

CFD MODELLING FOR HYDRAULIC AND PNEUMATIC CONVEYING THROUGH PIPELINE

ARVIND KUMAR



**DEPARTMENT OF CIVIL ENGINEERING
INDIAN INSTITUTE OF TECHNOLOGY, DELHI**

September 2010

CFD MODELLING OF HYDRAULIC AND PNEUMATIC CONVEYING THROUGH PIPELINE

by

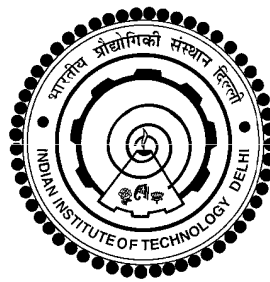
ARVIND KUMAR

Department of Civil Engineering

Submitted

**in fulfillment of the requirements of the degree of
Doctor of Philosophy**

to the



**INDIAN INSTITUTE OF TECHNOLOGY, DELHI
SEPTEMBER 2010**

CERTIFICATE

This is to certify that the thesis “**CFD MODELLING FOR HYDRAULIC AND PNEUMATIC CONVEYING THROUGH PIPELINE**” being submitted by **Mr. Arvind Kumar** to the *Indian Institute of Technology, Delhi (India)* for the award of the degree of Doctor of Philosophy in Civil Engineering Department is a bonafide research work carried out by him under my supervision and guidance. The thesis in my opinion, has reached the standard fulfilling the requirements for the Doctor of Philosophy Degree. The research report and the results presented in this thesis have not been submitted in parts or in full to any other University or Institute for the award of any degree or diploma.

(Dr. D. R. Kaushal)
Supervisor
Deptt. of Civil Engineering
Indian Institute of Technology
New Delhi-110016
INDIA

ACKNOWLEDGEMENT

I would like to express my sincere gratitude to my supervisor Dr. Deo Raj Kaushal, for his invaluable guidance, encouragement and interesting discussions through out the work, amidst his busy schedules. Words are inadequate to acknowledge the great care and interest taken by him in all aspects of the present work. His critical appraisal and suggestions have been priceless at every stage of this research.

I wish to extend my sincere thanks to Prof. A.K. Gosain, Head, Civil Engineering Department and members of my SRC, Prof. A.K. Jain, Prof. M.R. Ravi and Prof. B.R. Chahar for sparing their valuable time.

I acknowledge my sincere thanks to the staff members of Fluid Mechanics Laboratory, Mr. Onkar Singh, Mr. Dewan Singh and Mr. Manohar for their enthusiastic support. I also acknowledge my sincere thanks to Mr. Bikram Chand, Mr. Rajveer Aggarwal and Mr. N.R. Gehlot for providing me helpful cooperation in simulation laboratory of Water Resource Engineering throughout this research work.

I am thankful for the consistent moral and conducive support provided by my friends Dr. Umesh Maheswari, Mr. Anuj Prakash, Mr. Ganesh, Mr. Vishnu Tiwari, Mr. Naveen Joshi, Dr. Tsewang Thinglas, Dr. Sudhir Kumar, Mr Raktim Haldar, Mr. Maheswaran R., Mr. Shailender Kumar Jain, Mrs. Susmita Dadichi, Mr. Rajeev Saha, Mr. N.K.Tiwari, Mr. L. D. Kala, Mr. K.R. Ranjan, Mr. Navneet Kumar, Mr. Aswani Soni, Mr. Amod Kumar, Ms. Jayshri Patel, Ms. Pratiksha Pandey and Dr. Sanjay Sharma.

I am deeply indebted to Prof. Manish Jindal for his genuine help and encouragement and for blessing of my aunt and uncle, Mrs. Sashi Prabha Jindal and Mr. Gyan Chand Jindal. In fact, it is their blessings by virtue of which this thesis work has been realized.

I acknowledge my sincere thanks to Prof. Ahsok Arora and Prof. Sandeep Grover from YMCA University of Science and Technology, Faridabad for their encouragement and support.

Finally, I am indebted to my Guru Dev and my parents for their blessing in my life. I thank my wife, Mrs. Madhu Gupta, who took all the pain to manage the entire household along with the studies of our son, Sawar and daughter, Aditi. They happily allowed me to devote their share of my time in this research endeavor. At this point of time I would like to acknowledge the continuous support of my near and dear throughout my educational and research pursuits.

At last, I would like to thank all those who have directly or indirectly helped me during the period of the present work.

(Arvind Kumar)

ABSTRACT

Hydraulic and pneumatic conveying of solids through pipeline is widely used due to its many advantages. The prediction of two-phase flow of solid-fluid in hydraulic and pneumatic conveying remains a major challenge in many engineering and industrial applications. In recent years, new findings of turbulent flows and their modelling have provided the basic framework for development of mathematical models of solid-fluid flows. The emergence of powerful numerical techniques and computational fluid dynamics along with the accessibility of powerful computers has made it possible to test such models and to carry out investigations of the basic phenomena using computer simulation. The present research was undertaken to gain insight in the modelling of solid-fluid flows.

The flow behaviour of solid-liquid mixture in hydraulic conveying through slurry pipeline is very complex due to its dependence on large number of geometrical and dynamical parameters and their interdependence. In the present study, bench scale tests have been carried out to determine the physical properties of solids, carrier fluids and slurries to be transported through slurry pipeline. These physical properties are required as essential input to design any slurry pipeline system. Computational Fluid Dynamics (CFD) based commercial software FLUENT has been used to determine the pressure drop, solid concentration and velocity distribution in horizontal slurry pipeline and bend. In the development of the simulation method, single phase water flow has been considered as a basis for extension to the multiphase flow. Simulation results show good agreement with experimental data available in literature for flow of water, mono-sized

silica sand slurry and bimodal (silica sand mix with flyash) slurry in horizontal pipeline and bend.

In order to understand the flow mechanism and to calculate the energy dissipation in slurry pipeline, knowledge of the solid concentration distribution is essential. Kaushal and Tomita (2002) model for the prediction of solid concentration distribution gives good results across mid-vertical plane in horizontal slurry pipeline except near pipe bottom. A mathematical model is developed in the present study incorporating proposed correlation for settling velocity of particles near pipe bottom. The proposed model gives fairly accurate predictions for the experimental data considered in the present study from literature for solid concentration distribution in mid-vertical plane of horizontal slurry pipeline.

A mathematical model is proposed to predict the solid concentration distribution in lateral plane of horizontal slurry pipeline incorporating the effect of particle size in existing Sharp and O'Neill (1971) model. The proposed model gives fairly accurate predictions for the experimental data considered in the present study from literature for solid concentration distribution in lateral plane of horizontal slurry pipeline.

CFD based simulations have been carried out to investigate the pressure drop prediction capabilities of commercially available FLUENT software across horizontal and vertical 90° bend in pneumatic pipeline for conveying of cement. On comparison with experimental data from literature considered in the present study, broad qualitative agreement and large quantitative disparity is observed. At lower solids loading ratios the mixture model and steady state analysis have been found more appropriate. At large solids loading ratios, the deviation is large between measured and predicted values and requires further investigation.

TABLE OF CONTENTS

	Page No.
CERTIFICATE	i
ACKNOWLEDGEMENTS	ii-iii
ABSTRACT	iv-v
TABLE OF CONTENTS	vi-xiii
LIST OF FIGURES	xiv-xxii
LIST OF TABLES	xxiii-xxiv
LIST OF NOMENCLATURE AND ABBREVIATIONS	xxv-xxx
1 INTRODUCTION	1-22
1.1 GENERAL INTRODUCTION	1
1.2 HYDRAULIC CONVEYING THROUGH PIPELINE	1-7
1.2.1 Slurry Pipeline System	3-4
1.2.2. Flow Regimes in Slurry Pipeline	5-7
1.3 PNEUMATIC CONVEYING THROUGH PIPELINE	7-10
1.3.1 Pneumatic Conveying Pipeline System	7
1.3.2 Classification of Pneumatic Pipeline Systems	8-10
1.4 EFFECTS OF PIPE BEND IN HYDRAULIC AND PNEUMATIC CONVEYING	10-12
1.5 NUMERICAL MODELLING OF HYDRAULIC AND PNEUMATIC CONVEYING THROUGH PIPELINE	12-17
1.5.1 Advection Diffusion (AD) Model	12-13
1.5.2 Direct Numerical Simulations (DNS) Model	13
1.5.3 Computational Fluid Dynamics (CFD) Models	13-15
1.5.3.1 Lagrangian (Euler- Lagrange) model	15
1.5.3.2 Eulerian (Euler-Euler) model	15
1.5.3.3 Discrete Element Method (DEM) model	15-16
1.5.4 Coupled Models	16-17
1.5.4.1 The continuum mechanics model	16

1.5.4.2 The lattice Boltzmann method (LBM)	16
1.5.4.3 The semi-discrete-continuum model (Coupled CFD-DEM)	17
1.6 SOFTWARES USED IN CFD MODELLING	17-19
1.7 OBJECTIVES OF THE PRESENT STUDY	19
1.8 ORGANIZATION OF THE THESIS	20-21
2 LITERATURE REVIEW	22-91
2.1 INTRODUCTION	22
2.2 HYDRAULIC CONVEYING	22-48
2.2.1 Pressure Drop in Slurry Pipelines	23-32
2.2.2 Distribution of Solid Concentration in Slurry Pipelines	32-42
2.2.3 Slurry Flow in Pipe Bend	42-48
2.3 PNEUMATIC CONVEYING	48-53
2.3.1 Pressure Drop in Pneumatic Pipelines	48-49
2.3.2 Concentration Distribution of Solids in Pneumatic Pipelines	49-51
2.3.3 Pneumatic Transportation through Pipe Bends	51-53
2.4 VISUALISATION TECHNIQUES FOR THE HYDRAULIC AND PNEUMATIC CONVEYING THROUGH PIPE LINE	53-57
2.5 COMPUTATIONAL FLUID DYNAMICS (CFD) METHODOLOGY	57-72
2.5.1 Turbulence Modelling in CFD Methodology	58-62
2.5.2 Near Wall Treatment for Turbulent Flows	62-66
2.5.2.1 Standard wall functions	64-65
2.5.2.2 Non-equilibrium wall functions	65
2.5.2.3 Grid Adaption at the Wall	65-66
2.5.3 Numerical Discretization Techniques	66-68
2.5.3.1 The Finite difference method	66-67
2.5.3.2 The finite element method	67
2.5.3.3 The Finite volume method	67-68
2.5.4 Upwinding Scheme	69
2.5.4.1 First order upwind scheme	69

2.5.4.2 Second order upwind scheme	69
2.5.5 Solver used in FLUENT	70
2.5.6 Under Relaxation Factor	70
2.5.7 Solution Strategy and Convergence	71-72
2.6 COMPUTATIONAL MODELLING FOR HYDRAULIC AND PNEUMATIC CONVEYING	72-75
2.7 TWO PHASE FLOW MODELLING USING FLUENT SOFTWARE	75-84
2.7.1 Eulerian Model	76-83
2.7.2 Mixture Model	83-84
2.8 SCOPE OF THE PRESENT STUDY	85
2.9 RESEARCH METHODOLOGY	86-89
2.10 CONCLUDING REMARKS	90
3 BENCH SCALE TESTS, EXPERIMENTAL SET-UP AND INSTRUMENTATION	91-108
3.1 INTRODUCTION	91
3.2 BENCH SCALE TESTS	91-97
3.2.1 Determination of Particle Size Distribution	91-93
3.2.2 Determination of Static Settled Concentration	93-95
3.2.3 Specific Gravity of Solids	95
3.2.4 Specific Gravity and Concentration of Slurry Samples	95
3.2.5 Unhindered Settling Velocity	95-96
3.2.6 pH of the Slurry	96
3.2.7 Angle of Repose	96
3.2.8 Wear Test	96-97
3.2.9 Microscopic Study of Solid Particles	97
3.3 PILOT PLANT TEST LOOP	97-100
3.4 INSTRUMENTATION	100-102
3.4.1 Flow Rate Measuring Devices	100-101
3.4.2 Pressure Drop Measurement	101
3.4.3 Measurement of Concentration Distribution	102
3.5 BENCH SCALE ANALYSIS	102-105

3.6 SOURCES OF ERROR IN EXPERIMENTS	106
3.6.1 Sources of Errors in the Concentration of Tested Samples	106
3.6.2 Errors due to Measurements in the Pipe Loop	106
3.7 CONCLUDING REMARKS	107
4 CFD MODELLING AND SOLUTION METHODOLOGY AND ITS VALIDATION FOR PRESSURE DROP ACROSS 90° HORIZONTAL BENDS FOR WATER FLOW	108-132
4.1 INTRODUCTION	108
4.2 CFD SOFTWARE PACKAGE	108-115
4.2.1 Construction of Simulation Geometry	109
4.2.2 Generation of Mesh	109-111
4.2.2.1 Boundary layer mesh	110-111
4.2.2.2 Face and volume meshing	111
4.2.2.3 Quality of mesh	111
4.2.3 Grid Adaptation for y^+	112
4.2.4 Grid Independence Test	112-113
4.2.5 Solution Methodology	113-114
4.2.5.1 Enabling assumptions	113
4.2.5.2 Solver parameters	113-114
4.2.6 Operating Conditions	114
4.2.7 Boundary Conditions	114-115
4.2.7.1 Inlet boundary conditions	114-115
4.2.7.2 Outlet boundary conditions	115
4.2.7.3 Wall boundary conditions	115
4.2.8 Convergence	115
4.3 VERIFICATION AND VALIDATION OF CFD MODEL	116-117
4.4 CFD MODELING FOR PRESSURE DROP ACROSS 90° HORIZONTAL BENDS OF WATER	117-132
4.4.1 Geometries of Bends used in CFD Simulations	119-120
4.4.2 Range of Parameters used in CFD Simulations	120

4.4.3 Computational Grids	121
4.4.4 Solution Strategy and Convergence	122
4.4.5 CFD Modelling Results	123-132
4.5 CONCLUDING REMARKS	132
5 CFD MODELING FOR PRESSURE DROP AND CONCENTRATION DISTRIBUTION OF MONO-SIZED AND BIMODAL SLURRIES THROUGH 90° HORIZONTAL PIPE BEND	133-230
5.1 INTRODUCTION	133-134
5.2 PHYSICAL PROPERTIES OF THE MATERIALS USED IN EXPERIMENTAL DATA FOR VALIDATION OF CFD SIMULATION	134-141
5.2.1 Range of Parameters used in CFD Simulation	134-135
5.2.2 Simulation Geometry	135-136
5.2.3 Grid Generation	136-137
5.2.4 Pipe Bend	137-138
5.2.5 Numerical Simulation	138
5.2.6 Parameter Selections	138-140
5.2.6.1 Grid adaptation for y^+	140
5.2.6.2 Grid independence test	140
5.2.7 Residual Convergence	141
5.3 MODELING RESULTS	142-228
5.3.1 Mono-sized Silica Sand Slurry	142
5.3.1.1 Pressure drop	142-143
5.3.1.2 Comparison of CFD modelling results with experimental data for pressure drop	143-149
5.3.1.3 Concentration distribution	149-151
5.3.1.4 Comparison of CFD modelling results with experimental data for concentration distribution	151-161
5.3.1.5 Velocity distribution	162-180
5.3.2 Bimodal Silica Sand-Flyash Slurry	181
5.3.2.1 Pressure drop	181

	5.3.2.2 Comparison of CFD modelling results with experimental data for pressure drop	182-188
	5.3.2.3 Velocity distribution	188-189
	5.3.2.4 Concentration distribution	189-200
	5.3.2.5 Comparison of CFD modelling results with experimental data for solid concentration distribution	200-221
	5.3.2.6 Analysis of modeling results of mono-sized slurry and bimodal slurry flow through pipeline and bend	221-225
	5.4 CONCLUDING REMARKS	226-227
6	PREDICTION OF SOLID CONCENTRATION DISTRIBUTION ACROSS MID-VERTICAL PLANE IN HORIZONTAL SLURRY PIPELINE	228-244
	6.1 INTRODUCTION	228-236
	6.2 BASIC DETAILS OF KARABELAS (1977) MODEL	229-231
	6.3 EXPERIMENTAL DATA AVAILABLE IN LITERATURE USED FOR COMPARISON	231-237
	6.4 STEPS IN DEVELOPMENT OF PROPOSED MODEL FOR SETTLING VELOCITY OF PARTICLES NEAR PIPE BOTTOM	233-234
	6.5 STEPS FOR DEVELOPMENT OF COMPUTER PROGRAM FOR MODIFIED KARABELAS [KAUSHAL & TOMITA (2002)] MODEL INCORPORATING THE PROPOSED MODEL FOR SETTLING VELOCITY NEAR PIPE BOTTOM	234-236
	6.6 COMPARISON BETWEEN MEASURED, KAUSHAL AND TOMITA, (2002) AND PROPOSED MODEL FOR SOLID CONCENTRATED DISTRIBUTION ACROSS MID-VERTICAL PLANE IN HORIZONTAL SLURRY PIPELINE	237-243
	6.7 CONCLUDING REMARKS	244

7	PREDICTION OF SOLID CONCENTRATION DISTRIBUTION ACROSS LATERAL PLANE IN HORIZONTAL SLURRY PIPE LINE	244-258
	7.1 INTRODUCTION	245-246
	7.2 PROPOSED MODEL FOR SOLID CONCENTRATION DISTRIBUTION ACROSS LATERAL PLANE IN HORIZONTAL SLURRY PIPELINES	247-248
	7.3 STEPS IN DEVELOPMENT OF THE SOLID CONCENTRATION DISTRIBUTION MODEL FOR SOLID CONCENTRATION ACROSS HORIZONTAL PLANE	248-249
	7.4 EXPERIMENTAL DATA USED FOR COMPARISON	249
	7.5 COMPARISON BETWEEN MEASURED AND PREDICTED SOLID CONCENTRATION DISTRIBUTION ACROSS LATERAL PLANE IN HORIZONTAL SLURRY PIPE LINE	250-257
	7.6 CONCLUDING REMARKS	258
8	CFD MODELLING FOR PRESSURE DROP AND CONCENTRATION DISTRIBUTION IN PNEUMATIC CONVEYING THROUGH PIPE LINE	259-271
	8.1 INTRODUCTION	259
	8.2 EXPERIMENTAL DATA AVAILABLE IN LITERATURE USED FOR VALIDATION OF CFD MODELLING	259-261
	8.3 GEOMETRIES FOR CFD SIMULATION	262-263
	8.4 NUMERICAL SIMULATION	263-266
	8.4.1 Parameter Selections	263-264
	8.4.2 Grid Adaptation for y^+	264
	8.4.3 Grid Independence test	264-265
	8.4.4 Residual Convergence	265
	8.4.5 Boundary Conditions	265-266
	8.5 CFD MODELLING RESULTS	266-271
	8.5.1 Pressure Drop	266
	8.5.2 Comparison of CFD Modelling Results with Experimental Data for Pressure Drop	267-269

8.5.3 Concentration Distribution	270-271
8.6 CONCLUDING REMARKS	271
9 RESEARCH CONCLUSIONS AND SCOPE FOR FUTURE WORK	272-275
9.1 CONCLUSIONS	272-274
9.2 CONTRIBUTION TO KNOWLEDGE	274-275
9.3 SCOPE FOR FUTURE WORK	275
REFERENCES	276-297
ANNEXTURE	298-310
PUBLICATION FROM THE THESIS	311
BIO-DATA	312