it can be written the corresponding momentum equation by considering the dependent variable $\phi$ as a velocity component and the diffusion coefficient $\Gamma$ as Bachelor (1967):

$$\frac{\partial \rho}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\bar{\tau}) + \rho \vec{g} + \vec{F}$$  \[2-4\]

where $p$ is the static pressure, $\bar{\tau}$ is the stress tensor (described below), and $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively.

The stress tensor $\bar{\tau}$ is given by

$$\bar{\tau} = \left[(\rho \vec{v} \rho \vec{v} - \frac{2}{3} \nabla \cdot \vec{v} I) \right]$$  \[2-5\]
where $\rho$ is the molecular viscosity, $I$ is the unit tensor, and the second term on the right hand side is the effect of volume dilation.

The continuity and momentum equations that have been presented in the above sections are for the single phase fluid flow. However, the physical problem that describes the drinking water distribution network is concerned with multi-phase, multi-particle flow consisting of both liquid and solid phases. In the following multiphase model the liquid and the solids will be considered as primary and secondary phases respectively.

### 3.3 MODELLING MULTIPHASE FLOWS

A large number of flows encountered in nature and technology are a mixture of phases. Physical phases of matter are gas, liquid, and solid, but the concept of phase in a multiphase flow system is applied in a broader sense. In multiphase flow, a phase can be defined as an identifiable class of material that has a particular inertial response to and interaction with the flow and the potential field in which it is immersed. For example, different-sized solid particles of the same material can be treated as different phases because each collection of particles with the same size will have a similar dynamical response to the flow field.

In this section an overview of multiphase modelling has been presented, and later sections will provide details about the multiphase model that has been used in this research.

#### 3.3.1 Multiphase Flow Regimes

Multiphase flow can be classified by the following regimes, grouped into four categories, which have been illustrated in Figure 3-2.
Gas-liquid or liquid-liquid flows

- Bubbly flow: discrete gaseous or fluid bubbles in a continuous fluid;
- Droplet flow: discrete fluid droplets in a continuous gas;
- Slug flow: large bubbles in a continuous fluid; and
- Stratified/free-surface flow: immiscible fluids separated by a clearly-defined interface.

Gas-solid flows

- Particle-laden flow: discrete solid particles in a continuous gas;
- Pneumatic transport: flow pattern depends on factors such as solid loading, Reynolds numbers, and particle properties. Typical patterns are dune flow, slug flow, packed beds, and homogeneous flow; and
- Fluidized beds: consist of a bed of particles where gas is introduced through a distributor. The gas rising through the
bed suspends the particles. Depending on the gas flow rate, bubbles appear and rise through the bed, intensifying the mixing within the bed.

Liquid-solid flows

- Slurry flow: transport of particles in liquids. The fundamental behavior of liquid-solid flows varies with the properties of the solid particles relative to those of the liquid. In slurry flows, the Stokes number is normally less than 1. When the Stokes number is larger than 1, the characteristic of the flow is liquid-solid fluidization;
- Hydro-transport: densely-distributed solid particles in a continuous liquid; and
- Sedimentation: a tall column initially containing a uniform dispersed mixture of particles. At the bottom, the particles will slow down and form a sludge layer. At the top, a clear interface will appear, and in the middle a constant settling zone will exist.

3.3.2 Examples of Multiphase Systems

Specific examples of each regime are listed below:

- Bubbly flow: absorbers, aeration, air lift pumps, cavitation, evaporators, flotation, scrubbers
- Droplet flow: absorbers, atomizers, combustors, cryogenic pumping, dryers, evaporation, gas cooling, scrubbers
- Slug flow: large bubble motion in pipes or tanks
- Stratified/free-surface flow: sloshing in offshore separator devices, boiling and condensation in nuclear reactors
- Particle-laden flow: cyclone separators, air classifiers, dust collectors, and dust-laden environmental flows
Pneumatic transport: transport of cement, grains, and metal powders
Fluidized bed: fluidized bed reactors, circulating fluidized beds
Slurry flow: slurry transport, mineral processing
Hydro-transport: mineral processing, biomedical and physiochemical fluid systems
Sedimentation: mineral processing slug flow bubbly, droplet, or particle-laden flow stratified/free-surface flow pneumatic transport, hydro-transport, or slurry flow sedimentation fluidized bed

The physical problem of the turbidity spikes in drinking water distribution networks demonstrates the liquid-solid flows that include the sedimentation and resuspension only.

3.3.3 Approaches to Multiphase Modelling
Advanced computational fluid mechanics (CFD) have provided the basis for further insight into the dynamics of multiphase flows. Currently there are two approaches for the numerical calculation of multiphase flows: the Euler-Lagrange approach and the Euler-Euler approach.

The Euler-Lagrange Approach
The Lagrangian discrete phase model follows the Euler-Lagrange approach. The fluid phase is treated as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase. A fundamental assumption made in this model is that the dispersed second phase occupies a low volume fraction, even though high mass loading ($m_{\text{particles}} \geq m_{\text{fluid}}$) is acceptable. The particle or droplet
trajectories are computed individually at specified intervals during the fluid phase calculation.

The model is appropriate for the modelling of spray dryers, coal and liquid fuel combustion, and some particle-laden flows, but inappropriate for the modelling of liquid-liquid mixtures, fluidized beds, or any application where the volume fraction of the secondary phases is not negligible.

**The Euler-Euler Approach**

The different phases are treated mathematically as interpenetrating continua in the Euler-Euler approach. The concept of phasic volume fraction is introduced as the volume of a phase cannot be occupied by the other phases. These volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations, which have similar structure for all phases. These equations are closed by providing constitutive relations that are obtained from empirical information, or, in the case of granular flows, by application of kinetic theory.

When faced with the task of modelling turbulent particles deposition, or any multiphase flow, two general approaches are possible. One is Lagrangian approach, usually known as a “trajectory model” (Kallio and Reeks 1989), where the instantaneous motions of individual particles are tracked by solving their equations of motion. The trajectories of many particles (typically thousands) are realized in order to form the average behavior of the particle-fluid system. The other is Eulerian, often called a “two-fluid” model, where the particles are treated as a continuous phase, in much the same way that a tracer fluid would be regarded in a binary mixture. The motion of the particulate phase is mathematically described by mass, momentum and energy
conservation, similar to a fluid. In this study we followed the Eulerian approach, which is more suitable for re-suspension and/or re-deposition of particles than Lagrangian approach (Patankar and Joseph 2001). The Lagrangian approach would be more appropriate for this investigation because of the low volume fraction of solids. However, in Lagrangian approach the particles are tracked through the fluid domain and their effects on the fluid flow are introduced through forces like drag, but the physical existence of the particles creating blockages or voidages is ignored (Patankar and Joseph 2001; Patankar 1980). This physical existence of the particles is essential when there is a possibility of re-suspension or re-entrainment into the systems, which is the case in this research work. In order to study the behavior of particles in a turbulent flow field numerically, one needs a proper representation of turbulence itself.

The steady-particle Lagrangian discrete phase model is suited for flows in which particle streams are injected into a continuous phase flow with a well-defined entrance and exit condition (Patankar and Joseph 2001; Patankar 1980). The Lagrangian model does not effectively model flows in which particles are suspended indefinitely in the continuum, as occurs in solid suspensions within closed systems such as stirred tanks, mixing vessels, fluidized beds, and close channel flow. The unsteady-particle discrete phase model, however, is capable of modelling continuous suspensions of particles.

Therefore, Eulerian approach has been considered for this research to study the hydrodynamic behavior of particle in turbulent flow with very low volume fraction of solids introduced as secondary phases. There are three different Euler-Euler multiphase models available as follows:

1. The volume of fluid (VOF) model,
2 The mixture model, and

3 The Eulerian model.

The VOF Model

The VOF model is a surface-tracking technique applied to a fixed Eulerian fluid flow. This model is designed for two or more immiscible fluids where the accurate prediction of the interface between the fluids is of interest. In the VOF model, a single set of momentum equations is shared by the fluids, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. Applications of the VOF model include stratified flows, free-surface flows, filling, sloshing, the motion of large bubbles in a liquid, the motion of liquid after a dam break, the prediction of jet breakup (surface tension), and the steady or transient tracking of any liquid-gas interface.

The Mixture Model

The mixture model (described later) is designed for two or more phases (fluid or particulate). As in the Eulerian model, the phases are treated as interpenetrating continua. The mixture model solves for the mixture momentum equation and prescribes relative velocities to describe the dispersed phases. Applications of the mixture model include particle-laden flows with low loading, bubbly flows, sedimentation, and cyclone separators. The mixture model can also be used without relative velocities for the dispersed phases to model homogeneous multiphase flow.

The Eulerian Model

The Eulerian model is the most complex of the multiphase models. It solves a set of $n$ momentum and continuity equations for each phase. Coupling is achieved through the pressure and interphase exchange coefficients. This coupling is handled by depending upon
the type of phases involved; granular (fluid-solid) flows are handled differently than non-granular (fluid-fluid) flows. For granular flows, the properties are obtained from application of kinetic theory. Momentum exchange between the phases is also dependent upon the type of mixture being modelled. The commercial software Fluent’s user-defined functions (UDF) allow to customize the calculation of the momentum exchange. Applications of the Eulerian multiphase model include bubble columns, risers, particle sedimentation and suspension, and fluidized beds.

### 3.3.4 Choosing a Multiphase Model

The first step in solving any multiphase problem is to determine which of the regimes described in Figure 3-2 (page 31) best represents the flow domain.

Table 3-1: General guidelines for flow types.

<table>
<thead>
<tr>
<th>Flow Types</th>
<th>Model used for</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bubbly flows,</td>
<td>The discrete phase model</td>
</tr>
<tr>
<td>Droplet flows, and</td>
<td></td>
</tr>
<tr>
<td>Particle-laden flows (in which the</td>
<td></td>
</tr>
<tr>
<td>dispersed phase volume fractions</td>
<td></td>
</tr>
<tr>
<td>are less than or equal to 10%)</td>
<td></td>
</tr>
<tr>
<td>Bubbly flows,</td>
<td>The mixture model (described in later) or</td>
</tr>
<tr>
<td>Droplet flows, and</td>
<td>The Eulerian model</td>
</tr>
<tr>
<td>Particle-laden flows (in which the</td>
<td></td>
</tr>
<tr>
<td>phases mix and/or dispersed-phase</td>
<td></td>
</tr>
<tr>
<td>volume fractions exceed 10%)</td>
<td></td>
</tr>
<tr>
<td>Slug flows,</td>
<td>The VOF model</td>
</tr>
<tr>
<td>Stratified/free-surface flows,</td>
<td></td>
</tr>
<tr>
<td>Pneumatic transport,</td>
<td></td>
</tr>
<tr>
<td>Fluidized beds,</td>
<td>The VOF model</td>
</tr>
<tr>
<td>Slurry flows and</td>
<td></td>
</tr>
<tr>
<td>Hydro-transport,</td>
<td>The mixture model for homogeneous flow (described in</td>
</tr>
<tr>
<td>Sedimentation,</td>
<td></td>
</tr>
<tr>
<td>For general,</td>
<td></td>
</tr>
<tr>
<td></td>
<td>The Eulerian model for granular flow</td>
</tr>
<tr>
<td></td>
<td>The Eulerian model for granular flow</td>
</tr>
<tr>
<td></td>
<td>The mixture or</td>
</tr>
<tr>
<td></td>
<td>The Eulerian model</td>
</tr>
<tr>
<td></td>
<td>Complex multiphase flows that involve</td>
</tr>
<tr>
<td></td>
<td>multiple flow regimes</td>
</tr>
</tbody>
</table>

Table 3-1 provides the general guidelines for different flow types (Fluent 6.1 Manual 2001). In general, once the flow regime is selected that best represents multiphase system of the physical problem; the appropriate model can be selected based on the
following guidelines (Table 3-1). Additional details and guidelines for selecting the appropriate model for flows involving bubbles, droplets, or particles sedimentation and re-suspension can be found in the later section.

**Detailed Guidelines**

For stratified and slug flows, the choice of the VOF model, is straightforward. Choosing a model for the other types of flows is less straightforward. As a general guide, there are some parameters that help to identify the appropriate multiphase model for these other flows: the particulate loading, $\beta$, and the Stokes number, $St$. (Note that the word ‘particle’ is used in this discussion to refer to a particle, droplet, or bubble.)

**The Effect of Particulate Loading**

Particulate loading has a major impact on phase interactions. The particulate loading is defined as the mass density ratio of the dispersed phase (d) to that of the carrier phase (c) (Fluent 2001):

$$\beta = \frac{\alpha_d \rho_d}{\alpha_c \rho_c} \quad [3-1]$$

The material density ratio

$$\gamma = \frac{\rho_d}{\rho_c} \quad [3-2]$$

is greater than 1000 for gas-solid flows, about more than 1 for liquid-solid flows, and less than 0.001 for gas-liquid flows.

Using these parameters it is possible to estimate the average distance between the individual particles of the particulate phase. An estimate of this distance has been given by Crowe et al. (1998):

$$\frac{L}{d_d} = \left(\frac{\pi}{6} \frac{1 + \kappa}{\kappa}\right)^{\frac{1}{3}} \quad [3-3]$$
where $\kappa = \beta / \gamma$. Information about these parameters is important for determining how the dispersed phase should be treated. For example, for a gas-particle flow with a particulate loading of 1, the interparticle space $L/d_d$ is about 8; the particle can therefore be treated as isolated (i.e., very low particulate loading).

Depending on the particulate loading, the degree of interaction between the phases can be divided into three categories:

For very low loading, the coupling between the phases is one-way; i.e., the fluid carrier influences the particles via drag and turbulence, but the particles are assumed to have no influence on the fluid carrier. The discrete phase, mixture, and Eulerian models can all handle this type of problem correctly. Since the Eulerian model is the most expensive due to higher computational time as compared to other models, the discrete phase or mixture model is recommended (FLUENT INC. 2001).

For intermediate loading, the coupling is two-way; i.e., the fluid carrier influences the particulate phase via drag and turbulence, but the particles in turn influence the carrier fluid via reduction in mean momentum and turbulence. The discrete phase, mixture, and Eulerian models are all applicable in this case, but other factors need to be taken into account in order to decide which model is more appropriate. See below for information about using the Stokes number as a guide.

For high loading, there is two-way coupling plus particle pressure and viscous stresses due to particles (four-way coupling). Only the Eulerian model will handle this type of problem correctly (FLUENT INC. 2001).

The Significance of the Stokes Number
For systems with intermediate particulate loading, estimating the value of the Stokes number can help in selecting the most
appropriate model. The Stokes number can be defined as the relationship between the particle response time and the system response time:

\[ St = \frac{\tau_d}{t_s} \]  

where \( \tau_d = \frac{\rho_d d^2}{18 \mu_c} \) (\( \mu_c \) is the viscosity of the phase c) and \( t_s \) is based on the characteristic length (\( L_s \)) and the characteristic velocity (\( V_s \)) of the system under investigation: \( t_s = \frac{L_s}{V_s} \).

For \( St \ll 1.0 \), the particle will follow the flow closely and any of the three models (discrete phase, mixture, or Eulerian) is applicable; therefore, like other researchers the least expensive should be chosen (the mixture model, in most cases), or the most appropriate considering other factors. For \( St > 1.0 \), the particles will move independently of the flow and either the discrete phase model or the Eulerian model is applicable. For \( St \approx 1.0 \), again any of the three models is applicable; the least expensive or the most appropriate considering other factors should be chosen. In the study, the stokes number of the particle that has been investigated is very very less than 1 (\( St \ll 1.0 \)). Therefore, multiphase mixture model is the only choice as it is in the first category.

Considering the particle loading, which is very low in this research the Discrete Phase Model would be the most obvious choice. These modelling capabilities allow simulating a wide range of discrete phase problems including particle separation and classification, spray drying, aerosol dispersion, bubble stirring of liquids, liquid fuel combustion, and coal combustion. The dispersion of particles due to turbulence in the fluid phase can be predicted using the stochastic tracking model or the particle cloud model. The stochastic tracking (random walk) model includes the effect of instantaneous turbulent velocity fluctuations on the particle trajectories through the use of
stochastic methods. The particle cloud model tracks the statistical evolution of a cloud of particles about a mean trajectory. The concentration of particles within the cloud is represented by a Gaussian probability density function (PDF) about the mean trajectory. In both models, the particles have no direct impact on the generation or dissipation of turbulence in the continuous phase.

However, this modelling scheme has some limitations, which drive the researcher to find out other options to look at. The discrete phase formulation contains the assumption that the second phase is sufficiently dilute that particle-particle interactions and the effects of the particle volume fraction on the primary phase are negligible. In practice, these issues imply that the discrete phase must be present at a fairly low volume fraction. Note that the mass loading of the discrete phase must not exceed 10%. However, in this study the mass flow of the secondary phases equals or exceeds the limit due to accumulation of particulate phases into the domain investigated. Therefore, researchers may need to look at Table 3-1 for more information about the general multiphase models instead of the discrete phase model.

Other than discrete phase model, as Table 3-1 shows Eulerian and Mixture Model would be the best choices in the area of particle deposition and re-suspension where there is a possibility of excess volume fraction of particle (solids) due to deposition that is not possible to simulate with the present discrete phase model.

Furthermore, as stated earlier the Eulerian model in this case is expensive due to higher computational time (Fluent 2001; Lewis and Paul 1996; Rao et al. 2002). Therefore, the least expensive (Fluent 2001) or the most appropriate considering other factors the Multiphase Mixture Model is the better option to carry out this physical problem. Section 3.4 provides the overview, inevitable
limitations, and detailed description of the Multiphase Mixture Model.

3.4 THE MULTIPHASE MIXTURE MODEL

3.4.1 Overview of the Mixture Model

The Mixture Model, like the VOF model, uses a single-fluid approach. It differs from the VOF model in two respects:

The Mixture Model allows the phases to be interpenetrating, which is the case here. The volume fractions $\alpha_q$ and $\alpha_p$ for a control volume can therefore be equal to any value between 0 and 1, depending on the space occupied by phase $q$ and phase $p$.

The Mixture Model allows the phases to move at different velocities, using the concept of slip velocities. (Note that the phases can also be assumed to move at the same velocity, and the mixture model is then reduced to a homogeneous multiphase model.) The Mixture Model solves the continuity equation for the mixture, the momentum equation for the mixture, the energy equation for the mixture, and the volume fraction equation for the secondary phases, as well as algebraic expressions for the relative velocities (if the phases are moving at different velocities).

3.4.2 Limitations of the Mixture Model

Although the Mixture Model has got some inevitable limitations, it describes the physical problem of the distribution networks very well. The following limitations apply to the mixture model in CFD (Fluent 2001; Patankar 1980):

- Other than segregated solver the mixture model is not appropriate,
- Only one of the phases can be compressible (in this study there is no compressible phase).
Streamwise periodic flow (either specified mass flow rate or specified pressure drop) cannot be modelled. This has been ignored due to software limitation (Fluent 2001).

Species mixing and reacting flow cannot be modelled when the mixture model is used (surface reaction and biofilm growth can not be modelled).

The LES (Large Eddy Simulation) turbulence model cannot be used with the mixture model.

The second-order implicit time-stepping formulation cannot be used with the mixture model (in this study, first order implicit time-stepping formula has been used).

The mixture model cannot be used for inviscid flows.

The shell conduction model for walls cannot be used with the mixture model.

Despite of the limitations stated above, the Mixture Model is still the best choice to predict the hydrodynamic behavior of particle in turbulent flow due to such low volume fraction of the secondary phases and the smallest stokes numbers of the physical problem for this research. This is because of none of the above excluding the reacting flow describes the physical problem exclusively.

### 3.4.3 Continuity Equation for the Mixture

The term ‘mixture’ can be defined by the combination of all the primary and secondary phases. The continuity equation for the mixture is

\[
\frac{\partial}{\partial t} (\rho_m) + \nabla \cdot (\rho_m \vec{v}_m) = \text{\textit{r}} \star
\]

where \( \vec{v}_m \) is the mass-averaged velocity of the mixture, which is

\[
\vec{v}_m = \sum_{k=1}^{n} \alpha_k \frac{\rho_k \vec{v}_k}{\rho_m}
\]

[4-1]

[4-2]
and $\rho_m$ is the mixture density:

$$\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k$$  \[4-3\]

$\alpha_k$ is the volume fraction of phase $k$.

$\mathbf{m}$ represents mass transfer due to cavitation or user-defined mass sources.

### 3.4.4 Momentum Equation for the Mixture

The momentum equation for the mixture can be obtained by summing the individual momentum equations for all phases. It can be expressed as

$$\frac{\partial}{\partial t} (\rho_m \mathbf{v}_m) + \nabla \cdot (\rho_m \mathbf{v}_m \mathbf{v}_m) = -\nabla p + \nabla \cdot \left[ \mu_m \left( \nabla \mathbf{v}_m + \nabla \mathbf{v}_m^T \right) \right] +$$

$$\rho_m \mathbf{g} + \dot{\mathbf{F}} + \nabla \cdot \left( \sum_{k=1}^{n} \alpha_k \rho_k \mathbf{v}_{dr,k} \mathbf{v}_{dr,k} \right)$$  \[4-4\]

where $n$ is the number of phases, $\dot{\mathbf{F}}$ is a body force, and $\mu_m$ is the viscosity of the mixture:

$$\mu_m = \sum_{k=1}^{n} \alpha_k \mu_k$$  \[4-5\]

$\mathbf{v}_{dr,k}$ is the drift velocity for secondary phase $k$:

$$\mathbf{v}_{dr,k} = \mathbf{v}_k - \mathbf{v}_m$$  \[4-6\]

### 3.4.5 Relative (Slip) Velocity and the Drift Velocity

The relative velocity (also referred to as the slip velocity) is defined as the velocity of a secondary phase ($p$) relative to the velocity of the primary phase ($q$):

$$\mathbf{v}_{qp} = \mathbf{v}_p - \mathbf{v}_q$$  \[4-9\]
The drift velocity and the relative velocity \( \dot{V}_{qp} \) are connected by the following expression:

\[
\dot{V}_{dr,p} = \dot{V}_{qp} - \sum_{k=1}^{n} \frac{\alpha_k \rho_k}{\rho_m} \dot{V}_{qk}
\]  \[4-10\]

The mixture model makes use of an algebraic slip formulation. The basic assumption of the algebraic slip mixture model is that, to prescribe an algebraic relation for the relative velocity, a local equilibrium between the phases should be reached over short spatial length scales. The form of the relative velocity is given by

\[
\dot{V}_{qp} = \tau_{qp} \dot{a}
\]  \[4-11\]

where \( \dot{a} \) is the secondary-phase particle's acceleration and \( \tau_{qp} \) is the particulate relaxation time. Following Manninen et al. (1996) \( \tau_{qp} \) is of the form:

\[
\tau_{qp} = \frac{(\rho_m - \rho_p) d_p^2}{18 \mu_q f_{\text{drag}}}
\]  \[4-12\]

where \( d_p \) is the diameter of the particles of secondary phase \( p \), and the drag function \( f_{\text{drag}} \) is taken from Schiller and Naumann (1935):

\[
f_{\text{drag}} = \begin{cases} 
1 + 0.15 \text{Re}^{0.687} & \text{Re} \leq 1000 \\
0.0183 \text{Re} & \text{Re} > 1000 
\end{cases}
\]  \[4-13\]

and the acceleration \( \dot{a} \) is of the form

\[
\dot{a} = \ddot{g} - (\nabla \cdot \dot{V}) \dot{\rho}_m \rho_m - \frac{\partial \psi}{\partial t}
\]  \[4-14\]

The simplest algebraic slip formulation is the so-called drift flux model, in which the acceleration of the particle is given by gravity and/or a centrifugal force and the particulate relaxation time is modified to take into account the presence of other particles.

Note that, if the slip velocity is not solved, the mixture model is reduced to a homogeneous multiphase model. In addition, the
mixture model can be customized (using user-defined functions) to use a formulation other than the algebraic slip method for the slip velocity.

### 3.4.6 Volume Fraction Equation for the Secondary Phases

From the continuity equation for the secondary phase $p$, the volume fraction equation for secondary phase $p$ can be obtained:

$$
\frac{\partial}{\partial t} \left( \alpha_p \rho_p \right) + \nabla \cdot \left( \alpha_p \rho_p \mathbf{v}_m \right) = -\nabla \cdot \left( \alpha_p \rho_p \mathbf{v}_{dr,p} \right)
$$

[4-15]

### 3.5 TURBULENT MODELLING

In this study, the flow domains that are investigated numerically or analytically are fully turbulent with the lowest possible physical characteristics of the problem. Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities. There are the following choices of turbulence models:

- Spalart-Allmaras model
- $k$-$\varepsilon$ models
  - Standard $k$-$\varepsilon$ model
  - Renormalization-group (RNG) $k$-$\varepsilon$ model
3.5.1 Turbulence Model Selection for this Research

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model for the application, one needs to understand the capabilities and limitations of the various options.

The purpose of this section is to give an overview of issues related to the turbulence models. The computational effort and cost in terms of CPU time and memory of the individual models is discussed. While it is impossible to state categorically which model is best for a specific application, general guidelines are presented to help to select the appropriate turbulence model for the flow that is to be modelled.

Reynolds-Averaged Approach vs. LES

A complete time-dependent solution of the exact Navier-Stokes equations for high-Reynolds-number turbulent flows in complex geometries is unlikely to be attainable for some time to come. Two alternative methods can be employed to transform the Navier-Stokes equations in such a way that the small-scale turbulent
fluctuations do not have to be directly simulated: Reynolds averaging and filtering. Both methods introduce additional terms in the governing equations that need to be modelled in order to achieve closure. (Closure implies that there are a sufficient number of equations for all the unknowns.)

The Reynolds-averaged Navier-Stokes (RANS) equations represent transport equations for the mean flow quantities only, with all the scales of the turbulence being modelled. The approach of permitting a solution for the mean flow variables greatly reduces the computational effort. If the mean flow is steady, the governing equations will not contain time derivatives and a steady-state solution can be obtained economically. A computational advantage is seen even in transient situations, since the time step will be determined by the global unsteadiness in the mean flow rather than by the turbulence. The Reynolds-averaged approach is generally adopted for practical engineering calculations, and uses models such as Spalart-Allmaras, k-ε and its variants, k-ω and its variants, and the RSM.

LES provides an alternative approach in which the large eddies are computed in a time-dependent simulation that uses a set of filtered equations. Filtering is essentially a manipulation of the exact Navier-Stokes equations to remove only the eddies that are smaller than the size of the filter, which is usually taken as the mesh size. Like Reynolds averaging, the filtering process creates additional unknown terms that must be modelled in order to achieve closure. Statistics of the mean flow quantities, which are generally of most engineering interest, are gathered during the time-dependent simulation. The attraction of LES is that, by modelling less of the turbulence (and solving more), the error induced by the turbulence model will be reduced. One might also argue that it ought to be easier to find a universal model for the small scales, which tend to
be more isotropic and less affected by the macroscopic flow features than the large eddies.

However, it should be stressed that the application of LES to industrial fluid simulations is in its infancy. As highlighted in a review publication (Galperin and Orszag 1993), typical applications to date have been for simple geometries. This is mainly because of the large computer resources required to resolve the energy-containing turbulent eddies. Most successful LES has been done using high-order spatial discretization, with great care being taken to resolve all scales larger than the inertial subrange. The degradation of accuracy in the mean flow quantities with poorly resolved LES is not well documented. In addition, the use of wall functions with LES is an approximation that requires further validation. As a general guideline, therefore, it is recommended that the conventional turbulence models employing the Reynolds-averaged approach be used for practical calculations. The LES approach needs to have more computational resources and efforts. Therefore, for this study Reynolds-averaged approach has been chosen, which is presented in the following section.

Reynolds (Ensemble) Averaging

The solution variables in the instantaneous (exact) Navier-Stokes equations of Reynolds averaging turbulent model are decomposed into the mean of ensemble averaged or time-averaged and fluctuating components. They have the same general form as the instantaneous Navier-Stokes equations, with the velocities and other solution variables now representing ensemble-averaged (or time-averaged) values.

For variable-density flows can be interpreted as Favre-averaged Navier-Stokes equations (1975), with the velocities representing mass-averaged values.
**Boussinesq Approach vs. Reynolds Stress Transport Models**

The Reynolds-averaged approach to turbulence modelling requires that the Reynolds stresses be appropriately modelled. The Boussinesq hypothesis is used in the Spalart-Allmaras model, the $k - \varepsilon$ models, and the $k-\omega$ models. The advantage of this approach is the relatively low computational cost associated with the computation of the turbulent viscosity, $\mu_t$. In the case of the Spalart-Allmaras model, only one additional transport equation (representing turbulent viscosity) is solved. In the case of the $k-\varepsilon$ and $k-\omega$ models, two additional transport equations (for the turbulence kinetic energy, $k$, and either the turbulence dissipation rate, $\varepsilon$, or the specific dissipation rate, $\omega$) are solved, and $\mu_t$ is computed as a function of $k$ and $\varepsilon$. The disadvantage of the Boussinesq hypothesis as presented is that it assumes $\mu_t$ is an isotropic scalar quantity, which is not strictly true.

The alternative approach, embodied in the RSM, is to solve transport equations for each of the terms in the Reynolds stress tensor. An additional scale-determining equation (normally for $\varepsilon$) is also required. This means that five additional transport equations are required in 2D flows and seven additional transport equations must be solved in 3D.

In many cases, models based on the Boussinesq hypothesis perform very well, and the additional computational expense of the Reynolds stress model is not justified. However, the RSM is clearly superior for situations in which the anisotropy of turbulence has a dominant effect on the mean flow. Such cases include highly swirling flows and stress-driven secondary flows.

**3.5.2 The Spalart-Allmaras Model**

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modelled transport equation for the kinematic
eddy (turbulent) viscosity. This embodies a relatively new class of one-equation models in which it is not necessary to calculate a length scale related to the local shear layer thickness.

The Spalart-Allmaras model is effectively a low-Reynolds-number model, requiring the viscous-affected region of the boundary layer to be properly resolved. However, the Spalart-Allmaras model has been implemented to use wall functions when the mesh resolution is not sufficiently fine. This might make it the best choice for relatively crude simulations on coarse meshes. Furthermore, the near-wall gradients of the transported variable in the model are much smaller than the gradients of the transported variables in the $k-\varepsilon$ or $k-\omega$ models. This might make the model less sensitive to numerical error when non-layered meshes are used near walls. The Spalart-Allmaras model is relatively new, and robust. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow simulations. Furthermore, one-equation models are often criticized for their inability to rapidly accommodate changes in length scale, such as might be necessary when the flow changes abruptly from a wall-bounded to a free shear flow.

3.5.3 The Standard $k-\varepsilon$ Model

The simplest complete models of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard $k-\varepsilon$ model falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding (1972).

It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism. This model still has the strengths as well as weaknesses
of the standard $k-\varepsilon$ model. Two of these variants are RNG $k-\varepsilon$ model (Yakhot and Orszag 1986) and the realizable $k-\varepsilon$ model (Shih at el. 1995).

The RNG $k-\varepsilon$ model was derived using a rigorous statistical technique (called renormalization group theory). It is similar in form to the standard $k-\varepsilon$ model. The realizable $k-\varepsilon$ model is a relatively recent development and differs from the standard $k-\varepsilon$ model in two important ways: it contains a new formulation for the turbulent viscosity and a new transport equation for the dissipation rate, $\varepsilon$.

The term *realizable* means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Neither the standard $k-\varepsilon$ model nor the RNG $k-\varepsilon$ model is realizable.

### 3.5.4 The Standard $k-\omega$ Model

The standard $k-\omega$ model is based on the Wilcox $k-\omega$ model (1998), which incorporates modifications for low-Reynolds-number effects, compressibility, and shear flow spreading. The Wilcox model predicts free shear flow spreading rates that are in close agreement with measurements for far wakes, mixing layers, and plane, round, and radial jets, and is thus applicable to wall-bounded flows and free shear flows. A variation of the standard $k-\omega$ model called the SST $k-\varepsilon$ model, which was developed by Menter (Menter 1994) to effectively blend the robust and accurate formulation of the $k-\omega$ model in the near-wall region with the free-stream independence of the $k-\varepsilon$ model in the far field.

### 3.5.5 The Reynolds Stress Model (RSM)

The Reynolds stress model (RSM) is the most elaborate turbulence model. Abandoning the isotropic eddy-viscosity hypothesis, the RSM closes the Reynolds-averaged Navier-Stokes equations by solving transport equations for the Reynolds stresses, together with an
equation for the dissipation rate. This means that four additional transport equations are required in 2D flows and seven additional transport equations must be solved in 3D.

Since the RSM accounts for the effects of streamline curvature, swirl, rotation, and rapid changes in strain rate in a more rigorous manner than one-equation and two-equation models, it has greater potential to give accurate predictions for complex flows. However, the fidelity of RSM predictions is still limited by the closure assumptions employed to model various terms in the exact transport equations for the Reynolds stresses. The modelling of the pressure-strain and dissipation-rate terms is particularly challenging, and often considered to be responsible for compromising the accuracy of RSM predictions.

3.5.6 Computational Effort: CPU Time and Solution Behavior

In terms of computation, the Spalart-Allmaras model is the least expensive turbulence model of the options, since only one turbulence transport equation is solved.

The standard k-ε model clearly requires more computational effort than the Spalart-Allmaras model since an additional transport equation is solved. The realizable k-ε model requires only slightly more computational effort than the standard k-ε model. However, due to the extra terms and functions in the governing equations and a greater degree of non-linearity, computations with the RNG k-ε model tend to take 10-15% more CPU time than with the standard k-ε model. Like the k-ε models, the k-ω models are also two-equation models, and thus require about the same computational effort. On average, the RSM requires 50-60% more CPU time per iteration compared to the k-ε and k-ω models. Furthermore, 15-20% more memory is needed.
Aside from the time per iteration, the choice of turbulence model is due to the ability to obtain a converged solution. For example, the standard $k-\varepsilon$ model is known to be slightly over-diffusive in certain situations, while the RNG $k-\varepsilon$ model is designed such that the turbulent viscosity is reduced in response to high rates of strain. Similarly, the RSM may take more iterations to converge than the $k-\varepsilon$ and $k-\omega$ models due to the strong coupling between the Reynolds stresses and the mean flow.

There are some reasons behind the selection of Spalart-Allmaras model. This model requires least computational time and minimum storage and memory of CPU during simulations. It is also to this point the least expensive turbulence model among other options. Furthermore, the robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its necessity for this study. The all other turbulent models clearly require more computational effort than the Spalart-Allmaras model since an additional transport equation is solved (discussed earlier in this section). For the simple pipe networks geometry investigated in this study the Spalart and Allmaras model is the automatic choice for turbulence modelling.

### 3.6 THE SPALART-ALLMARAS MODEL

In turbulence models that employ the Boussinesq approach, the central issue is how the eddy viscosity is computed. The model proposed by Spalart and Allmaras (1992) solves a transport equation for a quantity that is a modified form of the turbulent kinematic viscosity.

#### 3.6.1 Transport Equation for the Spalart-Allmaras Model

The transported variable in the Spalart-Allmaras model, $\tilde{\nu}$, is identical to the turbulent kinematic viscosity except in the near-wall (viscous-affected) region. The transport equation for $\tilde{\nu}$ is
\[
\frac{\partial}{\partial x_i} \left( \rho_m \nabla \nabla m \right) = G_v + \frac{1}{\sigma_v} \left[ \frac{\partial}{\partial x_j} \left( \left( \mu_m + \rho_m \nabla \nabla \right) \frac{\partial}{\partial x_j} \nabla \right) + C_{b2} \rho_m \left( \frac{\partial}{\partial x_j} \nabla \right)^2 \right] - Y_v + S_v \tag{6-1}
\]

where \( G_v \) is the production of turbulent viscosity and \( Y_v \) is the destruction of turbulent viscosity that occurs in the near-wall region due to wall blocking and viscous damping. \( \sigma_v \) and \( C_{b2} \) are constants and \( \nu \) is the molecular kinematic viscosity. \( S_v \) is a user-defined source term. Note that the turbulence kinetic energy \( k \) is not calculated in the Spalart-Allmaras model.

### 3.6.2 Modelling the Turbulent Viscosity

The turbulent viscosity, \( \mu_t \), is computed from

\[
\mu_t = \rho_m \nu v f_v \tag{6-2}
\]

where the viscous damping function, \( f_{v1} \), is given by

\[
f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^{-3}} \tag{6-3}
\]

and

\[
\chi \equiv \frac{\nabla}{\nu} \tag{6-4}
\]

### 3.6.3 Modelling the Turbulent Production

The production term, \( G_v \), is modelled as

\[
G_v = C_{b1} \rho_m \tilde{S} \tilde{v} \tag{6-5}
\]

where

\[
\tilde{S} \equiv S + \frac{\nabla}{\kappa^2 d^2} f_{v2} \tag{6-6}
\]

and

\[
f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}} \tag{6-7}
\]
$C_{b1}$ and $\kappa$ are constants, $d$ is the distance from the wall, and $S$ is a scalar measure of the deformation tensor. As in the original model proposed by Spalart and Allmaras, $S$ is based on the magnitude of the vorticity:

$$S = \sqrt{2\Omega_{ij}\Omega_{ij}} \quad [6-8]$$

where $\Omega_{ij}$ is the mean rate-of-rotation tensor and is defined by

$$\Omega_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right) \quad [6-9]$$

The justification for the default expression for $S$ is that, for the wall-bounded flows that were of most interest when the model was formulated, turbulence is found only where vorticity is generated near walls. However, it has since been acknowledged that one should also take into account the effect of mean strain on the turbulence production, and a modification to the model has been proposed (Dacles-Mariani at el. 1995) and incorporated into this numerical simulation.

This modification combines measures of both rotation and strain tensors in the definition of $S$:

$$S = |\Omega_{ij}| + C_{prod} \min(0, |S_{ij}| - |\Omega_{ij}|) \quad [6-10]$$

where

$$C_{prod} = 2.0, \quad |\Omega_{ij}| = \sqrt{2\Omega_{ij}\Omega_{ij}}, \quad |S_{ij}| = \sqrt{2S_{ij}S_{ij}}$$

with the mean strain rate, $S_{ij}$, defined as

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \quad [6-11]$$

Including both the rotation and strain tensor reduce the production of eddy viscosity and consequently reduce the eddy viscosity itself in regions where the measure of vorticity exceeds that of strain
rate. One such example can be found in vortical flows, i.e., flow near the core of a vortex subjected to a pure rotation where turbulence is known to be suppressed. Including both the rotation and strain tensors more correctly accounts for the effects of rotation on turbulence. The standard implementation (including the rotation tensor only) tends to over-predict the production of eddy viscosity and hence over-predicts the eddy viscosity itself in certain circumstances.

3.6.4 Modelling the Turbulent Destruction

The destruction term is modelled as

\[ Y_\nu = C_{w1}\rho_m f_w \left( \frac{\tilde{\nu}}{d} \right)^2 \]  \hspace{1cm} [6-12]

where

\[ f_w = g \left[ \frac{1 + C_{w3}}{g^6 + C_{w3}} \right]^{\frac{1}{6}}, \quad g=r+C_{w2}(r^6-r), \quad \text{and} \quad r = \frac{\tilde{\nu}}{\tilde{S} \kappa^2 d^2}. \]

\( C_{w1}, C_{w2}, \) and \( C_{w3} \) are constants, and \( \tilde{S} \) is given by Equation [6-6]. Note that the modification described above to include the effects of mean strain on \( S \) will also affect the value of \( \tilde{S} \) used to compute \( r \).

3.6.5 Model Constants

The model constants \( C_{b1}, C_{b2}, \sigma, C_v, C_{w1}, C_{w2}, C_{w3}, \) and \( \kappa \) have the following values (Spalart and Allmaras 1992) that were used in this CFD investigation:

\[ C_{b1} = 0.1335, \quad C_{b2} = 0.622, \quad \sigma = 2/3, \quad C_v = 7.1 \]

\[ C_{w1} = \frac{C_{b1}}{\kappa^2} + \frac{(1 + C_{b2})}{\sigma}, \quad C_{w2} = 0.3, \quad C_{w3} = 2.0, \quad \kappa = 0.4187 \]
3.6.6 Wall Boundary Conditions

At walls, the modified turbulent kinematic viscosity, $\nu^*$, is set to zero. When the mesh is fine enough to resolve the laminar sublayer, the wall shear stress is obtained from the laminar stress-strain relationship:

$$\frac{u}{u_\tau} = \frac{\rho_m u y}{\mu}$$  \[6-13\]

If the mesh is too coarse to resolve the laminar sublayer, it is assumed that the centroid of the wall-adjacent cell falls within the logarithmic region of the boundary layer, and the law-of-the-wall is employed:

$$\frac{u}{u_\tau} = \frac{1}{\kappa} \ln E \left( \frac{\rho_m u y}{\mu} \right)$$  \[6-14\]

where $u$ is the velocity parallel to the wall, $u_\tau$ is the shear velocity, $y$ is the distance from the wall, $\kappa$ is the von Kármán constant (0.4187), and $E = 9.793$.

3.7 TURBULENT BOUNDARY CONDITIONS

3.7.1 Flow Inlets and Exits

The numerical model has a wide range of boundary conditions that permit the flow to enter and exit the solution domain. To select the most appropriate boundary condition for the application, this section includes descriptions of how each type of condition is used, and what information is needed for each one. Recommendations for determining inlet values of the turbulence parameters are also provided.

Using Flow Boundary Conditions

The boundary zone types that are frequently used and also available in commercial software Fluent 6.1 (2001) for the specification of flow inlets and exits, are as follows: velocity inlet, pressure inlet,
mass flow inlet, pressure outlet, pressure far-field, outflow, inlet vent, intake fan, outlet vent, and exhaust fan.

The inlet and exit boundary condition options, which are commonly used in numerical simulation, are as follows:

**Velocity inlet boundary conditions**: these are used to define the velocity and scalar properties of the flow at inlet boundaries.

**Pressure inlet boundary conditions**: these are used to define the total pressure and other scalar quantities at flow inlets.

**Mass flow inlet boundary conditions**: these are used in compressible flows to prescribe a mass flow rate at an inlet. When density is constant, it is not necessary to use mass flow inlets in incompressible flows; velocity inlet boundary conditions will fix the mass flow.

**Outflow boundary conditions**: these are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. They are appropriate where the exit flow is close to a fully developed condition, as the outflow boundary condition assumes a zero normal gradient for all flow variables except pressure. They are not appropriate for compressible flow calculations.

**Pressure outlet boundary conditions**: These are used to define the static pressure at flow outlets (and also other scalar variables, in case of backflow). The use of a pressure outlet boundary condition instead of an outflow condition often results in a better rate of convergence when backflow occurs during iteration.

**Pressure far-field boundary conditions**: these are used to model a free-stream compressible flow at infinity, with free-stream Mach number and static conditions specified. This boundary type is available only for compressible flows.
Inlet vent boundary conditions: these are used to model an inlet vent with a specified loss coefficient, flow direction, and ambient (inlet) total pressure and temperature.

Intake fan boundary conditions: these are used to model an external intake fan with a specified pressure jump, flow direction, and ambient (intake) total pressure and temperature.

Outlet vent boundary conditions: these are used to model an outlet vent with a specified loss coefficient and ambient (discharge) static pressure and temperature.

Exhaust fan boundary conditions: these are used to model an external exhaust fan with a specified pressure jump and ambient (discharge) static pressure.

In this study, only first two were considered as the turbulent boundary conditions for the inlet velocity and outflow depending upon the geometry and physical problems that described the drinking water distribution networks sufficiently enough.

Determining Turbulence Parameters

At the time of entering the flow into the domain at an inlet, outlet, or far-field boundary, the model requires specification of transported turbulence quantities. In this section the quantities are described which are needed for specific turbulence models and how they must be specified. The guidelines are also provided for the most appropriate way of determining the inflow boundary values. Turbulence quantities can be specified using non-uniform profiles if it is important to accurately represent a boundary layer or fully developed turbulent flow at the inlet. This should ideally be performed by setting the turbulence quantities by creating a boundary profile file from experimental data, empirical, or analytical formulas. If an analytical description of the profile is available, rather than data points, then either this analytical description to
create a boundary profile file, or create a user-defined function to provide the inlet boundary information can be used.

Once the profile function for the transported turbulence quantities is created, it can be used for the selected Spalart-Allmaras turbulent models. To simulate with this model, turbulent viscosity or turbulent viscosity ratio should be set. The commercial software Fluent 6.1 (2001) computes the boundary value for the modified turbulent viscosity, \( \nu \), by combining \( \mu_t/\mu \) with the appropriate values of density and molecular viscosity.

### 3.7.2 Uniform Specification of Turbulence Quantities

To specify a uniform value of the turbulence quantity at the boundary where inflow occurs is sometimes appropriate. Examples are fluid entering a duct, far-field boundaries, or even fully-developed duct flows where accurate profiles of turbulence quantities are unknown. In most turbulent flows, higher levels of turbulence are generated within shear layers than enter the domain at flow boundaries, making the result of the calculation relatively insensitive to the inflow boundary values. Nevertheless, caution must be taken to ensure that boundary values are not so unphysical as to contaminate your solution or impede convergence. This is particularly true of external flows where unphysically large values of effective viscosity in the free stream can swamp the boundary layers.

The turbulence specification methods described above to enter uniform constant values can be used instead of profiles. Alternatively, the turbulence can be specified in terms of more convenient quantities such as turbulence intensity, turbulent viscosity ratio, hydraulic diameter, and turbulence length scale can be specified. These quantities are discussed further in the following sections.
**Turbulence Intensity**

The turbulence intensity, $I$, is defined as the ratio of the root-mean square of the velocity fluctuations, $u'$, to the mean flow velocity, $u_{avg}$.

A turbulence intensity of 1% or less is generally considered low and turbulence intensities greater than 10% are considered high. Ideally, one should have a good estimate of the turbulence intensity at the inlet boundary from measured data. For example, simulating a wind tunnel experiment, the turbulence intensity in the free stream is usually available from the tunnel characteristics.

The turbulence intensity for internal flows is totally dependent on the upstream history of the flow. If the upstream flow was under-developed and undisturbed, low turbulence intensity can be used. If the flow is fully developed, the turbulence intensity may be as high as a few percent. The turbulence intensity at the core of a fully developed duct flow can be estimated from the following formula derived from an empirical correlation for pipe flows (Fluent 2001):

$$I = \frac{u'}{u_{avg}} = 0.16(Re_{D_i})^{1/8}$$  \[7-1\]

At a Reynolds number of 50,000, for example, the turbulence intensity will be 4%, according to this formula.

**Turbulence Length Scale and Hydraulic Diameter**

The turbulence length scale, $\ell$, is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows. In fully-developed duct flows, $\ell$ is restricted by the size of the duct, since the turbulent eddies cannot be larger than the duct. An approximate relationship between $\ell$ and the physical size of the duct is (Fluent 2001)

$$\ell = 0.07L$$  \[7-2\]
where \( L \) is the relevant dimension of the duct. The factor of 0.07 is based on the maximum value of the mixing length in fully-developed turbulent pipe flow, where \( L \) is the diameter of the pipe. In a channel of noncircular cross-section, \( L \) will be the hydraulic diameter.

If the turbulence derives its characteristic length from an obstacle in the flow, such as a perforated plate, it is more appropriate to base the turbulence length scale on the characteristic length of the obstacle rather than on the duct size.

It should be noted that the relationship of Equation \([7-2]\), which relates a physical dimension \( (L) \) to the turbulence length scale \( (\ell) \), is not necessarily applicable to all situations. For most cases, however, it is a suitable approximation.

Guidelines for choosing the characteristic length \( L \) or the turbulence length scale \( \ell \) for selected flow types are listed below:

- For fully-developed internal flows the hydraulic diameter \( L = D_H \).
- For flows downstream of turning vanes, perforated plates, etc. the characteristic length of the flow opening.
- For wall-bounded flows in which the inlets involve a turbulent flow the boundary-layer thickness, \( \delta_{99} \), to compute the turbulence length scale, \( \ell \), from \( \ell = 0.4\delta_{99} \). The value for \( \ell \) will be equal to the hydraulic diameter \( L \).

**Turbulent Viscosity Ratio**

The turbulent viscosity ratio, \( \frac{\mu_t}{\mu} \), is directly proportional to the turbulent Reynolds number (\( \text{Re}_t \equiv k^2/(\varepsilon V) \)). \( \text{Re}_t \) is large (of the order of 100 to 1000) in high-Reynolds-number boundary layers, shear layers, and fully-developed duct flows. However, at the free-stream
boundaries of most external flows, $\mu_t/\mu$ is fairly small. Typically, the turbulence parameters are set so that $1 < \mu_t/\mu < 10$.

### 3.7.3 Relationships for Deriving Turbulence Quantities

To obtain the values of transported turbulence quantities from more convenient quantities such as $I$, $L$, or $\mu_t/\mu$, one must typically resort to an empirical relation. Several useful relations, most of which are used within the turbulent model are presented below.

#### Estimating Modified Turbulent Viscosity from Turbulence Intensity and Length Scale

To obtain the modified turbulent viscosity, $\tilde{\nu}$, for the Spalart-Allmaras model from the turbulence intensity, $I$, and length scale, $\ell$, the following equation can be used:

$$\tilde{\nu} = \sqrt{\frac{3}{2}} u_{avg} I \ell$$  \[7-3\]

In this turbulent model the modified turbulent viscosity can be computed using the Equation [7-3] and viscosity ratio can estimated by combining $\mu_t/\mu$ with the appropriate values of density and molecular viscosity.

### 3.8 NUMERICAL SIMULATION AND GEOMETRY CONSTRUCTION

#### 3.8.1 Commercial Software

In recent time there are many CFD code available to simulate different flow behavior. Among them FLUENT, CFX, StarCD, and SWIFT are very well known in this CFD area. FLUENT is the best yet to date because it is a state-of-the-art computer program for modelling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge,
and mixed (hybrid) meshes. FLUENT also allows to refine or coarsen grid based on the flow solution.

This solution-adaptive grid capability is particularly useful for accurately predicting flow fields in regions with large gradients, such as free shear layers and boundary layers. In comparison to solutions on structured or block structured grids, this feature significantly reduces the time required to generate a good grid. Solution-adaptive refinement makes it easier to perform grid refinement studies and reduces the computational effort required to achieve a desired level of accuracy, since mesh refinement is limited to those regions where greater mesh resolution is needed.

FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all made possible. In addition, FLUENT uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstations and powerful compute servers, for efficient execution, interactive control, and complete flexibility of machine or operating system type.

Earlier of the project the researcher had to use the old version FLUENT 5. Now the current version FLUENT 6.1 was used in this study. It has the same great advantages than earlier version of FLUENT. For example, there is no limit of secondary phases, which was limited to only one in previous version of FLUENT.

3.8.2 The Grid Topology

Since FLUENT is an unstructured solver, it uses internal data structures to assign an order to the cells, faces, and grid points in a mesh and to maintain contact between adjacent cells. It does not, therefore, require i, j, k indexing to locate neighboring cells. This gives the flexibility to use the grid topology that is best for the
physical problem. Since the solver does not force an overall structure or topology on the grid. In 2D, quadrilateral and triangular cells are accepted, and in 3D, hexahedral, tetrahedral, pyramid, and wedge cells can be used. Both single-block and multi-block structured meshes are acceptable, as well as hybrid meshes containing quadrilateral and triangular cells or hexahedral, tetrahedral, pyramid, and wedge cells.

In addition, FLUENT also accepts grids with hanging nodes (i.e., nodes on edges and faces that are not vertices of all the cells sharing those edges or faces). Grids with non-conformal boundaries (i.e., grids with multiple sub-domains in which the grid node locations at the internal sub-domain boundaries are not identical) are also acceptable.

Choosing the Appropriate Grid Type

FLUENT can use grids comprised of triangular or quadrilateral cells (or a combination of the two) in 2D, and tetrahedral, hexahedral, pyramid, or wedge cells (or a combination of these) in 3D. The choice of which mesh type to use will depend on the application. To choose mesh type the following issues should be considered:

- Setup time
- Computational expense
- Numerical diffusion

To clarify the trade-offs inherent in the choice of mesh type, these issues are discussed further.

Setup Time

Many flow problems solved in engineering practice involve complex geometries. The creation of structured or block-structured grids (consisting of quadrilateral or hexahedral elements) for such problems can be extremely time-consuming, if not impossible. Setup time for complex geometries is, therefore, the major
motivation for using unstructured grids employing triangular or tetrahedral cells. If the geometry is relatively simple, however, there may be no clear saving in setup time with either approach.

**Computational Expense**

When geometries are complex or the range of length scales of the flow is large, a triangular/tetrahedral mesh can often be created with far fewer 5-10 cells than the equivalent mesh consisting of quadrilateral/hexahedral elements. This is because a triangular/tetrahedral mesh allows cells to be clustered in selected regions of the flow domain, whereas structured quadrilateral/hexahedral meshes will generally force cells to be placed in regions where they are not needed. Unstructured quadrilateral/hexahedral meshes offer many of the advantages of triangular/tetrahedral meshes for moderately-complex geometries.

One characteristic of quadrilateral/hexahedral elements that might make them more economical in some situations is that they permit a much larger aspect ratio than triangular/tetrahedral cells. A large aspect ratio in a triangular/tetrahedral cell will invariably affect the skewness of the cell, which is undesirable as it may impede accuracy and convergence.

Therefore, the relatively simple geometry in which the flow conforms well to the shape of the geometry, such as a long thin duct, a mesh of high-aspect-ratio quadrilateral/hexahedral cells can be used. The mesh is likely to have far fewer cells than if you use triangular/tetrahedral cells.

**Numerical Diffusion**

A dominant source of error in multidimensional situations is numerical diffusion, also termed *false diffusion*. (The term false diffusion is used because the diffusion is not a real phenomenon,
yet its effect on a flow calculation is analogous to that of increasing the real diffusion coefficient.)

The following points can be made about numerical diffusion:

- Numerical diffusion is the most noticeable when the real diffusion is small, that is, when the situation is convection-dominated.
- All practical numerical schemes for solving fluid flow contain a finite amount of numerical diffusion. This is because numerical diffusion arises from truncation errors that are a consequence of representing the fluid flow equations in discrete form.
- The second-order discretization scheme used in FLUENT can help reduce the effects of numerical diffusion on the solution.
- The amount of numerical diffusion is inversely related to the resolution of the mesh. Therefore, one way of dealing with numerical diffusion is to refine the mesh.
- Numerical diffusion is minimized when the flow is aligned with the mesh.

The last point is the most relevant to the choice of the grid. It is clear that if a triangular/tetrahedral mesh can be used the flow can never be aligned with the grid. On the other hand, if a quadrilateral/hexahedral mesh can be used this situation might occur, but not for complex flows. It is only in a simple flow, such as the flow through a long duct, in which we can rely on a quadrilateral/hexahedral mesh to minimize numerical diffusion. In such situations, there might be some advantage to using a quadrilateral/hexahedral mesh, since it will be able to get a better solution with fewer cells than using a triangular/tetrahedral mesh.

**3.8.3 Grid Requirements and Considerations**

This section contains information about special geometry/grid requirements and general comments on mesh quality.
Geometry or Grid Requirements

The following geometry setup and grid construction requirements at the beginning of the problem setup are to be considered:

Axisymmetric geometries must be defined such that the axis of rotation is the x axis of the Cartesian coordinates used to define the geometry.

FLUENT allows setting up periodic boundaries using either conformal or non-conformal periodic zones. For conformal periodic boundaries, the periodic zones must have identical grids. The conformal periodic boundaries can be created in GAMBIT or TGrid to generate the volume mesh. Although GAMBIT and TGrid can produce true periodic boundaries, most CAD packages do not. In this study the GAMBIT 2.2 supported by FLUENT has been used to generate mesh geometry for the pipe networks.

Mesh Quality

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. The attributes associated with mesh quality are node point distribution, smoothness, and skewness.

Node Density and Clustering

Since a continuous domain is defined discretely, the degree to which the salient features of the flow (such as shear layers, separated regions, shock waves, boundary layers, and mixing zones) are resolved depends on the density and distribution of nodes in the mesh. In many cases, poor resolution in critical regions can dramatically alter the flow characteristics. For example, the prediction of separation due to an adverse pressure gradient depends heavily on the resolution of the boundary layer upstream of the point of separation.
Resolution of the boundary layer (i.e., mesh spacing near walls) also plays a significant role in the accuracy of the computed wall shear stress and heat transfer coefficient. This is particularly true in laminar flows where the grid adjacent to the wall should obey

\[ y_p \sqrt{\frac{u_\infty}{\nu x}} \leq 1 \]  

where \( y_p \) = distance to the wall from the adjacent cell centroid

\( u_\infty \) = free-stream velocity

\( \nu \) = kinematic viscosity of the fluid

\( x \) = distance along the wall from the starting point of the boundary layer Equation [8-1] is based upon the Blasius solution for laminar flow over a flat plate at zero incidence (1979).

Proper resolution of the mesh for turbulent flows is also very important. Due to the strong interaction of the mean flow and turbulence, the numerical results for turbulent flows tend to be more susceptible to grid dependency than those for laminar flows. In the near-wall region, different mesh resolutions are required depending on the near-wall model being used.

In general, no flow passage should be represented by fewer than 5 cells. Most cases will require many more cells to adequately resolve the passage. In regions of large gradients, as in shear layers or mixing zones, the grid should be fine enough to minimize the change in the flow variables from cell to cell. Unfortunately, it is usually very difficult to determine in advance the locations of important flow features. Moreover, the grid resolution in most complicated three-dimensional flow fields will be constrained by CPU time and computer resource limitations (i.e., memory and disk space). Although accuracy increases with larger grids, the CPU and memory requirements to compute the solution and postprocess the results also increase. Solution-adaptive grid refinement can be used...
to increase and/or decrease grid density based on the evolving flow field, and thus provides the potential for more economical use of grid points (and, hence, reduced time and resource requirements).

**Smoothness**

Rapid changes in cell volume between adjacent cells translate into larger truncation errors. Truncation error is the difference between the partial derivatives in the governing equations and their discrete approximations. FLUENT provides the capability to improve the smoothness by refining the mesh based on the change in cell volume or the gradient of cell volume.

**Cell Shape**

The shape of the cell (including its skewness and aspect ratio) also has a significant impact on the accuracy of the numerical solution. Skewness can be defined as the difference between the cell's shape and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. For example, optimal quadrilateral meshes will have vertex angles close to 90 degrees, while triangular meshes should preferably have angles of close to 60 degrees and have all angles less than 90 degrees.

Aspect ratio is a measure of the stretching of the cell. As discussed earlier, for highly anisotropic flows, extreme aspect ratios may yield accurate results with fewer cells. However, a general rule of thumb is to avoid aspect ratios in excess of 5:1.

**Flow-Field Dependency**

The effect of resolution, smoothness and cell shape on the accuracy and stability of the solution process is strongly dependent on the flow field being simulated. For example, very skewed cells can be tolerated in benign/mild type of flow regions, but can be very
damaging in regions with strong flow gradients. Since the locations of strong flow gradients generally cannot be determined a priori, one should strive to achieve a high-quality mesh over the entire flow domain.

3.9 CONCLUSION

A comprehensive 3D numerical investigation that solves the hydrodynamics of particles flowing through a horizontal pipe networks in distribution system was studied making use of the least expensive or the most appropriate multiphase mixture model. For the turbulent quantities the Spalart-Allmaras model has been used since the robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its necessity for this study. The all other turbulent models clearly require more computational effort than the Spalart-Allmaras model since an additional transport equation is solved.

A well known CFD package FLUENT has been for numerical modelling because of its flexibility, multiplicity of solving flow problems with different meshes, and user friendly compare to other CFD code available in the current market.

The CFD investigation (simulation and results) of turbidity spikes (caused by solid particles) flowing through the drinking water distribution networks the above mentioned models have been presented in the following chapters.
Chapter 4
Part 1: Particle Deposition in a Pipe Loop

4.1 INTRODUCTION

In this chapter a comprehensive 3D numerical investigation of hydrodynamics of particles flowing through a horizontal pipe loop consisting of four bends has been modeled. In this numerical simulation five different particles have been used as secondary phases to calculate the hydrodynamic behavior of particles in which inter-particle interaction has been implemented. The deposition of particles, along the periphery of the wall and at different depths in the pipe, has been investigated. The effect of particle size and fluid velocity has also been investigated. A high concentration of particle is seen at the bottom wall of the pipe loop at the entry of the bend; whereas, at the downstream of the bend this deposition was relatively lower near the bottom wall. However, this deposition was seen at 60° inner wall from the bottom (explained later). The larger particles clearly showed a tendency to deposit on the bottom wall except downstream of the bend, but the smaller particles showed less tendency to deposit, especially at higher velocity. This numerical investigation showed good agreement with the experimental results of Grainger et al. (2003).

In the following sections a brief review of the previous works on understanding the particle deposition in turbulent flow through horizontal pipe loop has been demonstrated. Rest of the sections
has been dealt with detailed CFD investigation of particle deposition around a horizontal test rig followed by model validation. The motivation for this study is two-fold, to investigate the deposition of solid spherical particles with a specific gravity of 1.64 similar to those in dispersed multiphase flows in a water supply distribution networks and in nature. The dispersed phases of such flows consisted of particles with diameters ranging from 2 µm to approximately 20 µm for different Reynolds numbers. These above mentioned particle and fluid parameters were also used by Grainger et al. (2003).

The second point of interest of this study was to investigate the segregation of solid particles along the circumference of the pipe wall at upstream and downstream sides of different bends and at a certain measuring point (Figure 4-1).

![Figure 4-1: Schematic diagram of the pipe loop with four 90° bends.](image)

In order to study the hydrodynamics of particles behavior in a turbulent flow field numerically, the geometry shown in Figure 4-1 comprising a 41 m long and 100 mm diameter pipe, closed-loop with four 90° bends was modelled with a axial flow pump at the upstream of first bend same as Grainger et al. (2003) used in their experiment.

Table 4-1 shows the physical parameters of the geometry and hydraulic characteristics of the fluid flow system that has been used for the CFD investigation. The velocities range from 0.05 to 0.4 ms⁻¹ chosen to compare this study with the experimental results. The
spherical solid particles having the density of 1640 kgm\(^{-3}\) were modelled. These fluid velocities and the particle density are typical of those found in the drinking water distribution networks of South East Water Ltd. (SEWL), Sydney Water, Melbourne Water, etc. Table 4-1 shows the particle sizes, which also can be found in the current distribution networks around Australia (Grainger et al. 2003).

Table 4-1: Physical and hydraulic characteristics of the system used for CFD simulation

<table>
<thead>
<tr>
<th>Characteristics</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pipe loop length (m)</td>
<td>41.0</td>
</tr>
<tr>
<td>Diameter of the pipe D (m)</td>
<td>0.1</td>
</tr>
<tr>
<td>No. of phases</td>
<td>6</td>
</tr>
<tr>
<td>VF of each secondary phases (ppm)</td>
<td>310</td>
</tr>
<tr>
<td>Average water velocities (ms(^{-1}))</td>
<td>0.05, 0.1, 0.2, 0.3, and 0.4</td>
</tr>
<tr>
<td>Particle density (kgm(^{-3}))</td>
<td>1640</td>
</tr>
<tr>
<td>Particles sizes (µm)</td>
<td>2, 5, 10, 15, and 20</td>
</tr>
<tr>
<td>No of computational cells</td>
<td>129654 (produce grid independence result)</td>
</tr>
</tbody>
</table>

4.2 NUMERICAL TECHNIQUE

Table 4-1 represents the boundary conditions and the dimensions of the geometry that were used for the CFD modelling. Since this was a closed loop pipe (See Figure 4-1) an axial flow pump was used to maintain the velocities relative to its pressure boundary conditions.

In literature only the velocity data were available instead of pump specification data. Therefore a different technique had been applied to find out the parameters that were required for a pump. A similar geometry was used with inlet and outlet boundary conditions in the very first simulation. Using the velocity inlet and outflow boundary conditions the pressure drop was calculated between the inlet and outlet of the pipe.

To maintain homogeneity of the particle distribution along the pipe loop, the solution domain was initialized with required volume fraction (VF) for each particle size. A two steps procedure was used to obtain a converged solution. Firstly, an intermediate converged
solution was obtained solving only the mass and momentum equations. Secondly, all equations including turbulence and volume fraction equations were solved until a fully converged solution was obtained by reducing the residuals by the three orders of magnitude.

For postprocessing the integral values at required point surfaces were imported and then plotted in graphs using excel broad sheet (MS Excel).

4.3 RESULT AND DISCUSSION

4.3.1 Model Validation

In the following two sections the results of the simulations were validated against the experimental data of Grainger et al. (2003). The validation has been carried out in two stages. First, the volume fraction of particle as a function of height at the measuring point (see Figure 4-1) of experiment has been compared with that of numerical results (Figure 4-2); secondly the cumulative volume fraction of particle as a function of velocity at the center of the pipe cross section has been compared with that of numerical simulation (Figure 4-3).

Figure 4-2: Comparison of CFD results and experimental data (Grainger et al. 2003) for the velocity 0.4 ms\(^{-1}\) at center of the pipe (D is the diameter of pipe).
Figure 4-2 represents the cumulative particle volume fraction (summation of all particles) as a function of height across the pipe at a certain location in the loop (measuring point, see Figure 4-1) at the velocity of 0.4 ms\(^{-1}\). Figure 4-2 shows a similar tendency of the particle concentration between experimental and CFD results. Nevertheless, the trend is similar, but the experimental results show lower volume fraction than that of simulation results. This is because of the some shortcomings of the measuring instruments that were used in the experiment. Some distinguishing features of the physical characteristics of particles reported by Grainger \textit{et al.} (2003), which resulted in lower concentration of particle showing in Figures 4-2 and 4-3 were as follows:

- Particles larger than 20 \(\mu\text{m}\) could not be detected by the instruments. However, particle distribution (Graingerg et al. 2003) showed some particles larger than 20 \(\mu\text{m}\) still were introduced. Some of them were even as large as 100 \(\mu\text{m}\) (these particles were collected from the main flush). In the CFD simulation the 20 \(\mu\text{m}\) particles, which were one fifth of total particles (see Table 4-1), resulted in higher concentration of particle (Figure 4-2).

- The particles introduced into the experiment were not identical in shape. They might be wedge, tetrahedral, cubic or semi spherical, which may behave peculiarly in deposition. However, in CFD simulation the particles were spherical in shape, which resulted in higher deposition.

- In CFD investigation the average density of particle was used, which was 1640 kgm\(^{-3}\), but these were not constant in experiment. There were some lighter particles, which may not deposit on the bottom wall resulting in lower deposition at very bottom of the pipe wall (Figure 4-2). However, the heavier particles introduced in experiment does not show on
the top region of the pipe resulting in same concentration of particle.

- It was also reported that some particles might have attached to the wall of the pipe, which were no longer travelling with the flow, were not counted in the experiment. This resulted in lower concentration for experimental data as compared with the CFD simulation.

- In experiment the particles were injected in pulse into the system. An axial flow pump had also been used to make the distribution of particle homogeneous. But in fact, the particle distribution was not completely homogeneous at all. Although in CFD simulation homogeneous distribution was perfectly achieved, which also resulted in higher concentration around the pipe.

However, at the bottom of the pipe wall the deference between CFD results and experimental data is nearly 15% and at the top wall it is less than 8% (Figure 4-2).

Figure 4-3: Comparison of CFD results and experimental data (Grainger et al. 2003) for different velocities at center of the pipe.

Figure 4-3 shows the particle volume fraction for both CFD and experimental results as a function of velocity at the center of vertical cross section plane at measuring point (Figure 4-1). As expected the experimental results show lower particle volume
fraction than that of CFD. The reasons behind this have been explained in the earlier section. This difference is more pronounced for lower velocity region, which can be explained by:

- The particles introduced into the experimental system were reported lost during the experiment, which resulted in lower volume fraction of particle.
- The lighter particles (specific gravity less than 1) may have floated or stuck to the pipe wall and not found at center of the pipe at lower velocity. This may have resulted in lower volume fraction at low velocity.
- The heavier particles (specific gravity ranges from 3 to 6) may have settled quickly just after injection into the system and lost as well at lower velocity.

Due to multiplicity of particle identity (different sizes, shapes, and densities), the particle deposition is not linear with fluid flow for experimental results. At lower velocity only lighter particles (sometimes even lighter than water) were found suspended, which resulted in less particle concentration in the central region of the pipe cross section. However, due to turbulence, which increases as velocity increases, the lost and lighter particles may have re-entrained into the system and been found at center of the pipe. This resulted in marginally lower volume fraction (VF) of particles at higher velocity than that of CFD results shown in Figure 4-3. At higher velocity the larger and heavier (than water) particles were also re-entrained into the system in the experiment and were found in the central region, which may also have been the cause of the marginal lower particle concentration than the CFD results.

4.3.2 Particle Concentration at Different Depths across the Pipe

Figures 4-4 [a-f] show the contour plot of total particles volume fraction at different heights for the velocity of $0.05 \text{ ms}^{-1}$. The result
shown in Figures 4-4 [a-f] were only for 3rd bend at which this study particularly focused on as it was assumed to be free from any disturbances created by pump. These contour plots were for the lower velocity only since similar trends were seen for other velocities with marginal lower volume fraction of particle. The height was measured from bottom of the vertical plane of the pipe. From these plots, it can be seen that bottom wall (Figure 4-4a) has got more deposition than that of center plane (Figure 4-4f). The fluid flow was directed from left to right down as seen in Figure 4-5 below. Particles can deposit up to 30 mm from bottom wall (Figures 4-4 [a-d]). Figures 4-4 [a-c] noticed that the upstream deposition of particle is higher than that of downstream (explained later). Near wall the deposition of particle is more than that of center of the pipe because of higher turbulence due to higher velocity.

![Contour of volume fraction of total particles at different heights](image)

Figure 4-4: Contour of volume fraction of total particles at different heights (a, b, c, d, e, and f are for at 1, 5, 10, 20, 30, and 50 mm measured from bottom, respectively) for the velocity of 0.05 ms$^{-1}$.

Figures 4-5 [a-d] and 4-6 [a-d] represent the velocity vector plotted on a plane at different heights measured from bottom for the velocity of 0.05 ms$^{-1}$ and 0.4 ms$^{-1}$ respectively. As expected near the
wall the velocity is slow resulting in particle deposition near wall shown in Figures 4-4 [a-f].

At inner wall of the bend velocity is higher, however, just after the bend (downstream) as shown in Figures 4-5 [a-d] and 4-6 [a-d], the velocity is slowed down, which resulted more particle deposition (also see the circumferential deposition later in this chapter). After the bend, the magnitude of the velocity is seen higher at outer wall and thus less deposition occurs, which is also explained later.
Figure 4-6: Velocity vector on a plane at different heights (a, b, c, and d are at 5, 10, 30, and 50 mm measured from bottom, respectively) at velocity 0.4 ms⁻¹.

Figure 4-7 shows the particle deposition variation at different depths in the pipe as a function of velocity. In the Figure 4-7 it is shown that above \( y = 0.4D \) (D diameter of the pipe) the particle deposition increased as the velocity increased, whereas, the particle concentration below \( y = 0.4D \) decreased at higher velocity. This is because the higher velocity induces with higher turbulence and caused the particle to re-entrain into the flow and the particle concentration to increase near the top region and decrease below \( y = 0.4D \). This difference in concentration between the top and bottom regions was more pronounced at lower velocity.

Figure 4-7: CFD investigation of particle concentration as function of velocity at different heights across the pipe at the measuring plane.
Figure 4-8a: Relative concentration of particles for different depths along the pipe at 0.05 ms\(^{-1}\).

Figures 4-8 [a-b] show the relative concentration plotted along the pipe for the heights of 0.25D, 0.5D, 0.75D, and 1D from the bottom wall of the pipe. Relative concentration is a dimensionless parameter, which represents the ratio of local particle concentration to that of bottom of the pipe wall. Figures 4-8 [a-b] show a more homogeneous distribution of particles at different depths in the bend region. Due to the high stream line curvature and associated centrifugal force the fluid at different depths the particle become mixed well and resulted in homogeneous particle distribution around the pipe cross sectional area. This homogeneous particle distribution is more pronounced at higher velocity (0.4 ms\(^{-1}\)). At the downstream of the bend, the streamline curvature and associated centrifugal force disappeared and particles started to segregate into different concentration at different depths. This segregation or stratification was more pronounced at the lower velocity. At the higher velocity the particles did not get enough time to segregate between the bends. Higher turbulence at higher velocity also contributed to homogeneity of the particles (Figure 4-8b).
Figure 4-8b: Relative concentration of particles for different depths along the pipe at 0.4 ms$^{-1}$.

4.3.3 Circumferential Particle Deposition at Bends

Figure 4-9 shows a cross-section of the pipe loop near the measuring point. The angle $\theta$ represented the circumferential angle, which increased clockwise. The circumferential angle $\theta = 0^0$ represented the top wall and when $\theta = 180^0$ referred to the bottom wall of the.

Local deposition along the pipe circumference can be obtained from the simulation. Figures 4-10 [a-e] show the circumferential distribution of particle volume fraction at different velocities ranging from 0.05 to 0.4 ms$^{-1}$ (center line velocity) for the 1$^{\text{st}}$, 3$^{\text{rd}}$, and 4$^{\text{th}}$ bends. Since there was a pump just before the 1$^{\text{st}}$ bend, author chose the 3$^{\text{rd}}$ bend along with 1$^{\text{st}}$. 
Figure 4-10a: Cumulative particle deposition as a function of circumferential pipe angles at three different up and down stream of bends at 0.05 ms\(^{-1}\).

Figure 4-10b: Cumulative particle deposition as a function of circumferential pipe angles at three different up and down stream of bends at 0.1 ms\(^{-1}\).

Figure 4-10c: Cumulative particle deposition as a function of circumferential pipe angles at three different up and down stream of bends at 0.2 ms\(^{-1}\).
Particle Deposition in a Pipe Loop

Figure 4-10d: Cumulative particle deposition as a function of circumferential pipe angles at three different up and down stream of bends at 0.3 ms\(^{-1}\).

Figure 4-10e: Cumulative particle deposition (CFD) as a function of circumferential pipe angles at three different up and down stream of bends at 0.4 ms\(^{-1}\).

However, the 2\(^{nd}\) bend was ignored since the deposition tendency was found similar to that of the 4\(^{th}\) bend. The profiles at the upstream of the bend exhibited a smooth circumferential variation with the maximum deposition at the bottom of the pipe. Similar trends for the entry of the bends were observed from the experimental data of Anderson & Russell (1970) and the analytical results of Mols and Oliemans (1998), Laurinat et al (1985), and Hossain et al. (peer review). However, the present CFD results
obtained for Liquid-Solid system can not be compared quantitatively with those obtained for Gas-Solid system.

The peak deposition at the bottom wall was high when the velocity was low. This can be easily explained as particles disperse at higher velocity and can be found in a higher concentration across the cross-section of the pipe due to high turbulence (Laurinat et al. 1985; Mols and Oliemans 1998). However, the trends of the particle deposition at upstream of bends (Figures 4-10 [a-e]) were not symmetrical along vertical plane. This was because the particle entrainment between the bends was governed by the particle distribution of bends. This was different as seen for a straight pipe flow (Laurinat et al. 1985; Mols and Oliemans 1998; Hossain et al. 2003).

Figures 4-10 [a-e] show that the maximum particles deposit at downstream of the bends at 60° skewed from bottom towards inner side of the bends (also see Figures 4-12 [a-e]). As shown in Figures 4-10 [a-e] at downstream of the bend the maximum deposition does not occur at the bottom (180°) rather it occurs at on the inner wall of the bend.

The location of the peak deposition is situated at inner side of the cross-sectional plane at an angular displacement of 60° with respect to the bottom of the wall. The fluid was subjected to centrifugal force as it flowed around the bend. Particles having higher specific gravity became segregated and deposited in the bend region. This deposition was pronounced near the inner wall of the bend exit. Since the turbulent diffusion increased with velocity, the peak deposition of particles was less at the higher velocity (Figures 4-10 [a-e]).
4.3.4 Circumferential deposition at 3rd bend

Figure 4-11 shows a simple schematic diagram of a bend. Each of the bends has been divided into four circumferential planes with equal segment. The angle $\phi$ is measured clockwise positive. The cutting plane across the pipe bend is shown in the Figure 4-11 with its position in degrees. Later in this section the particle deposition will be shown along the periphery of these cutting planes across the pipe-bends.

![Schematic diagram of a bend](image)

Figure 4-11: Schematic diagram of a bend.

Figures 4-12 [a-e] show the contour plot of particle volume fraction on three different cross-sectional planes of 3rd bend only at different velocities. The 3rd bend was assumed to be free from any turbulence caused by the pump. The fluid is flowing from right plane to left plane as shown in Figures 4-5 and 4-6. The right side plane was taken at $\phi = 22.5^0$, middle plane was at $\phi = 45^0$, and left one was at $\phi = 67.5^0$ inclined from upstream plane (see Figures 4-13 [a-e] for more details). The scale shown here was not similar to that of Figures 4-4 [a-f]. This is because the depositions shown in Figures 4-12 [a-e] were for only three planes locally instead of whole solution domain. Figure 4-12a showed a clear deposition at the inner wall for low velocity. This deposition was gradually decreased from right plane to left plane. However, as seen in Figure 4-12a, the peak deposition was moved toward the inner side of the bend, which was seen at the bottom wall ($\theta = 0^0$) for the straight section.
of the pipe loop (the circumferential deposition at upstream side of the bend as shown in Figures 4-10 [a-e]). This deposition became homogeneous with lower deposition at the bottom wall due to the turbulence, which increased with the increased of velocity.

![Contour plot of particle volume fraction](image)

Figure 4-12: Contour plot of particle volume fraction on three different cross-sectional planes for the velocity of 0.05, 0.1, 0.2, 0.3, and 0.4 ms\(^{-1}\) respectively.

In Figures 4-13 [a-e] the volume fraction of particles was plotted as a function of circumferential angles for different planes (\(\phi = 0^0, 22.5^0, 45^0, 67.5^0,\) and \(90^0\)) for the 3\(^{rd}\) bend (Figure 1) at different velocities of 0.05, 0.1, 0.2, 0.3, and 0.4 ms\(^{-1}\)). The planes at angles \(\phi = 0^0\) and \(\phi = 90^0\) were known as upstream and downstream side planes of the bend respectively. The peak deposition (Figures 4-13...
Figure 4-13a: Particle deposition as a function of circumferential pipe angles $\theta$ at different planes of 3rd bend at 0.05 ms$^{-1}$.

Figure 4-13b: Particle deposition as a function of circumferential pipe angles $\theta$ at different planes of 3rd bend at 0.1 ms$^{-1}$.

Figure 4-13c: Particle deposition as a function of circumferential pipe angles $\theta$ at different planes of 3rd bend at 0.2 ms$^{-1}$.
Figure 4-13d: Particle deposition as a function of circumferential pipe angles $\theta$ at different planes of 3rd bend at 0.3 ms$^{-1}$.

Figure 4-13e: Particle deposition as a function of circumferential pipe angles $\theta$ at different planes of 3rd bend at 0.4 ms$^{-1}$.

[a-b]) was moved from the bottom wall ($\theta = 180^0$) to $240^0$ (i.e. $60^0$ skewed to the inner wall towards bend as shown in the contour plot 4-12 [a-e]) as it moved forward to the downstream plane ($\phi = 90^0$). As shown in Figures 4-12e and 4-13e the particle diffusivity increased with velocity resulting in less particles deposition at the bottom inner wall and the peak deposition of particle was decreased.

The particle deposition changed with the velocity and the size of particles presented in Figures 4-14 [a-e]. These figures showed the volume fraction of particle as a function of circumferential angle ($\theta$)
at the upstream and downstream sides (entry and exit) for 3rd bend only at different velocities. For a higher velocity, the particle diffusivity increased as a result of fluid diffusivity, which resulted in less particle deposition on the bottom wall. However, for the particles sizes investigation, it appeared that this diffusivity reached a certain threshold at a velocity of 0.3 ms\(^{-1}\) beyond which the fluid velocity did not produce any change in particle deposition (see Figures 4-14 [d-e]). The deposition that was still visible was due to associated centrifugal force at the bends. The deposition patterns (Figures 4-14 [a-e]) for different particle sizes were similar to that seen in Figures 4-10 [a-e], which indicates that the fluid velocity is the dominant factor in re-entrainment of particles at the periphery wall. As expected the 20 \(\mu\)m particles show relatively pronounced deposition at the bend entry and exit. This peak deposition of the 20 \(\mu\)m as well as other particles reduced with the increasing velocity.

Figure 4-14a: Particle deposition (CFD) as a function of circumferential pipe angles at the up and down stream of 3rd bend for different particles at 0.05 ms\(^{-1}\).
Figure 4-14b: Particle deposition (CFD) as a function of circumferential pipe angles at the up and down stream of 3rd bend for different particles at 0.1 ms$^{-1}$.

Figure 4-14c: Particle deposition (CFD) as a function of circumferential pipe angles at the up and down stream of 3rd bend for different particles at 0.2 ms$^{-1}$.

Figure 4-14d: Particle deposition (CFD) as a function of circumferential pipe angles at the up and down stream of 3rd bend for different particles at 0.3 ms$^{-1}$.
Figure 4-14e: Particle deposition (CFD) as a function of circumferential pipe angles at the up and down stream of 3rd bend for different particles at 0.4 ms⁻¹.

4.4 CONCLUSION

In this chapter the effect of particle sizes and flow velocity on the particle deposition in a horizontal pipe loop consisting of four bends was investigated numerically. The CFD results were validated against the experimental data of Grainger et al. (2003). Good agreement between the simulation results and experiment data was achieved. The relative deposition at each velocity indicated that particles were evenly distributed around bend. The circumferential particle deposition on the planes in between upstream and downstream sides of the bend provided a very clear view that the peak deposition on the downstream side did not occur at the bottom wall, rather skewed 60⁰ to the inner side of the wall. There are very few experimental works have been performed to date that can be compared these CFD simulation results. Furthermore, the larger particles showed relatively pronounced deposition at the bend. However, the peak deposition of all the particles reduced with the increasing velocity.
Chapter 5
Part 2: Turbidity Spikes Movement

5.1 INTRODUCTION

A comprehensive 3D numerical investigation into the hydrodynamics of particles flowing through a horizontal pipe has been presented in this chapter. The multiphase mixture model available in Fluent 6.1 (2001) was used in this study. The hydrodynamics of particles flowing in a turbulent unsteady state flow through a horizontal pipe were studied to compare with current drinking water distribution systems in Australia such as: South East Water Ltd (SEWL), Melbourne Water, Sydney Water, etc. In this simulation six different inlet velocity profiles and particle-load profiles were used to compute particles deposition and re-entrainment into the systems and to identify the conditions of the deposition and suspension mechanisms. The different transient flow profiles were chosen to relate this study with real data of turbidity spike phenomena in the distribution networks. The deposition of particle along the pipe were also investigated. Furthermore, the effect of fluid velocity on the particle loads was investigated. Results showed that after a certain length and time, when the velocity is steady after an initial deceleration period, the shear stress was sufficiently high to cause a particle re-entrainment and deposition phenomenon where the particles roll along the bottom line of pipe wall (Powel, 1983; Mohauas, 1974). When this deposition was large enough, it broke up and accumulated to form another group of particles called secondary kink.
A numerical simulation was carried out by Hossain et al. (2003) for the circumferential particle deposition in a straight pipe for turbulent flow. Hossain et al. (peer review) developed a mathematical model for particle deposition and dispersion, which was extended from the Mols and Oliemans (1998) liquid-gas model to solid-liquid phase for turbulent flow. The researcher also explained the circumferential deposition for the straight pipe. But these models are for steady flow.

The motivation for this study was two-fold, first analyze the turbidity spikes traveling through the pipe for different particular flow profiles (single spike, similar to what is seen in reality (Prince and McManus 1999; Prince et al. 2001). Second point of interest of this study was to investigate the effect of velocity ramp-up (acceleration) and ramp-down (deceleration) on particles cluster movement. These both studies have provided a broad understanding about the hydrodynamics of particles moving together and/or in a cluster as a spike. The deposition patterns and places can be known from these studies.

5.2 BACKGROUND

Drinking water systems are supposed to transport only dissolved matter without visible particles. This is difficult to achieve in reality and some suspended solid are expected. The volume fraction of these suspended solids in the SEWL is very low as compared to other distribution systems around the world, nevertheless, presence of any particles in drinking water causes a deterioration of the water quality. Drinking water of present SEWL distribution system contains Aluminium, Antinomy, Barium, Boron, Cadmium, Chromium, Copper, Iron, Lead, Manganese, Mercury, Molybdenum, Nickel, Selenium, Silica, Zinc (South East Water Ltd. 2004). Those particles are heavier than water but they remain suspended because of their micro size.
This is not a problem when this low volume fraction goes to the customer’s taps because it is not visible. But Rachael et al. (1999; 2001) found that when there is any disturbance in the flow of the distribution systems, SEWL gets more complaints. The complaints came as the SEWL distribution network experienced turbidity spikes at anywhere of the systems. The spikes travel along the pipe networks resulting in dirty water at consumers end.

5.2.1 Definition of Turbidity

Turbidity is defined as an expression of the optical property that causes light to be scattered and absorbed, rather than transmitted, in straight lines through the sample. Simply stated, turbidity is the measure of relative sample clarity. Turbidity is measured in Nephelometric Turbidity Units or NTU, which represents the average volume scattering over a defined angular range. Both particle size and concentration of suspended solids as well as dissolved solids can affect this turbidity. It provides a qualitative estimate of the muddiness or cloudiness of the water due to clay, silt, finely divided organic or inorganic matter, soluble colored organic compounds, plankton, and microscopic organisms. Turbidity essentially refers to how clear the water is. The greater the amount of total suspended solids (TSS) in the water, the murkier it appears and the higher the measured turbidity. The major source of turbidity in a water distribution networks is typically phytoplankton, particulates may also be clays and silts from pipe erosion, re-suspended bottom sediments of the pipe wall, and organic detritus from stream. High flow rates may stir up bottom sediments and increase the cloudiness of the water.

5.2.2 Formation of a Turbidity Spike

The primary cause of turbidity attributed to the accumulation and subsequent re-suspension of colloidal material (Prince et al. 2003). The colloidal material that is accumulated has been attributed to the
turbid water entering from unfiltered source water (0.7–2.3 NTU), as the distribution system itself has few if any unlined mains. Preliminary evidence suggests that this material is predominately clay and silt (Prince et al. 2000) with no health issues due to the protected catchments from where the water originated. This water is dosed with chlorine for disinfection, lime for pH correction, and fluoride for public health considerations.

The primary method of addressing turbidity in water has been to counteract the accumulation of sediment in water mains through cleaning activities. These activities include routine planned flushing from service reservoirs to the extremities of the system, plus isolated flushing following aesthetic related incidents. The cost of these activities is in the order of hundreds of thousands of dollars.

Particles causing problems in the water distribution system can arise from a number of locations, from source water (Yarra Valley Water 1999; Gauthier et al. 1999) and treatment (Gauthier et al. 2001); or generated within the water distribution system itself from pipe and fitting corrosion, erosion (Kallio and Reeks 1989), biological growth (1989; Clark et al. 1993; Gauthier et al. 1996; Brunone et al. 2000; Gauthier et al. 2001), external contamination such as during pipe repairs (Gauthier et al. 1996), and chemical reactions from the formation of iron or manganese oxides (Gauthier et al. 1996; 1989). Results to date indicate that the majority of material in Melbourne’s water mains arises from source water. The Melbourne system receives unfiltered water of turbidity typically less than 2 NTU, which is only likely to cause an aesthetic issue if concentration of particles occurs (most likely due to accumulation in the water mains).

It is generally accepted in literature and in industry that any particles that are deposited at the pipe wall are resuspended into the bulk flow when a critical minimum shear stress is exceeded
(Prince et al. 2003; Boxall et al. 2001; Hoven and Vreeburg 1992). The frequency of these events is related to the frequency at which this minimum shear stress is exceeded, and the severity is due to the amount of material that has accumulated i.e. the time between critical shear stress exceedences. Thus the worst discoloured water events will occur when conditions are favourable for accumulation of material and there is a significant time interval between critical shear stress exceedences. This includes transfer mains that are oversized or during winter flows where the shear stress only infrequently exceeds the critical minimum. In such circumstances operational events such as flushing procedures, bursts, and fire fighting may result in a sudden increase in demand greater than normal daily flow, could cause turbidity spikes event.

A literature survey was able to locate only a few articles that directly referred to operational changes, with all other articles classifying them as abnormal events. Other papers that mention operational events causing turbidity spikes events include Hoven & Vreeburg (1992) – hydrant operations, Gray (1994) – repair and maintenance work.

Therefore, the following reasons of operational events that are responsible for turbidity spikes formation are based on the previous study.

High risk:

- Mains cleaning using flushing
- Material from upstream of the service reservoir
- Water hammer due to valve closure during mains repair
- Routine maintenance of distribution networks

Medium risk:

- Burst/leakage main or hydrant causing high flows
- Typical demand higher than usual causes high mass flow
- Water supplied from alternative source resulting in reverse flows
- Chemical reactions of fluid and pipe wall
- Pipe erosion and corrosion

*Low risk:*
- Hydrant used, either for fire fighting or illegally
- Hydrant or mains repair causing high flows
- Fire sprinkler system
- Pipe vibration due to natural calamities like earthquake etc.
- Dead end (no flow zone) where particles accumulate
- Biological growth of biofilm and/or algae

The ability to determine the origins of the particulate material varies between different water supply systems, with possible sources being from catchment, treatment processes, biofilm growth within the water supply pipes, corrosion products (Jayaratne et al. 2004). There are also some causes of turbidity spikes, which have not yet been identified.

### 5.3 NUMERICAL SIMULATION OF TURBIDITY SPIKES

#### 5.3.1 Online Monitoring Data

Continuous online testing (COLT) refers to the automated on site, frequent sampling of water parameters (Prince 2004). In this study the author identified turbidity spikes to be an adequate measure of discoloured water formation in an unfiltered water distribution system.
Figure 5-1: Flow rate and turbidity spikes in particular days for two monitoring stations (MS1 & MS2) (Prince et al. 2001).

In particular, Prince (2004) found that COLT is an improvement over grab sampling methods. The basis for this conclusion is that the turbidity COLT time series data obtained from a sampling rate of
one per 10 minutes allowed typical turbidity readings to be distinguished from typical turbidity readings that constituted discoloured water events.

With the online monitoring data (Prince 2004) it is possible to pick up hundreds of turbidity spikes in a day. Figures 5-1 [a-c] represented water flow and turbidity as a function of time for two monitoring stations (MS1 & MS2). The distance between MS1 and MS2 was 330.4 m. The trends of the flow and turbidity profiles shown in Figures 5-1 [a-c] as well as the magnitudes and shapes were not equal at same time for each different day. There was no similarity in magnitude, shape, or even positioning of spikes between the flow and turbidity profiles.

Prince (2004) showed that there was no reason that all the spikes had to be same in shape or magnitude for same time everyday. Every spike did have different size and shape no matter what the water flow was. Therefore, the turbidity profiles of MS1 were found in many arbitrary shapes, which were completely different to what were found at MS2. Most of the spikes either disappear or stick around the surface of the pipe wall (Prince and McManus 1999; Prince et al. 2001).

With this analogy, six different inlet velocity and particle load profiles were assumed to describe the physical problem of distribution networks. The inlet velocity and the particle load profile represented the flow and turbidity profiles respectively that were observed in distribution networks.

5.3.2 Velocity and Particle-Load Profiles

To better understand of the hydrodynamics of turbidity spikes we need to investigate for the different profiles of the turbidity spikes and velocity profiles shown in Figures 5-2 [a-f].
In Figure 5-2a both the inlet velocity and particle load varied over a period of 2 hours followed by a steady state condition for another 30 minutes.

- The inlet velocity and particle load shown in Figure 5-2b varied over a period of 1 hour followed by a steady state condition for another 90 minutes.

- In Figure 5-2c both inlet velocity and particle load varied over time and the sinusoidal profile had two peaks within an hour followed by a steady state condition for another 90 minutes.

- In Figure 5-2d the inlet particle load varied over a period of an hour followed by a constants value for rest of the time, whereas, the inlet velocity increased at a constant rate over a
period of 2 and half hours followed by a steady state condition for another 30 minutes.

- In Figure 5-2e only the inlet particle load varied over a period of an hour followed by a steady state condition for rest of time but the velocity was kept constant at 0.1ms\(^{-1}\).
- In Figure 5-2f the inlet velocity and particle load varied arbitrarily over a period of an hour followed by a steady state condition for another 90 minutes.

The inlet profiles shown in Figure 5-2 [a-f] were introduced separately as subroutines to simulate the transient behavior of the particles. Different simple shapes and magnitudes were considered for the inlet velocity and particle load profiles to make them comparable to some of the spikes formed into the water distribution networks.

### 5.3.3 Geometry and Boundary Conditions

In order to study the hydrodynamics of particles behavior in a turbulent flow field, a geometry shown in Figure 5-3 comprising 330.4 m long and 472 mm diameter pipe was considered as seen at SEWL’s distribution system. The inlet turbid particle load and velocity profiles were introduced from monitoring station 1 (MS1) separately for each run.

![Figure 5-3: Schematic diagram of the horizontal straight pipe.](image)

Table 5-1 shows the boundary conditions that were implemented in the CFD investigation. A grid of 296000 cells was found to give results that were independent of the grid.
Table 5-1: Physical and hydraulic characteristics of the system used for CFD simulation.

<table>
<thead>
<tr>
<th>Characteristics</th>
<th>Numerical values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pipe length (m)</td>
<td>330.4</td>
</tr>
<tr>
<td>Diameter of the pipe D (m)</td>
<td>0.472</td>
</tr>
<tr>
<td>Total volume of water (m³)</td>
<td>0.322</td>
</tr>
<tr>
<td>VF of each secondary phase (ppm)</td>
<td>300</td>
</tr>
<tr>
<td>Particle density (kgm⁻³)</td>
<td>3000</td>
</tr>
<tr>
<td>Particles sizes (µm)</td>
<td>10</td>
</tr>
<tr>
<td>No. of computational cells</td>
<td>296540 produce grid</td>
</tr>
<tr>
<td></td>
<td>independent results</td>
</tr>
<tr>
<td>Velocity and Particle-load profiles</td>
<td>See Figures 5-2[a-f]</td>
</tr>
</tbody>
</table>

5.4 RESULTS AND DISCUSSION

The subroutines written for the inlet profiles shown in Figures 5-2[a-f] were implemented in the CFD simulations. The solution domain was initialized with converged results for 10 ppm particle loading steady flow. This is because drinking water was assumed not to be free from particles at any time. Figures 5-(4-15) show the particles distribution along the pipe averaged over the cross-section. For each case (profile) this average deposition was represented in three figures. First figure showed the average distribution of particle after 15, 30, 45, and 60 minutes along the length of the pipe. Second figure showed this after 60, 75, 90, 105, 120, 135, and 150 minutes. Finally, third figure showed the average volume fraction of particle as a function of time at different cross-sections (at 0, 80, 160, 240, and 330.4 meters) along the length of the pipe.

5.4.1 Average Particle Distribution along the Pipe

Figures 5-4 [a-b] showed the average particle volume fraction over the length of pipe for the profile 5-2a. With time, the peak particle distribution was moving towards the end of the pipe although the movement was slower than the average steady velocity of 0.1 ms⁻¹ as expected. This was because of gravitational force, which pulled the particles down to the bottom of the pipe where the velocity was low due to formation of boundary layer. Therefore, most of the
particles were either rolling or moving through the low velocity region, which resulted in slower movement of particles. The particle volume fraction increased marginally as it travelled along the pipe. This can be explained as the particles that introduced later travel faster than the earlier particles some of which were travelling or rolling at the lower velocity region near the bottom wall resulting in marginal accumulation. These happened more or less for all the profiles shown in Figures 5-(4-8).

Figure 5-4a: Average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2a.

Figure 5-4b: Average particle distribution along the pipe at different times (60-150 minutes) for the profile of 5-2a.

Figures 5-5 [a-b] showed the same average particle volume fraction over the length of pipe for the profile of 5-2b. These figures initially
showed the similar trend observed in Figures 5-4 [a-b]. However, in Figure 5-4b it was shown that after 105 minutes the profiles became distorted and particles accumulated to form secondary *kink*. The secondary kink was born from the parent peak deposition. The kink can be defined as a group of particles that accumulates and moves/rolls near the bottom region of the pipe wall.

Figure 5-5a: Average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2b.

Figures 5-6 [a-b] showed the same average particle volume fraction over the length of pipe for the profile 5-2c. The profiles (Figure 5-6b) showed marginal particle accumulation and formation of secondary kink (explained later).

Figure 5-5b: Average particle distribution along the pipe at different times (60-150 minutes) for the profile of 5-2b.
Figures 5-6a and 5-6b showed the average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2c. Figures 5-7 [a-b] showed again the average particle volume fraction over the length of pipe for the profile of 5-2d. This profile (5-2d) did not have any deceleration period. Therefore, the particle did not get enough time to accumulate such big cluster on the bottom wall of the pipe that would divide into two to form another kink. The continuous increase of turbulence resulted in no distortion or secondary kink formation within the length of pipe investigated.
Figure 5-7a: Average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2d.

Figure 5-7b: Average particle distribution along the pipe at different times (60-150 minutes) for the profile of 5-2d.

Figure 5-8 [a-b] show the same average particle volume fraction over the length of pipe for the profile shown in Figure 5-2e. In the Figure 5-8b the profiles showed distortion after a certain period of time and formed secondary kink. This was because the gravitational force dominated at constant velocity. This force pulled the particles down and allowed particles to form large group of particles, which then broke and formed the secondary kink.
Figure 5-8a: Average particle distribution along the pipe at different times (0-60 minutes) for the profile of 5-2e.

Figure 5-8b: Average particle distribution along the pipe at different times (60-150 minutes) for the profile of 5-2e.

Figure 5-9 [a-b] showed the same average particle volume fraction over the length of pipe for the profile of 5-2f. The velocity was suddenly dropped and kept constant at a value of 0.1 ms\(^{-1}\). However, the peak velocity was high compared to other profiles. This resulted less accumulation of particle due to high turbulence in fluid flow. Therefore, the deposited particles were not enough that could form secondary kink.
**5.4.2 Average Particle Deposition at Different Cross-sections**

Figures 5-(10-15) show particle distributions as a function of time at different cross-sectional planes of pipe. These 5 planes have been taken at 0 (inlet), 80, 160, 240, and 330.4 (outlet) meters. These figures provide an opportunity to investigate the hydrodynamic behavior included shapes and magnitudes of the particle load profiles (Figures 5-2[a-f]). Figures 5-(10-15) show that the sharp peak concentration was smoothened out as particles move further downstream of the pipe.
Immediately after the inlet the particle dispersed at the development region. The peak particle concentration became marginally lower before increasing again at the downstream location of the pipe (Figures 5-(10-15)). This was because of initial impact of turbulence at developing region, where particles may have distributed and resulted in lower volume fraction. After that region some of the particles slowed down or stopped, which resulted in the marginally higher concentration as particles moved further downstream of the pipe.

Figure 5-10: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 5-2a.

Figure 5-11: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 5-2b.
Figure 5-12: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 5-2c.

Figure 5-13: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 5-2d.

Figure 5-14: Average particle volume fraction as a function of time for different cross-section of pipe along length for the profile of 5-2e.
The gradient of the inlet velocity and particle load profile of 5-2b is higher than that of 5-2a resulting in more pronounced particles accumulation (see Figure 5-11). The 5-2e profile did not have any velocity gradient resulting in slower accumulation of particles (Figure 5-14). The profile 5-2d had only the positive velocity gradient and this resulted in less accumulation of particles at further downstream location (Figure 5-13) of the pipe. For the two successive spikes scenario (Figure 5-2c), the low concentration mid region appeared to get more particles due to turbulence mixing and hence it becomes smoothen (Figure 5-12). For the arbitrary profile 5-2f, the particle distribution appeared to be smoothened out quickly after the inlet (see Figure 5-15).

5.4.3 Gravity Deposition and Kink Formation

Figures 5-16 [a-c] represent the particle concentration at different depths along the pipe length for the profile of 5-2e. The profile 5-2e generated a clear secondary kink as shown in Figures 5-8b and 5-16c. Figure 5-16a shows the concentration of particle, which is less than that of bottom after the flow development.
Figure 5-16: Particle concentration at different depths along the pipe length at different time for the profile of 5-2e (a, b, and c for top, center, and bottom respectively).
Figure 5-16b shows the particle concentration at the center of the pipe. The bottom deposition shown in Figure 5-16c is very high as compared to top and center concentration. This is due to gravitational force that results in higher deposition at the bottom of the pipe wall. The scales for each figure are made different so that the volume fraction can be shown clearly. Figure 5-16c shows the formation of the secondary kink after certain period of time and length of pipe. It was becoming larger while travelling further downstream of the pipe.

The center line particle concentration showing in Figure 5-16b was decreasing due to gravitational deposition as the particle travelled further downstream of the pipe resulted in more deposition in bottom wall of the pipe. If the pipe was longer enough than the investigation domain then the most part of particles may have deposited on the bottom wall of the pipe resulting in more secondary kinks in bottom wall of the pipe.

*Kink Formation*

Downstream of the pipe the Figures 5-5b, 5-6b, and 5-8b show that small secondary kinks were extruded from parent-kink of their respective inlet profiles. The kink can be defined as the group of particles that accumulate to form a cluster/band of particles that are moving/rolling together.

The formation of kink can be explained by the increasing of particle accumulation with shear stress (Prince *et al.* 2003; Boxall *et al.* 2001; Hoven and Vreeburg 1992). This shear stress increases with the age of deposition (Mohauas, 1974) and the growth of the deposited layer of particles (Powel, 1982). After a certain length this shear stress is sufficiently high to cause excessive particle deposition on the bottom line of pipe wall (Figure 5-16c). Due to gravity the peak deposition was becoming larger. At a point the large kink broke up and formed two peaks. As shown in Figure 5-
a preceding or following kink was formed according to the deposition trend. After formation the kink was becoming larger again. The reason behind the kink formation helps to presume that if the flow domain was long enough then each and every profile would have more secondary kinks in their particle distribution.

The kink or secondary group of particle formation can also be explained by the nature of the lift force (Kallio and Reeks 1989). Kallio and Reeks (1989) reported that if a particle velocity is higher than the fluid \( (v_p > v_f) \), the lift force is negative and the particle moves “down” with the velocity gradient towards the wall resulted in more deposition on the bottom wall. In this study the profiles of 5-2 [b and c] had the faster deceleration period, where particles lead to the fluid motion because of higher inertia resulted in more deposition on the bottom wall. Furthermore, the profiles showing Figures 5-2 [a, c, and e] allowed particles a long time to deposit on the bottom wall resulted in secondary kinks. According to Kallio and Reeks (1989) an increasing density ratio will tend to reduce the effect of lift and so will lower the particle r.m.s. velocity, which leads more particles to accumulate, which may separate and form secondary kink.

A theoretical analysis of Yiantsios and Karabelas (Yiantsios and Karabelas 1995) suggested that a rolling mechanism is responsible for breaking up the parent larger peak deposition resulting in a secondary kink, which may be the case here.

5.5 CONCLUSION

In this chapter several unsteady flow fields have been simulated to explain the turbidity spikes movement that observed in the drinking water distribution networks such as Sydney Water, Melbourne Water, South East Water, etc. The arbitrary distributions of particle for every profile introduced in this chapter were different in shape and size due to multiplicity of turbidity spikes. Therefore, this study
provided an opportunity to understand the wider range of particle movement and the distortion phenomenon of spikes or kinks observed in the drinking water distribution networks. Results show the particle accumulated near the bottom wall of the pipe due to gravitational force. After a certain length of pipe and period of time this deposition became large and dispersed. The deposited particles then split and formed secondary kinks. Depending upon the velocity profile the kink was different in shape and size. If the flow domain was longer single spike might have more kinks by getting more particles into the system.
Chapter 6
Part 3: Turbulent Diffusion and Dispersion Model

6.1 INTRODUCTION

The quality of drinking water is highly dependent on the amount of particulate material present in the reticulation system. Little is understood of the origins, transport and fate of such material, yet it may cause up to 60 per cent of water quality related customer complaints to a water authority in developed countries like Australia (Prince et al. 2000). Therefore, the mechanisms of particle dispersion and deposition have to be better understood to identify the cause of discoloration of drinking water in water distribution networks. However, the ability to determine the origins of the particulate material varies between different water supply systems, with possible sources being from catchments, treatment processes, biofilm growth within the water supply pipes, corrosion products or other unidentified factors (Jayaratne et al. 2004).

Thus, particle dispersion and deposition in two-phase flows is very important and has been well recognized in numerous fields of research and industry. Some examples are the transport of pollutants in the atmosphere and oceans, droplets in sprays and internal combustion engines, slurries in pipes, sediment transport in coastal areas, catalyst particles in riser flows, particles deposition in annular dispersed two-phase flow, fluidized beds, dust deposition and dust removal in clean rooms, etc. In most applications one is
interested in how particles are transported by turbulent flows and what their fate is.

This chapter is concerned with one of these examples, namely a horizontal annular dispersed liquid/solid flow in a pipe. In these flows, the core of turbulent liquid contained spherical solid as particulates with diameters ranging from 5 to 200 µm. The solid particles were entrained from the core flow and deposited on the circumference of pipe wall. Figure 6-1 gives a sketch of this entrainment/deposition mechanism in a horizontal annular dispersed liquid-solid flow on a cross-section of the pipe. Fukano and Ousaka (1989), Laurinat et al. (1985), and Mols and Oliemans (1998) also proposed other mechanisms, in which liquid droplets were deposited in a gaseous medium. Until now it has not been clear which mechanism or which combination of mechanisms is/are responsible for the annular character of the circumference deposition.

Instead of immediately solving the basic liquid film equations for conservation of mass and momentum, where empirical correlations or simple models were needed for all the four mentioned mechanisms (Mols and Oliemans 1998; Laurinat et al. 1985; Fukano and Ousaka 1989), the deposition mechanism in a turbulent flow field was studied theoretically, separately from the other mechanisms. This is expected to lead to an expression for the deposition flux with a wider range of applicability than the correlations that have been used so far.
A 2D analytical turbulent diffusion model for particle dispersion and deposition at different heights across the pipe flow and circumferential deposition was re-developed. The liquid-solid turbulent diffusion model presented in this chapter has emanated from an existing gas-liquid turbulent diffusion model of Mols and Oliemans (1998). Simultaneously a comprehensive 3D numerical investigation was also carried out to compare with the results of the above analytical turbulent diffusion models making of multiphase mixture model available in a commercial software Fluent 6.1 (2001). In both studies different particles sizes and densities were used. The deposition was studied as a function of particle diameter, density and fluid velocity. The deposition of particle along the periphery of the wall and at different depth was also investigated. Both studies showed that the deposition of heavier particles at the bottom of the pipe wall was found to be higher at lower velocities and lower at higher velocities. The lighter particles were found mostly suspended with a homogeneous distribution. Smaller particles were also suspended with marginal higher concentration near the bottom of the wall. This marginal higher concentration of the smaller particles was found to be slightly more pronounced for lower velocities. The larger particles clearly showed deposition near the bottom of the wall. These analogies of particles were discussed with the ratio between free flight velocity and the gravitational settling velocity in later section of this chapter.
The motivation for this study was two-fold, first interested in the deposition of solid spherical particles with specific gravity of 3.0 typical of those found in annular dispersed two phase flows in water supply networks. The dispersed phases of these flows consisted of particles with diameters ranging from 5 to 100 µm. This study was also conducted for five different particle densities ranging from specific gravity (sg) 1.5 to 6.0 keeping the diameter constant at 10 µm. These sizes and density ranges were chosen in order to accurately compare with the drinking water distribution networks data (Grainger et al. 2003). These flows are rather complex and it is difficult to obtain detailed experimental data on the contribution to deposition in relation to particle size. The second point of interest of this study was to investigate the segregation of solid particles along the circumference of the pipe wall.

In order to obtain more insight into the process of particle dispersion and deposition 3D numerical CFD simulations were carried out at different Reynolds numbers. These analytical solutions were also validated with CFD investigation results using the same particle parameters and the help of Multiphase mixture model using Fluent 6.1.

6.2 ANALYTICAL TURBULENT DIFFUSION MODEL

When faced with the task of modeling turbulent particle deposition, or any multiphase flow, two general approaches are possible. One is Lagrangian approach, usually known as a “trajectory model” (Kallio and Reeks 1989), where the instantaneous motions of individual particles are tracked by solving their equations of motion. The trajectories of many particles (typically thousands) are realized in order to form the average behavior of the particle-fluid system. The other method is Eulerian, often called a “two-fluid” model, where the particles are treated as a continuous phase, in much the same way that a tracer fluid would be regarded in a binary mixture. The
motion of the particulate phase is mathematically described by mass, momentum and energy conservation, similar to a fluid. In this study the Eulerian approach was followed, which was more suitable for re-suspension and re-deposition of particles than the Lagrangian approach. In the Lagrangian approach the particles are tracked through the fluid domain and their effects on the fluid flow are introduced through drag forces. But the physical existence of the particles creating blockages or voidages is ignored because they are treated only as a point, with no volume. The physical existence of the particles is essential when there is a possibility of re-suspension and re-entrainment and/or re-deposition, which is the case in this problem. In order to study the behavior of particles in a turbulent flow field numerically, one needs a proper representation of turbulence itself.

The use of a cylindrical geometry in combination with a large range of particles (5 different sizes and 5 different mass particles) makes this study of relevance for many practical applications and makes a comparison with the numerous experiments on particle deposition possible.

The deposition flux of particles at a certain circumferential angle in the pipe was known from literature reviews (Fukano and Ousaka 1989; Laurinat et al. 1985). Except for very large particles (>200 μm), for which the motion is totally dominated by gravity and the particle’s initial entrainment velocity (Anderson and Russell 1970b; Anderson and Russell 1970a; James et al. 1987), there is at present no theoretical analysis of this deposition flux in a two-dimensional geometry. Anderson and Russell (1970a) developed a semi-empirical expression to correlate deposition and entrainment fluxes. The model used to derive this expression assumes that droplet deposition is caused by deterministic drop trajectories intersecting the liquid film. The work of James et al. (1987) is an extension of
the work of Anderson and Russell (1970b). In both approaches no effect of turbulence was taken into account because only very large particles were considered. Laurinat et al. (1985) proposed an empirical fit to a representative deposition flux profile measure by Anderson and Russell (1970a),

\[ R_D(\theta) = k_D \left[ 1 + 10 \exp\{2(\cos \theta - 1)\} \right] \tag{2-1} \]

where \( R_D \) is the deposition flux, \( k_D \) is a constant that has to be calculated from the entrainment of particles, and \( \theta \) is the angle around the pipe circumference. Figure 6-2 gives a sketch of this correlation (Equation [2-1]) in units of \( k_D \). The angle \( \theta = 0^\circ \) refers bottom and \( \theta = \pm 180^\circ \) is the top wall of the pipe shown as in Figure 6-3. Figure 6-3 represents the vertical cross-section of pipe. The position vector of each measuring point is \((R, \theta)\), where \( R \) is the radius of the pipe. Deposition is the highest at the bottom and the lowest at the top. In all state-of-the-art horizontal annular flow models a correlation like Equation [2-1] or even any more simple deposition flux independent of the circumferential pipe angle, \( \theta \), has been used.

![Figure 6-2: The deposition flux in a tube as a function of the circumferential angle in the pipe used by Laurinat (1985).](image)

The behavior of \( R_D(\theta) \) must depend on the fluid velocity. The difference between deposition at the bottom and top is expected to
decrease with increasing fluid velocity due to the increase of turbulence. In this study investigation was performed to look at more generalized form of this equation that was derived from a turbulent diffusion model (Mols and Oliemans 1998). This study was then compared with the CFD investigation results using the same boundary conditions and geometric domain.

Taylor (1921) introduced the concept of turbulent diffusion in a study of the spread of scalar properties like smoke, heat and soluble matter. Friendlander and Johnstone (1957) used this concept for modeling a two-phase flow with particles. They also introduced the “diffusion/free-flight” concept for particles depositing at a wall. In order to improve agreement with experimental data, different modifications of this concept were proposed in the course of time such as varying free-flight distance from the wall, modifying free-flight velocity, particle diffusivity unequal to eddy diffusivity, and changing concentration boundary condition at the free-flight distance (Kallio and Reeks 1989).

In the field of particle deposition, the most recent contribution is the work of Binder and Hanratty (1992), which is the starting point of this chapter. They considered the dispersion and deposition of particles in a two-dimensional horizontal rectangular channel by a convection/diffusion model. The diffusion part of this model represents the influence of turbulence and the convection part
represents the influence of gravity on the particles. Particles are emitted from an instantaneous point source at the bottom of the channel with some initial entrainment velocity and can deposit at either of the perfectly absorbing boundaries. The particle diffusivity and the particle deterministic fall velocity are taken to be functions of the time that a particle has been in the flow field. The resulting convection/diffusion equation and the equation for the time-dependent deterministic velocity of the particles are solved numerically. One of their conclusions was that two dimensionless groups determine the resulting concentration profiles, the ratio of the time scale of the particle to the time scale of the flow, $\tau_p/T_L$, and the Reynolds number based on the friction velocity, $Re^*$. $\tau_p$ is the particle relaxation time and $T_L$ is the integral flow time scale.

The main differences between the method used in this chapter and the approach of Binder and Hanratty (1992) are two-fold. First, the particle diffusion coefficient and the gravitational settling velocity were assumed to be stationary instead of time-dependent. This assumption has the great advantage that the one-dimensional problem can then be solved analytically, so that a general expression can be found for the deposition flux independent of the exact quantitative modeling of the particle diffusion coefficient and the gravitational settling velocity. It furthermore has the advantage that an analytical two-dimensional deposition flux in a pipe can be calculated containing the relevant physical parameters of the problem that are hidden in empirical correlations like Equation [2-1]. Of course it has the disadvantage of not taking into account the fact that the particle deterministic velocity is generally time-dependent and that the particle diffusion coefficient is also initially time-dependent. The second difference is that, the inertial and crossing trajectories effects in the particle diffusion coefficient are explicitly included. Thus the particle diffusion coefficient is equal to
the fluid diffusivity for $\tau_p/T_L<1$, but smaller than the fluid diffusivity for $\tau_p/T_L>1$ (Mols and Oliemans 1998). Binder and Hanratty (1992) assumed the particle diffusion coefficient to be equal to the fluid diffusivity. They did not consider at all the crossing trajectories effect.

The rest of this chapter is organized as follows. Firstly, a definition of the problem under consideration has been given specifying the assumptions on which the model was based, and the relevant length and time scales in the problem are introduced. Thereafter, a time-dependent, one-dimensional convection/diffusion problem were specified to solve the accompanying equation analytically and used the solution to calculate one and two-dimensional deposition fluxes of particles with fluid-solid flow instead of gas-solid flow (Mols and Oliemans 1998). The two-dimensional deposition flux would be compared with the CFD investigation that was simulated by using the Multiphase Mixture Model available in FLUENT 6.1. At the end of this chapter the most important conclusions of the analysis would be given based on the ratio between free-flight velocity and settling velocity.

**6.2.1 Definition of the turbulent diffusion problem**

Particles are dispersed by turbulence and convected in between two infinite horizontal plates by gravity and they can deposit at the walls. The problem is sketched in Figure 6-4. The streamwise turbulence is assumed to have little effect on the particle deposition at the walls, because the fluid mean velocity is dominant in the streamwise direction.
Figure 6-4: The diffusion/free-flight problem between two infinite horizontal plates.

It is assumed that particle can cross the boundary layers at the walls on their inertia. The time-dependent problem, with perfectly absorbing walls are studied and the two initial conditions were considered as such

- The initial conditions at which all the particles are homogeneously distributed on a pipe cross-section without having an initial radial velocity, and
- The initial condition of an instantaneous source at the bottom wall.

The aim of this model is to predict the relative concentration of particles and its dispersion from the turbulent diffusion model.

In the turbulent diffusion model the following assumptions are considered.

- Turbulent water flow in a horizontal pipe containing particles with particle/fluid density ratios ranges from 1.5 to 6.0.
- Dilute particle suspension (volume fraction $O(10^{-6})$: one way coupling) without break-up and coalescence.
- Uniform and axial average fluid velocity (plug flow).
- Homogeneous turbulence up to the boundary layers.
- The particle mean free path is larger than the thickness of the boundary layer (so that there is a free flight of particles through the boundary layer to the wall) and the particle mean free path is smaller than the pipe diameter.
- Fick’s law is valid, so that particles are in local equilibrium with the surrounding fluid and a diffusion equation can be applied. For homogeneous turbulence this condition implies that the particle relaxation time must be much greater than
the integral time scale of turbulence and much less than the particle diffusion time (Reeks 1983). The motion of the particles is then statistically similar to Brownian motion.

- Stationary Particle free fall velocity, \( v_g \), is considered.
- Particle diameters are ranging from 5 to 200 \( \mu \text{m} \), and
- The particle motion is not fully dominated by gravity.

The assumptions of homogeneous turbulence and uniform, axial fluid velocity are more or less justified by assumption (viii). High inertia particles effectively see almost homogeneous turbulence (they do not respond much to gradients in the fluid r.m.s. velocity normal to the wall), and they will not respond much to variations in the mean flow of the fluid. These assumptions will be discussed more elaborately later in this chapter.

The typical Eulerian eddy length scale is estimated to be 50 mm and the smallest scale of the turbulent structures (the Kolmogorov length scale, \( \lambda_K \), which is approximately \( O(10^{-4}) \)) is calculated according to

\[
\lambda_K \sim \left( \frac{v_f^3}{\varepsilon} \right)^{\frac{1}{4}}
\]  

[2-2]

where \( v_f \) is the kinematic viscosity of the fluid, and \( \varepsilon \) the kinetic energy dissipation, given by

\[
\varepsilon = k \cdot \frac{U^3}{L}
\]

[2-3]

Velocity scale \( U \) is related to the friction velocity \( u^* \) and is approximately equal to one tenth of the average fluid velocity. Length scale \( L \) is approximately one tenth of the diameter of the pipe, and for a pipe proportional constant \( k \approx 0.01 \). These have been estimated according to Mols and Oliemans (1998).

The integral time scale \( T_L \) of the fluid is given by
\[ T_L = \int_0^\infty \frac{\langle v'_r(t)v'_r(t_0) \rangle}{\langle v'^2 \rangle} dt \]  

[2-4]

where \( v'_r \) is the fluctuating velocity of the fluid and \( t_0 \) some initial time. In a pipe flow, \( T_L \) will depend on the spatial position, but as homogeneous turbulence is assumed, \( T_L \) is taken to be constant. \( T_L \) can be calculated approximately by

\[ T_L \approx \frac{L}{U} \]  

[2-5]

Large eddies are assumed to be dominant for the dispersion of particles with a time scale of \( T_L \). However, there is in fact a whole range of time scales in turbulence. The particle relaxation time based on Stokes drag is equal to

\[ \tau_p = \frac{1}{18} \frac{d_p^2}{v_f} \frac{(\rho_p - \rho_f)}{\rho_f} \]  

[2-6]

with \( v_f \) the kinematic viscosity of the fluid, and \( \rho_p \) and \( \rho_f \) the densities of, respectively, the particle and the fluid. For particle Reynolds numbers larger than one, the real relaxation time will be smaller due to the increased drag in the non-Stokes case. The ratio \( \tau_p/T_L \) is called the Stokes number \( S \) and can be interpreted as a measure of the influence of particle inertia on the dispersion of the particles by fluid turbulence.

Three distinguish cases are considered to lead to the different responses on the turbulent fluctuations, different behavior in the boundary layer, and different concentration gradients. These are (1) the particle relaxation time is much smaller \( (S \ll 1) \); (2) of the same order of magnitude \( (S \approx 1) \); or (3) much larger than the fluid integral time scale \( (S \gg 1) \). For particle relaxation times much smaller than the fluid integral time scale \( (S \ll 1) \), particles precisely follow the velocity fluctuations of the fluid, the deposition is delayed by the boundary layer and the deposition flux is low all around the pipe.
wall. For particle relaxation times of the same order of magnitude as the fluid integral time scale \( S \approx 1 \), particles follow the turbulent fluctuations quite well, the effect of the boundary layer on the deposition is limited, and the deposition flux due to turbulence is high all around the pipe wall. For particle relaxation times much larger than the fluid integral time scale \( S \gg 1 \), the response to the turbulent fluctuations is slow, and the particles effectively see a randomly fluctuating velocity field. The deposition is not affected by the boundary layer, and the deposition flux is larger at the bottom and small at the top, due to gravity. This is expected to hold for the larger particles.

Annular dispersed liquid-solid flows in industrial processes often operate under different conditions from laboratory water-particle flows. Pressures are often different along with the physical dimensions of the pipe. To some extent there is a dynamic similarity between a laboratory experiment and a field experiment. Therefore, it is necessary to consider the characteristic dimensionless parameters. Whereas, for an isothermal single-phase flow there is only one characteristic parameter, the Reynolds number, for two-phase flows there are five similarity parameters that determine a general isothermal two-phase flow problem (Chesters 1975); these are the Reynolds number, the Froude number, the Weber number, the density ratio, and the viscosity ratio. There is a dynamic similarity between two liquid-solid flows if these five similarity parameters are the same, and if there is a geometrical similarity of the imposed boundary conditions.

In this model solid spherical particles were assumed to be flowing through a horizontal pipe in turbulent field. To calculate Reynolds Number water velocity of 0.1 ms\(^{-1}\) and pipe diameter of \( 4.72 \times 10^{-1} \) m were used, but to calculate length scale \( L \), only the representative diameter of a particle, 100 \( \mu \)m was used.
6.2.2 Mathematical formulation of the model

The diffusion equation concept makes physically sensible when the relevant length scale over which the diffusion process is considered (here pipe diameter, $D$) is larger than the particle mean free path (defined in homogeneous turbulence), and the time of observation is larger than the mean free time ($T_L$). Following Swailes and Reeks (1994) a particle mean free path can be defined in a turbulent flow as the distance traveled by a particle in a time over which its motion is correlated. The particle mean free path $l$ is then defined as

$$ l = \sqrt{\langle v_p^2 \rangle T_p} \quad [2-7] $$

with $\langle v_p^2 \rangle$ is the particle mean square velocity and $T_p = T_L(1+S)$ the particle integral time scale. With increasing particle relaxation time, the particle r.m.s. velocity decreases, but the correlation time of the particle velocity increases more than the particle r.m.s. velocity decreases. Figures 6-5a and 6-5b give the ratio between pipe diameter and mean free path, $D/l$, at two different velocities.

Figure 6-5a shows that for the largest particles that have been used in this calculation, 200 $\mu$m, the ratio $D/l$ is still larger than one, which supports assumption (v) in turbulent diffusion problem. In Figure 6-5b the heaviest particles that have been considered, 6000 kgm$^{-3}$, still give $D/l$ well above one, which also supports assumption (v). The mean free path for very small particle are assumed to be determined by the Eulerian integral length scale, which is 0.11 of the pipe diameter. $D/l$ is then more than 27. However, in the limit of very large particles, the mean free path goes to infinity, and $D/l$ goes to 0. The particle mean free path increases with increasing velocities.
The gradient diffusion model in inhomogeneous turbulence is not valid unless \( \tau_p^* = \tau_p \left( \frac{u'}{\langle u' \rangle} \right)^2 / \nu_t \ll 3 \) (Reeks 1983). For the relaxation time larger than this limit, turbophoresis becomes important. Turbophoresis is the effect that particles migrate in inhomogeneous turbulence from a region of high turbulent velocity fluctuations to a region of low velocity fluctuations (Reeks 1983). In the present model, however, it has been neglected, because homogeneous turbulence is assumed.

![Figure 6-5a](image1)

**Figure 6-5a:** Ratio between pipe diameter and particle mean free path as a function of particle diameter at 0.1 and 0.5 m/s fluid velocity \((D = 4.3 \times 10^{-1} \text{ m})\).

![Figure 6-5b](image2)

**Figure 6-5b:** Ratio between pipe diameter and particle mean free path as a function of particle density at 0.1 and 0.5 m/s fluid velocity \((D = 4.72 \times 10^{-1} \text{ m})\).

According to the definition of the physical problem and considering the relevant length and time scales, a mathematical formulation of
the problem can be derived. A time-dependent convection/diffusion equation in one spatial dimension is generally written as

\[ \frac{\partial C(y, t)}{\partial t} = D_p \frac{\partial^2 C(y, t)}{\partial y^2} + V_g \frac{\partial C(y, t)}{\partial t} \] \[ \text{[2-8]} \]

where \( C(y, t) \) is the particle concentration as a function of the spatial position \( y \) (measured from bottom of the pipe wall) and time \( t \). \( D_p \) is the particle diffusion coefficient and \( V_g = g \tau_p \) the gravitational settling velocity of the particle. The first term on the right hand side is the diffusive term due to the influence of turbulence on the particles, the second term is the convective term due to the influence of gravity on the particles. Following Binder and Hanratty (1992) the variables are made dimensionless according to

\[ y^* \rightarrow \frac{y}{D}; \quad t^* \rightarrow \frac{tu^*}{D}; \quad D_p^* \rightarrow D_p \frac{D}{u^* D}; \quad V^* \rightarrow \frac{V}{u^*}; \quad C^* \rightarrow \frac{C u^*}{R_e} \] \[ \text{[2-9]} \]

where \( R_e \) is the entrainment flux of the particles and \( u^* \) is the friction velocity. The friction velocity is calculated using the Blasius correlation for a smooth pipe

\[ C_f = 0.0791 \cdot Re_f^{-0.25} \] \[ \text{[2-10]} \]

where \( C_f \) is the friction coefficient and \( Re_f \) is the fluid Reynolds number, which is equal to \( V_f D / \nu_f \). From the friction coefficient the wall shear stress \( \tau_s \) is calculated as

\[ \tau_s = \frac{1}{2} \rho_f V_f^2 C_f \] \[ \text{[2-11]} \]

where \( V_f \) is the pipe average fluid velocity. The wall shear stress is related to the friction velocity \( u^* \) by

\[ u^* = \sqrt{\frac{\tau_s}{\rho_f}} \] \[ \text{[2-12]} \]
The one-dimensional convection/diffusion equation for the concentration of particles $C^+$ can then be written in the dimensionless form

$$\frac{1}{D_p} \frac{\partial C^+}{\partial t^+} = \frac{\partial^2 C^+}{\partial (y^+)^2} + P \frac{\partial C^+}{\partial y^+} \tag{2-13}$$

where Peclet number $P$ is defined by

$$P: = g \frac{\tau_p \cdot \rho}{D_p} \cdot D \tag{2-14}$$

The Peclet number is the ratio of the convection term due to gravitational settling and the diffusion term due to turbulent diffusion. For Peclet number much smaller than one the diffusion term dominates. Whereas, for Peclet numbers much larger than one, the gravitational settling becomes dominant.

The particle diffusion coefficient $D_p$ is related to the fluid diffusivity $D_f$ by

$$D_p = \gamma_{\text{inert}} \gamma_{\text{cross}} D_f \tag{2-15}$$

where the fluid diffusivity is given by

$$D_f = \int_0^\infty \langle v'_r(0)v'_r(t) \rangle dt = \langle v'^2 \rangle \int e^{-\frac{t}{\tau_f}} dt = \langle v'^2 \rangle T_L \tag{2-16}$$

(Taylor 1921). The fluid mean square velocity $\langle v'^2 \rangle$ can be approximated by $(0.7 \times u^*)^2$ in the part of the pipe where turbulence is considered to be homogeneous (Mols and Oliemans 1998). In the absence of gravity, the ratio of particle and fluid diffusivity is governed by the ratio of particle relaxation time and fluid integral time scale and the inertial coefficient $\gamma_{\text{inert}}$, which can then be estimated from Mols and Oliemans (1998) as

$$\gamma_{\text{inert}} = \frac{1}{\sqrt{1 + \tau_p / T_L}} \tag{2-17}$$
Equation [2-17] corresponds to a decreasing response of particles to fluid turbulence (in a wall bounded flow) if $\tau_p > T_L$. In the presence of a gravity field, the effect of crossing trajectories generally has to be taken into account according to Csanady (1963). A particle falling through an eddy loses its velocity correlation more rapidly than a fluid element. Thus it sees a fluctuating velocity field that varies more rapidly in time than a fluid element. The velocity correlation of a fluid element is determined only by the decay of an eddy. The result is that the crossing trajectories effect leads to a decreased particle diffusivity. It is determined by the ratio between the fluid integral time scale $T_L$ and the time spent by a particle within an eddy, $L/\nu_g$ with $L$ the Eulerian eddy length scale and $\nu_g = g \tau_p$ the gravitational settling velocity. The crossing trajectories coefficient $\gamma_{\text{cross}}$ is then given by

$$
\gamma_{\text{cross}} = \frac{1}{\sqrt{1 + \left(\frac{g \tau_p T_L}{L}\right)^2}} \quad [2-18]
$$

Figures 6-6a and 6-6b represent both inertial and crossing trajectories coefficients at velocity of 0.1 ms$^{-1}$ and 0.5 ms$^{-1}$. Figure 6-6a shows that for 50 $\mu$m particles the crossing trajectories effect can reduce particle diffusivity by about 8%, but for larger particles 100 $\mu$m this has been reduced by about 35% for an average fluid velocity of 0.1 ms$^{-1}$. This effect is negligible for 50 $\mu$m particles and less than 5% for 100 $\mu$m particles at velocity 0.5 ms$^{-1}$. The crossing trajectories effect is much more important for the particle sizes larger than 50 $\mu$m. However, Figure 6-6b shows that the crossing trajectory coefficients for heavier particles change somewhat at higher velocity (0.5 ms$^{-1}$), where gravitational force may influence the settling at particles, whereas, the inertial trajectory coefficients are considered to be independent of velocity and particle density.
Since the particle’s mean free path is larger than the thickness of the boundary layer, a convection/diffusion equation is no longer valid for the particle behavior in the boundary layer. It is assumed that particles are projected towards the wall at the beginning of the boundary layer, leading to a free-flight flux \( v \cdot C \), where, \( v \) has the dimensions of a velocity and concentration \( C \) is dimensionless. The free-flight flux is then approximated by

\[
v \cdot C = \int_0^\infty C \cdot v p(v) dv
\]

[2-19]

Figure 6-6a: The inertial and crossing trajectories coefficients as a function of particle diameter for the velocity 0.1 and 0.5 ms\(^{-1}\).

Figure 6-6b: The inertial and crossing trajectories coefficients as a function of particle specific gravity for the velocity 0.1 and 0.5 ms\(^{-1}\).
where $p(v)$ is the velocity distribution at the point from which the particles are projected in the direction of the wall. Integration is performed only over the particle velocities directed towards the wall. Following Binder and Hanratty (1992), $p(v)$ is assumed to be Gaussian. With this assumption Equation [2-19] follows that

$$v \cdot C = \frac{1}{2} \sqrt{\frac{2}{\pi}} \sqrt{\langle v^2_p \rangle} \cdot C$$

[2-20]

so that the free-flight velocity $v$ is equal to

$$v = \frac{1}{2} \sqrt{\frac{2}{\pi}} \sqrt{\langle v^2_p \rangle}$$

[2-21]

The particle mean square velocity is calculated from the ratio of velocity fluctuations between particle and fluid given in Hinze (1975) in which the large density ratio between particle and fluid leads to

$$\frac{\langle v^2_p \rangle}{\langle v^2_f \rangle} = \frac{1}{1+S}$$

[2-22]

In the effect of crossing trajectories analysis shown in Figures 6-6a and 6-6b is fairly high especially at lower velocity for particles under consideration. Equation [2-22] is also used by Swailes and Reeks (1994). At the beginning of the boundary layers $\sqrt{\langle v^2_f \rangle} = 0.9 \cdot u'$. Then it follows from Equations [2-21] and [2-22] that

$$v = \frac{1}{2} \sqrt{\frac{2}{\pi}} \sqrt{\langle v^2_f \rangle} \frac{1}{1+S}$$

[2-23]

Binder and Hanratty (1992) used the following empirical approximation:

$$\frac{\langle v^2_p \rangle}{\langle v^2_f \rangle} = \frac{1}{1 + (0.7 \tau_p / T_L)}$$

[2-24]

For Stokes relaxation time, this expression reduces to
\[
\frac{\langle v_p'^2 \rangle}{\langle v_r'^2 \rangle} = \frac{1}{1 + 0.7 \cdot S}
\]

having the same form as Equation [2-22].

Now the boundary conditions can be derived by applying conservation of mass at the beginning of the boundary layer. The diffusive plus the gravitational flux towards the boundary layer \((D_p(\partial C)/(\partial y) + (v_g C))\) must be equal to the free-flight plus the gravitational flux from the boundary layer to the wall \((v C + v_g C)\). Assuming that the boundary layer is enough to apply this condition exactly on the wall and the diffusion free-flight boundary condition (Binder and Hanratty 1992) at the bottom wall \((y^+ = 0)\):

\[
D_p^+ \frac{\partial C^+}{\partial y^+} = v^+ C^+ \quad [2-26]
\]

At the top wall \((y^+ = y/D = 1)\), the

\[
-D_p^+ \frac{\partial C^+}{\partial y^+} = v^+ C^+ \quad [2-27]
\]

A more theoretical discussion and justification of this type of boundary condition is given in Morse and Feshbach (1953). The ratio \(D_p/v\) is of the order of the particle mean free path. Because the particle’s mean free path is large compared to the length scale characterizing the variation in the particle concentration, \(\partial C/\partial y\), there can be a finite concentration at the wall. For particles with \(S \ll 1\) the particle mean free path vanishes at the wall, leading to the boundary condition, which normally represents perfect absorption, \(C = 0\). To close the one-dimensional model, two initial conditions have been considered. The first initial condition is a uniform concentration:

\[
C^+(t^+ = 0) = C_0 = 1 \quad [2-28]
\]

The second initial condition is a delta source at the bottom wall:
\[ C^+(t^+ = 0) = \delta(y^+) \]  \[2-29\]

Other initial conditions can be used without difficulty. This initial source at the bottom is the initial condition that was considered by Binder and Hanratty (1992).

The mathematical formulation of the problem now consists of Equation [2-13] with boundary conditions (Equations [2-26] and [2-27]) and initial conditions (Equation [2-28] or [2-29]).

6.3 ANALYTICAL SOLUTION OF THE PROBLEM

6.3.1 One Dimensional Problem

Equation [2-13] with boundary conditions (Equations [2-26] and [2-27]) and initial conditions (Equation [2-28] or [2-29]) can now be solved analytically by the separation of variables, leading to a series solution for \( C^+(y^+,t^+) \):

\[
C^+(y^+,t^+) = \exp\left(-\frac{\gamma}{2} Py^+ \right) \sum_{n=0}^{\infty} \gamma_n \left[ \cos(b_n y^+) \right] + \\
\beta_n \sin(b_n y^+) \exp\left( -k_n^2 D_p t^+ \right) \tag{3-1}
\]

The eigenvalues \( k_n \) are determined by the boundary conditions. The eigenvalues \( k_n \) can be defined in terms of \( b_n \):

\[
k_n^2 = b_n^2 + \left( \frac{1}{2} P \right)^2 \tag{3-2}
\]

where \( b_n \) then satisfies the transcendental equation

\[
\tan b_n = \frac{2\lambda b_n}{(a^2 - \lambda^2) + b_n^2} \tag{3-4}
\]

\( \lambda \) is the dimensionless free-flight/diffusion ratio equal to \( vD/D_p \). \( \gamma_n \) is used to satisfy the initial condition and is given by

\[
\gamma_n = \frac{\int_0^1 C^+(t = 0) \exp(ay^+) \left( \cos b_n y^+ + \beta_n \sin b_n y^+ \right) dy^+}{\int_0^1 \exp(2ay^+) \left( \cos b_n y^+ + \beta_n \sin b_n y^+ \right) dy^+} \tag{3-4}
\]
\( \gamma_n \) can be solved analytically. For \( C^+(t=0) = d(y^+) \) the numerator in Equation [3-4] is equal to 1. Furthermore, it can be defined

\[
a = -1/2P \quad [3-5]
\]

and

\[
\beta_n = \frac{\nu D}{D_p} + \frac{1}{2} P \quad [3-6]
\]

Tables 6-1 and 6-2 give the properties of particles that are used in the Turbulent Diffusion Model. The first values given in each row are at \( V_f = 0.1 \text{ ms}^{-1} \), the second values are given at \( V_f = 0.5 \text{ ms}^{-1} \). Table 6-3 gives the properties of the fluid that were used in this model \( \text{Re}_f^* = u^*D/\nu_f \) and \( \text{Fr}_f^* = (u^*)^2/gD \).

Table 6-1: Properties of the different size particles with specific gravity 3.0 that are used in the Turbulent Diffusion Model.

<table>
<thead>
<tr>
<th>( d_p (\mu m) )</th>
<th>( \tau_p )</th>
<th>( S )</th>
<th>( D/l )</th>
<th>( \nu_0 (\text{m/s}) )</th>
<th>( v (\text{m/s}) )</th>
<th>( \gamma_{inert} )</th>
<th>( \gamma_{cross} )</th>
<th>( D_p (\text{m}^2/\text{s}) )</th>
<th>( P )</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>2.78\times10^{-6}</td>
<td>5.89\times10^{-7}</td>
<td>27.6</td>
<td>2.73\times10^{-5}</td>
<td>1.86\times10^{-3}</td>
<td>1.0</td>
<td>1.00</td>
<td>6.21\times10^{-5}</td>
<td>0.2</td>
</tr>
<tr>
<td></td>
<td>2.94\times10^{-6}</td>
<td>33.7</td>
<td></td>
<td>7.61\times10^{-3}</td>
<td>1.00</td>
<td>2.08\times10^{-4}</td>
<td>0.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>1.11\times10^{-5}</td>
<td>2.35\times10^{-6}</td>
<td>27.6</td>
<td>1.09\times10^{-4}</td>
<td>1.86\times10^{-3}</td>
<td>1.0</td>
<td>1.00</td>
<td>6.21\times10^{-5}</td>
<td>0.8</td>
</tr>
<tr>
<td></td>
<td>1.18\times10^{-5}</td>
<td>33.7</td>
<td></td>
<td>7.61\times10^{-3}</td>
<td>1.00</td>
<td>2.08\times10^{-4}</td>
<td>0.2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>4.44\times10^{-5}</td>
<td>9.42\times10^{-6}</td>
<td>27.6</td>
<td>4.36\times10^{-4}</td>
<td>1.86\times10^{-3}</td>
<td>1.0</td>
<td>0.999</td>
<td>6.20\times10^{-5}</td>
<td>3.3</td>
</tr>
<tr>
<td></td>
<td>4.71\times10^{-5}</td>
<td>33.7</td>
<td></td>
<td>7.61\times10^{-3}</td>
<td>1.00</td>
<td>2.07\times10^{-4}</td>
<td>1.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>50</td>
<td>2.78\times10^{-4}</td>
<td>5.89\times10^{-5}</td>
<td>27.6</td>
<td>2.73\times10^{-3}</td>
<td>1.86\times10^{-3}</td>
<td>1.0</td>
<td>0.965</td>
<td>5.99\times10^{-5}</td>
<td>21.5</td>
</tr>
<tr>
<td></td>
<td>2.94\times10^{-4}</td>
<td>33.7</td>
<td></td>
<td>7.60\times10^{-3}</td>
<td>1.00</td>
<td>2.07\times10^{-4}</td>
<td>6.2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>100</td>
<td>1.11\times10^{-3}</td>
<td>2.35\times10^{-4}</td>
<td>27.6</td>
<td>1.09\times10^{-2}</td>
<td>1.86\times10^{-3}</td>
<td>1.0</td>
<td>0.676</td>
<td>4.19\times10^{-5}</td>
<td>122.6</td>
</tr>
<tr>
<td></td>
<td>1.18\times10^{-3}</td>
<td>33.7</td>
<td></td>
<td>7.60\times10^{-3}</td>
<td>1.00</td>
<td>2.03\times10^{-4}</td>
<td>25.4</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 6-2: Properties of the different weighted particles with diameter 10 \( \mu \text{m} \) that are used in the Turbulent Diffusion Model.

<table>
<thead>
<tr>
<th>( sg )</th>
<th>( \tau_p )</th>
<th>( S )</th>
<th>( D/l )</th>
<th>( \nu_0 (\text{m/s}) )</th>
<th>( v (\text{m/s}) )</th>
<th>( \gamma_{inert} )</th>
<th>( \gamma_{cross} )</th>
<th>( D_p (\text{m}^2/\text{s}) )</th>
<th>( P )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5</td>
<td>2.78\times10^{-6}</td>
<td>5.89\times10^{-7}</td>
<td>27.6</td>
<td>2.73\times10^{-5}</td>
<td>1.86\times10^{-3}</td>
<td>1.0</td>
<td>1.00</td>
<td>6.21\times10^{-5}</td>
<td>0.2</td>
</tr>
<tr>
<td></td>
<td>2.94\times10^{-6}</td>
<td>33.7</td>
<td></td>
<td>7.61\times10^{-3}</td>
<td>1.00</td>
<td>2.08\times10^{-4}</td>
<td>0.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.5</td>
<td>8.33\times10^{-6}</td>
<td>1.77\times10^{-6}</td>
<td>27.6</td>
<td>8.18\times10^{-5}</td>
<td>1.86\times10^{-3}</td>
<td>1.0</td>
<td>1.00</td>
<td>6.21\times10^{-5}</td>
<td>0.6</td>
</tr>
<tr>
<td></td>
<td>8.83\times10^{-6}</td>
<td>33.7</td>
<td></td>
<td>7.61\times10^{-3}</td>
<td>1.00</td>
<td>2.08\times10^{-4}</td>
<td>0.2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.0</td>
<td>1.67\times10^{-5}</td>
<td>3.53\times10^{-6}</td>
<td>27.6</td>
<td>1.64\times10^{-4}</td>
<td>1.86\times10^{-3}</td>
<td>1.0</td>
<td>1.00</td>
<td>6.20\times10^{-5}</td>
<td>1.2</td>
</tr>
<tr>
<td></td>
<td>1.77\times10^{-5}</td>
<td>33.7</td>
<td></td>
<td>7.61\times10^{-3}</td>
<td>1.00</td>
<td>2.08\times10^{-4}</td>
<td>0.4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.0</td>
<td>2.22\times10^{-5}</td>
<td>4.71\times10^{-6}</td>
<td>27.6</td>
<td>2.18\times10^{-4}</td>
<td>1.86\times10^{-3}</td>
<td>1.0</td>
<td>1.00</td>
<td>6.20\times10^{-5}</td>
<td>1.7</td>
</tr>
</tbody>
</table>
Table 6-3: Properties of the fluid that are used in the Turbulent Diffusion Model.

<table>
<thead>
<tr>
<th>$V_f$ (ms$^{-1}$)</th>
<th>Re$_f$</th>
<th>Re$_f^*$</th>
<th>Fr$_*$</th>
<th>T$_i$ (s)</th>
<th>$u^*$ (ms$^{-1}$)</th>
<th>$v$ (ms$^{-1}$)</th>
<th>$D_f$ (m$^2$/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>47200</td>
<td>2445</td>
<td>5.79 x 10$^{-6}$</td>
<td>4.72</td>
<td>0.0052</td>
<td>0.00186</td>
<td>6.2 x 10$^{-5}$</td>
</tr>
<tr>
<td>0.5</td>
<td>236000</td>
<td>9997</td>
<td>9.69 x 10$^{-5}$</td>
<td>0.944</td>
<td>0.0212</td>
<td>0.00761</td>
<td>2.1 x 10$^{-4}$</td>
</tr>
</tbody>
</table>

The solution $C^+(y^+, t^+)$ given by Equation [3-1] depends on three physical parameters, the Peclet number $P$, the dimensionless free-flight/diffusion ratio $vD/D_p$ (determining the boundary conditions), and the initial condition $C^+(t = 0) = C_0$.

The deposition flux, $R_D$, given by the sum (for the bottom wall) or the difference (for the top wall) of the free-flight flux and the gravitational settling flux:

$$R_D = (v \pm v_g)C$$  [3-7]

For $v_g > v$, the particle concentration at the top is equal to 0. The ratio $D_{rel}(t^+)$ of deposition fluxes at the top and at the bottom then follows directly from the solution for the concentration (Mols and Oliemans 1998) and Equation [3-7]:

$$D_{rel}(t^+) = \frac{v \pm v_g}{v + v_g} \exp\left[-\frac{1}{2}P\right] \sum_{n=0}^{\infty} \gamma_n \frac{\cos(b_n) + \beta_n \sin(b_n)}{\exp(-k_n^2D_p^t)}$$  [3-8]

Particles can deposit at the top wall only when free-flight velocities are much larger than the gravitational settling velocity. For as time approaches to infinity (i.e. $t \to \infty$), the initial condition does not affect the solution anymore, then

$$D_{rel} = \frac{|v - v_g|}{v + v_g} \exp\left[-\frac{1}{2}P\right]$$  [3-9]
Table 6-4 shows the velocity ratio (ratio of gravitational settling velocity, $v_g$, to free-flight velocity, $v$) for different particles at velocities of 0.1 and 0.5 ms$^{-1}$. When $v_g > 0.25v$ the particles will settle at the bottom wall under the influence of gravitational force; whereas, if $v_g < 0.25v$, the particles will remain suspended in a mostly with homogenous distribution across the pipe due to turbulence diffusion. Table 6-4 shows that the particles ranges from 5 to 20 µm have the gravitational settling velocities, $v_g$, less than one fourth of the free flight velocities ($v_g < 0.25v$) so these particles can be found at the top of the pipe (Figures 6-7[a, b] and 6-8[a, b]).

Table 6-4: Gravitational settling velocity, $v_g$ and its ratio to free-flight velocity, $v$.

<table>
<thead>
<tr>
<th>Particle Diameter $d_p$ (mm)</th>
<th>Specific Gravity $s_g$</th>
<th>Settling Velocity $v_g$ (ms$^{-1}$)</th>
<th>Velocity Ratio $v_g/v$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>0.1 ms$^{-1}$</td>
<td>0.5 ms$^{-1}$</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td>4.09×10^{-05}</td>
<td>0.02</td>
</tr>
<tr>
<td>10</td>
<td></td>
<td>1.64×10^{-04}</td>
<td>0.09</td>
</tr>
<tr>
<td>20</td>
<td>3.0</td>
<td>6.54×10^{-04}</td>
<td>0.35</td>
</tr>
<tr>
<td>50</td>
<td></td>
<td>4.09×10^{-03}</td>
<td>2.21</td>
</tr>
<tr>
<td>100</td>
<td>1.5</td>
<td>1.64×10^{-02}</td>
<td>8.79</td>
</tr>
<tr>
<td>10</td>
<td>2.5</td>
<td>8.18×10^{-05}</td>
<td>0.04</td>
</tr>
<tr>
<td>2.5</td>
<td></td>
<td>1.36×10^{-04}</td>
<td>0.07</td>
</tr>
<tr>
<td>4.0</td>
<td></td>
<td>2.18×10^{-04}</td>
<td>0.12</td>
</tr>
<tr>
<td>5.0</td>
<td></td>
<td>2.73×10^{-04}</td>
<td>0.15</td>
</tr>
<tr>
<td>6.0</td>
<td></td>
<td>3.27×10^{-04}</td>
<td>0.20</td>
</tr>
</tbody>
</table>

Figures 6-7a and 6-8a show the relative concentration plotted as a function of particle diameter for different heights of 0.25D, 0.5D, 0.75D, and 1D from the bottom wall of the pipe. Relative concentration is a dimensionless parameter, which represents the ratio between local particle concentration at any heights and that of at bottom of the pipe wall.
Figure 6-7a: Relative concentration of particles for different height as a function of particle diameter for the velocity 0.1 ms\(^{-1}\).

Figure 6-7b: Relative concentration of particles for different height as a function of specific gravity for the particle of 10 µm at velocity 0.1 ms\(^{-1}\).
Figure 6-8a: Relative concentration of particles for different height as a function of particle diameter for the velocity 0.5 ms\(^{-1}\).

![Graph showing relative concentration for different heights as a function of particle diameter.]

Figure 6-8b: Relative concentration of particles for different height as a function of specific gravity for the particle of 10 µm at velocity 0.5 ms\(^{-1}\).

Particles larger than 25 µm were not able to reach at top wall for the velocity 0.1 ms\(^{-1}\) but for 0.5 ms\(^{-1}\) (Figure 6-8a) particles larger than 50 µm did not reach to top wall. For both cases particles exhibited that \(v_g > 0.25v\) (Table 6-4). Therefore, the particle settled at the bottom by the influence of gravitational force.

In Figures 6-7b and 6-8b the relative concentration is plotted as a function of particle density, where most of the particles remain suspended due to \(v_g < 0.25v\) (Table 6-4).

In Figures 6-9a and 6-9b the relative concentration is plotted at fluid velocities of 0.1 ms\(^{-1}\) and 0.5 ms\(^{-1}\) for the pipe diameter \(D = 4.72 \times 10^{-1}\) m (top wall). Figure 6-9a shows that the curve for velocity 0.5 ms\(^{-1}\) lied much above the one for velocity 0.1 ms\(^{-1}\) in the region 5 µm < \(d_p\) < 50 µm. In Figure 6-9b this occurs for all ranges of particles. This is due to two effects, which compensate each other to a certain extent. For a higher velocity the correlation time \(T_L\) decreases (Table 6-3), leading to a larger Stokes number and a decreasing particle diffusivity as a result of inertial effects, which are very minor and are shown in Figures 6-6a and 6-6b. On
the other hand, the particle diffusivity increases due to the increasing fluid diffusivity. The latter increases more than the

![Graph showing relative concentration vs. particle diameter](image1)

**Figure 6-9a**: Comparison of the relative concentration of particles between top and bottom of the pipe as a function of the particle diameter at 0.1 and 0.5 ms\(^{-1}\).

![Graph showing relative concentration vs. specific gravity](image2)

**Figure 6-9a**: Comparison of the relative concentration of particles between top and bottom of the pipe as a function of the particle density at 0.1 and 0.5 ms\(^{-1}\).

former decreases, so that according to Equation [2-18], the nett particle diffusivity increases slightly at higher velocity. Relative concentration for the velocity 0.1 ms\(^{-1}\) drops to zero for particles 50 µm and larger, whereas, it happens for the particle size larger than 55 µm when the fluid velocity is 0.5 ms\(^{-1}\).
6.3.2 Extension to a two-dimensional deposition flux comparison with a semi empirical correlation

Paras and Karabelas (Paras and Karabelas 1991) found in their experiment with a horizontal annular dispersed gas-liquid flow that the particle distribution was more or less constant. Using this experimental result as an extra assumption, Mols and Oliemans (1998) proposed a one-dimensional model, which can be extended to a quasi two-dimensional model by substituting

\[
\gamma^+ = \frac{Y}{D} \to \frac{1}{D} \left( \frac{1}{2} D (1 - \cos \theta) \right) = \frac{1}{2} (1 - \cos \theta)
\]

in \( R^+_D(\theta, t^+) = (v^+ + v^+_g) C^+(y^+, t^+) \), where \( C^+(y^+, t^+) \) is given in Equation [3-1]. \( D \) is the diameter of the pipe. Furthermore, by substituting \( v^+_g \to v^+_g \cos \theta \) in the expression for the deposition flux, it leads to

\[
R^+_D(\theta, t^+) = (v^+ + v^+_g \cos \theta) \exp \left[ \frac{1}{2} P(\cos \theta - 1) \right] \cdot \sum_{n=0}^{\infty} \gamma_n \left[ \cos \left( \frac{1}{2} b_n (1 - \cos \theta) \right) + \beta_n \sin \left( \frac{1}{2} b_n (1 - \cos \theta) \right) \right] \exp \left( k_n^2 D_p^+ t^+ \right) \tag{3-10}
\]

Since the turbulence is assumed to be homogeneous, the free-flight velocity is independent of \( \theta \). The Equation [3-10] is not of a genuine physical origin for the two-dimensional case. Therefore, the results of Equation [3-10] lead to a local minimum at \( \theta = 0 \) and local maxima at \( \theta \) values slightly larger than 0, which is an artificial effect. As the series term in Equation [3-10] is a term depending on the initial entrainment condition (via \( \gamma_n \)), Mols and Oliemans (1998) proposed to write it for the stationary case as some unknown constant \( C_E \), to be determined by the initial entrainment condition. \( C_E \) will generally differ for different particle relaxation times. The final result for the two-dimensional deposition flux that follows from their analysis can then be written as
\[ R_D(\theta, \tau_p) = k_D(\theta, \tau_p) \cdot \exp\left[\frac{1}{4} P(\cos \theta - 1)\right] \]  

[3-11]

where the local deposition constant \( k_D(\theta, \tau_p) \) is defined as

\[ k_D(\theta, \tau_p) = C_E(\tau_p) \cdot (v + v_g \cos \theta) \]  

[3-12]

having the dimension of velocity. The constant \( C_E \) can be determined from the fact that in a fully developed annular liquid-solid flow the total entrainment flux equals the total deposition flux (Mols and Oliemans 1998):

\[ \int_0^\pi R_E(\theta, \tau_p) d\theta = \int_0^\pi R_D(\theta, \tau_p) d\theta \]  

[3-13]

From Equations [3-11], [3-12], and [3-13] it follows that

\[ C_E(\tau_p) = \frac{\int_0^\tau R_E(\theta, \tau_p) d\theta}{\int_0^\tau [v + v_g \cos \theta] \exp\left[\frac{1}{4} P(\cos \theta - 1)\right] d\theta} \]  

[3-14]

From equations [3-11 and 3-12] the deposition flux can be normalized by \( C_E \):

\[ \frac{R_D}{C_E} = (v + v_g \cos \theta) \cdot \exp\left[\frac{1}{4} P(\cos \theta - 1)\right] \]  

[3-15]

In Figures 6-10a and 6-10b, the deposition flux normalized by \( C_E \) (Equation [3-15]) has been plotted for two velocities as a function of the circumferential pipe angle for six different particle sizes (5, 10, 20, 50, and 100 µm particles). Figure 6-10c also represents the deposition flux for different density of particles ranges from specific gravity 1.5 to 6.0. Although these curves have been plotted in one figure, \( C_E \) is generally recalled different for different particle relaxation times. This means that for each particle relaxation time the curves in Figures 6-10 [a, b, and c] have to be multiplied by a different multiplication factor \( C_E \) with respect to the horizontal axis.

Figure 6-10b shows that 50 µm particles are just able to deposit at the bottom of the pipe wall, 100 µm particles can only deposit up to
one third of the pipe wall circumference and 200 µm particles deposit up to one sixth of the pipe wall circumference. In Figure 6-10a, the smaller particles exhibit broader deposition with minimum scale.

However, depending on weights, all particles were more or less disperse everywhere across the pipe with broader minimum scale deposition.

For the lighter and smaller particles, the influence of gravity becomes less, which results in particles depositing more uniformly around the pipe circumference. In both cases the deposition flux increases at higher velocity. The width of the deposition curves is mainly determined by $\frac{1}{4} P$. The width decreases if the Peclet number increases. The Peclet number increases if the acceleration of gravity and the radius of the pipe increase, and/or if the particle diffusion

---

**Figure 6-10a**: Deposition flux normalized by $C_e(\tau_p)$ vs circumferential pipe angle for different particle sizes (5, 10, and 20 µm) for the velocity 0.1 and 0.5 ms$^{-1}$. 

---
Figure 6-10b: Deposition flux normalized by $C_E(\tau_p)$ vs circumferential pipe angle for different particle sizes (50, 100, and 200 µm) at 0.1 and 0.5 ms$^{-1}$.

Figure 6-10c: Deposition flux normalized by $C_E(\tau_p)$ vs circumferential pipe angle for five different particle densities (sg 1.5, 2.5, 4, 5, and 6) at 0.1 and 0.5 ms$^{-1}$.

The coefficient decreases (Mols and Oliemans 1998). In the limit of $P \rightarrow 0$ the influence of gravity is negligible and $R_D(\theta)$ becomes a constant and independent of $\theta$. However, in the limit of $P \rightarrow \infty$ the influence of gravity is infinite, and there can only be deposition at the bottom of the pipe (Mols and Oliemans 1998).

6.4 CFD INVESTIGATION AND RESULTS COMPARISON

A comprehensive 3D numerical simulation has been carried out to compare with the analytical model described above making use of the existing Multiphase Mixture Model available in Fluent 6.1. The model proposed by Spalart and Allmaras (1992) solves a transport
equation for a quantity that is a modified form of the turbulent kinematic viscosity. Table 6-5 shows the physical and hydraulic characteristics of the system that has been used for CFD investigation.

Table 6-5: Physical and hydraulic characteristics of the system used for CFD simulation.

<table>
<thead>
<tr>
<th>Characteristics</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pipe length (m)</td>
<td>100.0</td>
</tr>
<tr>
<td>Diameter of the pipe D (cm)</td>
<td>4.72</td>
</tr>
<tr>
<td>No. of phases</td>
<td>6</td>
</tr>
<tr>
<td>VF of each secondary phases (ppm)</td>
<td>300</td>
</tr>
<tr>
<td>Average water velocities (ms(^{-1}))</td>
<td>0.1 and 0.5</td>
</tr>
<tr>
<td>Particles sizes ((\mu)m)</td>
<td>5, 10, 20, 50, and 100</td>
</tr>
<tr>
<td>Particle specific gravities</td>
<td>1.5, 2.5, 4.0, 5.0, and 6.0</td>
</tr>
<tr>
<td>No of computational cells</td>
<td>345849 (produce grid independent results)</td>
</tr>
</tbody>
</table>

6.4.1 Relative Concentration along Height

Figures 6-11 [a-d] show the analytical and simulated relative concentration plotted as a function of particle diameter for different heights of 0.25D, 0.5D, 0.75D, and 1D measured from the bottom wall of the pipe.

The Figures 6-11 [a-d] show that the analytical relative concentration deviated between the higher and lower velocity for the smaller particles. At higher velocity the relative concentration of

Figure 6-11a: Comparison of analytical and simulated relative concentration as a function of particle diameter at the top of pipe at 0.1 and 0.5 ms\(^{-1}\).
Figure 6-11b: Comparison of analytical and simulated relative concentration as a function of particle diameter at $y = 0.75D$ at 0.1 and 0.5 m/s$^{-1}$.

Figure 6-11c: Comparison of analytical and simulated relative concentration as a function of particle diameter at the center of pipe at 0.1 and 0.5 m/s$^{-1}$.

Figure 6-11d: Comparison of analytical and simulated relative concentration as a function of particle diameter at $y = 0.25D$ at 0.1 and 0.5 m/s$^{-1}$.
particle for analytical model was also higher than that at lower velocity. This is because of turbulence diffusion, which increases due to increase of velocity. However, simulated relative concentration showed no difference for larger particles. This is because larger particles were influenced by gravity than turbulent diffusion.

Figures 6-12 [a-d] show analytical and simulated relative concentration plotted as a function of particle specific gravity for different heights of 0.25D, 0.5D, 0.75D, and 1D from the bottom.
wall of the pipe. Most of the particles remained suspended everywhere across the pipe. For the same size of particles, density did not affect for their deposition much. This can be easily explained by considering the ratio between $v_g$ and $v$.

Table 6-4 shows that when the free flight velocity was much more higher than the gravitational settling velocity, the turbulent diffusivity dominated over body force, which may have been the cause of suspension and/or dispersion of particles.

Figures 6-11 [a-d] and 6-12 [a-d] show that the analytical relative concentration exhibited higher values than that of the CFD
simulated concentration. This may be explained by particle interaction (Manninen et al. 1996) and 3D effect, which were considered for the CFD simulation, and caused higher deposition for the CFD simulation whereas the particle interaction was ignored in the analytical turbulent diffusion model.

6.4.2 Circumferential Deposition for the CFD Simulation

Local deposition rates along the pipe circumference can be obtained from the simulation. Figures 6-13 [a-d] show typical circumferential distributions of particle volume fraction for the velocity of 0.1 and 0.5 ms$^{-1}$. Most of the profiles exhibited a smooth variation with the maximum deposition at the bottom of the pipe. The overall qualitative trend of the present numerical results were similar to those of the experimental data of Anderson & Russell (1970a) and simulated results of Mols and Oliemans (1998). However, the present results obtained for liquid-solid system cannot be compared quantitatively with those of Mols and Oliemans (1998) and Anderson and Russell (1970a), which were obtained for gas-solid system.

Figures 6-13 [a-b] show the volume fraction of particle sizes of 5, 10, 20, 50, and 100 μm, specific gravity 3.0 plotted as a function of circumferential pipe angles for five different particle sizes for the velocity 0.1 ms$^{-1}$.

![Figure 6-13a: Circumferential deposition as a function of circumferential pipe angles for five different particle sizes for the velocity 0.1 ms$^{-1}$](image-url)
Figure 6-13b: Circumferential deposition as a function of circumferential pipe angles for five different particle sizes for the velocity 0.5 ms\(^{-1}\).

Figure 6-13c: Circumferential deposition as a function of circumferential pipe angles for five different particle densities for the velocity 0.1 ms\(^{-1}\).

Figure 6-13d: Circumferential deposition as a function of circumferential pipe angles for five different particle densities for the velocity 0.5 ms\(^{-1}\).
the circumferential pipe angles, $\theta$. Particles $\geq 20$ $\mu$m showed greater concentration near the bottom wall of the pipe.

The influence of the velocity on the deposition on the pipe wall has also been shown in the Figures 6-13 [a-b]. For the smaller particle sizes the influence of the velocity change was larger (also see Figures 6-12 [a-d]). This is an effect, which is expected on the basis of the fact that for the smaller particles ($v_g < 0.25v$) the influence of turbulent diffusion was relatively large in comparison with the influence of gravity. Therefore, the particles $\leq 20$ $\mu$m ($v_g < 0.25v$) have not been influenced by gravity and showed higher deposition on the bottom at lower velocity. The uniformity of dispersion increased with the flow velocity. Larger particles ($v_g > 0.25v$), which were influenced by the gravity, settled more at higher velocity. This phenomenon was not observed in the analytical calculation.

The influence of the velocity on the deposition of 10 $\mu$m particle with different densities has been shown in the Figures 6-13 [c-d]. These figures showed the volume fraction of different particle densities of 1.5, 2.5, 4.0, 5.0, and 6.0 gm/cm$^3$ keeping constant diameter of 10 $\mu$m plotted as a function of the circumferential pipe angles, $\theta$. The particles, which exhibited $v_g < 0.25v$ (Table 6-4), were less sensitive to gravitational force and strongly influenced by the turbulent diffusivity of the fluid, which increased due to increase of the velocity (Mols and Oliemans 1998). Table 6-4 shows that the ratio of the free flight velocity to the settling velocity for 0.1 ms$^{-1}$ was relatively higher than that for 0.5 ms$^{-1}$. This resulted in higher concentration of heavier particles near the bottom of the pipe at the lower velocity 0.1 ms$^{-1}$ (Figure 6-13c) as compared to that at 0.5 ms$^{-1}$ (Figure 6-13d).

Froude number also influenced the deposition flux shown in Figures (6-10 and 6-13). The smaller and lighter particles were influenced more due to changes in the Froude number (Table 6-3). Again, this
was an effect which was expected on the basis of the fact that for the smaller and lighter particles the influence of turbulent diffusion was relatively large in comparison with influence of gravity. For a larger Froude number the deposition flux at a certain $\theta$ was larger. For the smaller and lighter particles, this effect decreased with the increasing particle relaxation time. For the particles larger than 100 $\mu$m this effect was quite small because for larger fluid velocities the effect of an increasing fluid diffusion coefficient was partly compensated by a decreasing inertial coefficient (see Tables 6-1 and 6-2).

6.5 CONCLUSION

In this chapter the effect of particle size, density and velocity on the deposition and dispersion in a horizontal pipe was presented using both analytical methods and CFD simulation has been presented. The larger particles, which had a gravitational settling velocity to free flight velocity ratio (Table 6-4) more than 0.25, in general, were influenced by the gravity. These particles showed a tendency to settlement at the bottom wall of the pipe. The smaller and lighter particles with a gravitational settling velocity to free flight velocity ratio of less than 0.25, were influenced more by turbulent diffusivity than by gravity. They were dispersed more or less uniformly across the cross section of the pipe. In the analytical model, homogeneous turbulence and uniform axial velocity through out the pipe flow were considered, which may have been the cause of the slightly higher relative concentration in comparison to the CFD simulation in which particle interaction forces were also included.

In the analytical Turbulent Diffusion Model the physical variables, which totally determine the solution were acceleration of gravity, particle relaxation time, integral fluid time scale, pipe diameter, mean square fluid velocity, and the Eulerian length scale. Particle time scale, integral fluid time scale, and fluid mean square velocity
were in fact microscopic, dependent parameters used in this model. As the influence of the liquid film was neglected in this model, the total solution depended on the macroscopic parameters, acceleration of gravity, pipe diameter, and average fluid velocity. The Froude number was the characteristic dimensionless number that was constructed from these three parameters. The distribution of particle relaxation times was determined by the Froude number and the superficial liquid velocity in a way that it was only known qualitatively that with increasing fluid velocity the particle mean diameter decreased rapidly, whereas with the increasing superficial liquid velocity the particle diameter decreased at low fluid velocities, but increased at high fluid velocities (Hay et al. 1996). The influence of Froude number on the particle size distribution was greater than the influence of the superficial liquid velocity.

CFD simulation results and analytical results differed 1%-2% (see Figures 6-7a/b) at 0.1 ms⁻¹ and less than 1% at 0.5 ms⁻¹ (see Figures 6-8a/b). This is because of the assumptions on which the analytical Turbulent Diffusion Model was based. In reality the average fluid velocity would not be uniform over the whole cross-section of the pipe, as was assumed, but only over about 80% of the pipe diameter. Turbulence would also be inhomogeneous close to the wall. This would be for turbophoresis effect that results in an extra deposition flux. However, on the other hand, especially the smaller and lighter particles would become trapped in the boundary layer. The turbulent fluctuations here became too small to support the motion of these particles to the wall. However, it was expected that the high inertia particles that were considered effectively, saw the homogeneous turbulence. Whereas, in the CFD simulation the velocity gradient and inhomogeneous turbulence near wall were considered (Fluent 6.1 2001), which resulted in slight difference in relative particle concentrations.
In this chapter a Turbulent Diffusion Model was used to predict the particle deposition flux in a horizontal turbulent pipe flow. The results were then compared with the results of CFD investigation. The new aspects of the Turbulent Model were that a one-dimensional solution was found, which was used to calculate an approximate two-dimensional solution for the deposition flux. A particle diffusion coefficient was used with the inertial and crossing trajectories effects included and these effects were calculated quantitatively.

For the one-dimensional case, the analytical solution for the particle concentration was used to calculate the relative deposition between the top and the bottom wall. The relative depositions dependence on the pipe diameter, particle size and density, and the velocity was investigated. For a certain pipe diameter the curve of the relative deposition for $V_f = 0.5 \text{ ms}^{-1}$ lay only slightly above the one for $V_f = 0.1 \text{ ms}^{-1}$ (Figures 6-11 and 6-12). This was due to the two partly compensating effects of a decreasing inertial coefficient and an increasing fluid diffusivity. The latter increased more than the former decreased, so that the total particle diffusivity increases slightly with increasing fluid velocity.
Chapter 7
Conclusion and Recommendations

7.1 CONCLUSION

The advantage of high-speed and large-memory computers has enabled CFD to obtain solutions for numerous flow problems including compressible or incompressible, laminar or turbulent, reacting or non-reacting, single phase or multiphase, steady or transient flows, and multiple combinations of above. CFD allows solid particles to be tracked through water distribution systems, and predictions can be made of locations with high potential for accumulation and deposition. Accurately predicting this deposition will reduce the amount of guesswork used to find these locations and thereby reduce the costs associated with physically exhuming pipe networks.

The CFD investigation is a quick, easy to scale, easy accessible and associated low cost technique, whereas, experiments are time consuming, expensive, some times non-accessible, and not free from instrumental and human errors. However, to established numerical results, some experimental and/or analytical data are also needed to be validated with.

The entire body of this research work was divided into three major parts and represented in corresponding chapters. First of all a comprehensive CFD investigation of particle deposition was studied in a turbulent flow field. This steady multi-particles study was
performed in a horizontal pipe loop consisting of four $90^0$ bends. The results of particle accumulation/deposition were presented along length of the pipe, height from the bottom wall and circumferential surfaces of pipe wall. These results has showed good agreement with the experimental data. The particles may accumulate at low velocity regions like bends, junctions, and dead ends and then re-entrained into the system at an abnormal high velocity.

Secondly, CFD investigation of particle deposition in a straight horizontal pipe for transient flow using current online monitoring data was also studied. This study established that particles might have accumulated and deposited at low velocity after a long journey. There was a significant affect of velocity gradient on the particle deposition and re-suspension while flowing through pipe networks. Depending on the age of deposition and length of travelling path, particles showed clear accumulation at bottom and then this accumulated particle divided to form a secondary group of particle called *kink*.

Finally, an extended analytical turbulent diffusion model was redeveloped for liquid-solid phases from an existing gas-solid turbulent diffusion model. This model established that after a certain limit of velocity ratio (for large particles), particles settled on the bottom wall of the pipe at low velocity where gravitational force dominated. However, light and small particles distributed more evenly across the pipe due to associated turbulence. This analogy was established for the CFD simulation as well with the same boundary conditions used in analytical model.

The entire research works were concerned with the multiphase flow simulation and/or analytical solution for both steady and unsteady state in turbulent flow field. The numerous particles, velocity and particle-load profiles, and pipes were selected to make this study
substantiate with the drinking water distribution networks. It was believed that considering some limitations this was the most important study, which could be helpful and evidence for understanding and solving the most of the problems regarding the turbidity spikes movement in water distribution networks of the modern world.

7.2 RECOMMENDATIONS

The simulation for biofilm growth, surface reacting flow, and charge particle would be the most important tools to understand the hydrodynamics of particle behavior in the water distribution networks.

Due to limitation of time frame for this research work, the biofilm growth, charge particle, and surface reaction were kept beyond this research. However, the simulation for biofilm growth with exact chemistry (has not yet established) could improve the understanding of turbidity spikes movement flowing through pipe networks. Charge particles, which can move towards opposite charged pipe surface, also play important role while flowing through distribution networks. These electro-magnetic forces along with Van der Waals forces between the charge particles, which are seen in real pipe networks, could be added as source terms for further study. Through the addition of these two source terms the simulations may predict the movement of turbidity spikes more accurately. Particles can sometimes chemically react with each other or with the surface of the pipe. The hydrodynamics of these products may not be same as the original particles, which is definitely important in the drinking water distribution networks for turbidity spikes flow.
Chapter 8

References


References

Journal of Water Resources Planning and Management, 126(4), 236-244.


FLUENT. FLUENT INC. 2001. USA.


Morse, P. M. and Feshbach, H. (1953). Methods of Theoretical Physics, McGraw-Hill, New York, USA.


Drinking Water Guidelines." Australia.


Taylor, G. I. (1921). "Diffusion by continuous movements."


List of Personal Publications

