CFD MODELING AND EXPERIMENTAL VALIDATION OF SOLID - LIQUID TWO - PHASE FLOW IN CLOSED CONDUIT AND OPEN CHANNEL

By

Tsewang Thinglas

Department of Civil Engineering

Thesis submitted
in fulfillment of the requirements for the degree of

DOCTOR OF PHILOSOPHY

to the

INDIAN INSTITUTE OF TECHNOLOGY, DELHI

INDIA

JANUARY, 2008

1. Fluid flow
2. Solid - liquid two phase flow

DEDICATED TO MY GRANDMOTHER

(LATE. SONAMSKIT DAIDAY)

CERTIFICATE

This is to certify that the thesis **CFD MODELING AND EXPERIMENTAL VALIDATION OF SOLID-LIQUID TWO-PHASE FLOW IN CLOSED CONDUIT AND OPEN CHANNEL** being submitted by **Tsewang Thinglas** 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.

Deo Raj Kaushal
31.07.2008

(Deo Raj Kaushal)

Assistant Professor

Deptt. Of Civil Engineering
Indian Institute of Technology
New Delhi-110016

ACKNOWLEDGEMENT

I feel immense pleasure in expressing my profound sense of gratitude to my supervisor, Dr. D. R. Kaushal, Department of Civil Engineering, Indian Institute of Technology, Delhi for his consistent guidance and constructive criticism during the course of the present investigation, preparation of the manuscript and utmost cooperation at every stage, which culminated in successful completion of my research work. It's an honour to be his student and no amount of appreciation can be good enough to express my gratitude. I am deeply indebted to him for giving me a chance to study this subject and providing constant guidance throughout this work.

I am grateful to my M.Tech supervisor, Prof. Kapil Gupta, Department of Civil Engineering, Indian Institute of Technology, Bombay for his inspiring and most keen guidance and invaluable suggestions during my M.Tech at IIT Bombay, which created a strong base in the field of research.

I am also grateful to Prof. Manoj Datta, Prof. Shashi Mathur and Prof. Sanjeev Sanghi of IIT Delhi for their critical review, valuable suggestions and support to this work. Thanks to M.Tech student, Mr. Vineet Upadhyay, for helping me in performing experimental work and data collection.

I am thankful to Department of Science and Technology, Ministry of Human Resources Department, Government of India for sponsoring and funding me for higher studies at Indian Institute of Technology Delhi and IIT Delhi for giving me the plate form for the research work.

Special thanks must go to my elder brother Tashi Namgail, Mom and Dad, for without them, this thesis and time in IIT Delhi would have not been possible.

Also I am thankful to my special friends Amit Batra, Deepak Rajput, Arshad Hashmi, Somnath Jha, Ramana Reddy, Jeeva Rathinam, Yogesh Sharma, Kapil Sharma, who have helped and emotionally supported me through the period of the study in IIT Delhi.

This acknowledgement shall remain incomplete unless I thank my fellow research scholars Shailendra Jain, Amardeep Singh, D. L. Parmar, R K Prasad, G P Vadodaria, Brijesh Yadav, Pradeep Lodha, Mehmood, Debajit Sathapati, Shishir Gaur, Nagraj Patil, Sushmita, Manika, Monalisha, and Ravi Kumar for their help, encouragement during this study helping me to recover my health fast to complete my research work on time.. The help provided by Mr. Bikram Chand , Mr. Rajveer Agrawal and Mr. Gehlot, administrative staff of Civil Engineering Department, I. I. T. Delhi is gratefully acknowledged. Everyone else who has helped me throughout the period of the study also deserves a big thank you.

New Delhi, January 2008

(Tsewang Thinglas)

ABSTRACT

Solid-liquid two-phase flows through closed conduit and open channel are widely employed in the chemical, mining industries and are encountered in slurry transport, sewer line, irrigation channel and natural phenomena such as river mechanics. Computational Fluid Dynamics (CFD) based approach is capable of evaluating three-dimensional solid-liquid two phase flow problems under a wide range of flow conditions and sediment characteristics rapidly, which is almost impossible experimentally.

Conveyance of solids through closed conduit such as pipelines on large scale has now come to be accepted as a viable alternative to the conventional modes of transportation. The advection-diffusion (AD) model has been extensively used to predict the particle concentration distribution in slurry pipeline. AD model accounts for two effects, i.e., settling attributable to gravity and turbulent diffusion attributable to sediment-turbulence interaction. However, AD model is unable to predict the concentration profiles with points of maximum concentration away from pipe bottom, which is a common feature for finer and coarser particles at higher and all values of concentrations and velocities, respectively. The use of CFD-based approach considering flow as three-dimensional and taking into account the effects of slip-velocity and the sediment stress gradient is able to predict the concentration profiles with points of maximum concentration away from pipe bottom. In the present study, pipeline flow of mono-dispersed finer particles at higher concentration is numerically simulated using Mixture and Eulerian two-phase CFD models. Mixture model uses a single-fluid approach. Eulerian model implements Euler–Euler coupled governing equations for fluid and solid phases considering both the flow–particle and particle–particle interactions, and

a modified $k - \varepsilon$ turbulence closure for the fluid phase. Both the modeling systems are part of the CFD software package FLUENT. A hexagonal shape and cooper type non-uniform three-dimensional grid are chosen to discretize the entire computational domain, and a control volume finite difference method is used to solve the governing equations. The modeling results are compared with the experimental data available in literature in 54.9 mm diameter horizontal pipe on glass beads with mean diameter of 125μm, for flow velocity up to 5m/s and four overall concentrations up to 50% (namely, 0%, 30%, 40% and 50%) by volume for each velocity. The modeling results by both the models for pressure drop in the flow of water through pipeline are found to be in good agreement with experimental data. For flow of slurry, Mixture model fails to predict pressure drops correctly. The amount of error increases rapidly with the slurry concentration. However, Eulerian model gives fairly accurate predictions for both the pressure drop and concentration profiles at all efflux concentrations and flow velocities considered in the present study. Velocity and slip-velocity distributions that have never been measured experimentally at such higher concentrations, predicted by two-phase Eulerian model are presented for the concentration and velocity ranges covered in the present study. Slip velocity between fluid and solids dragged most of the particles in the central core of pipeline, resulting in to the central concentrations higher than the concentration in near boundary zone of slurry pipeline at higher overall average concentrations and mean flow velocities.

Solid-liquid two phase flows in open channel occur in many practical situations like sediment transport through irrigation channel, sewer line, etc. In modern times of rapid urbanization, the waste water discharges into the urban drainage channels are increasing. It is observed that construction debris, natural sediments and urban solid waste are carried in flows in drainage channels. Sediments get deposited along the invert of the drainage channels during the periods of dry weather flow due to non-availability of sufficient flow velocity in some reaches which leads to the sedimentation and reduction in the carrying capacity of the channel. Invert traps have successfully been used to collect sewer sediments at convenient locations within the system, where large volumes of solids can be stored. Most of the earlier studies available in literature on invert trap have considered the various possible trap openings, sizes, sediment types and flow rates to trap sediment optimally by using CFD-based approach and experimentally, however, only a few studies are carried out on the possible shapes of invert trap to trap sediments optimally. In the second part of the present study, flow field prediction and optimization of invert trap configuration have been carried out by using CFD modeling with the help of FLUENT software using Renormalization Group (RNG) k-ε along with Discrete Phase Model (DPM). Five different invert trap shapes (namely, rectangular, trapezoidal, trapezoidal with rectangular base, rectangular with trapezoidal base with lid on both sides) for three trap openings viz. 15 cm, 9 cm and 5 cm, have been simulated for the flow of seven different inert sediment types (namely, two types of sand, glass beads and four types of plastic beads) at different flow rates for each trap. The discharges selected in present study cover entire range of discharge expected in channels during dry weather flow and monsoon. The simulation results are capable of showing particle trajectories, the effect of flow rate and trap geometry on the flow patterns, developed within the trap. Based on 3D CFD modeling and experimental measurements, it is concluded that the invert trap having rectangular shape with trapezoidal base with lid on both the sides is the most efficient trap configuration with highest sediment retention ratio

TABLE OF CONTENTS