

CFD Investigation of Particle Deposition in a Turbulent Flow: Horizontal Pipe Loop with 4 Bends

By Alamgir Hossain¹, Jamal Naser², and Kerry McManus³

ABSTRACT

In this paper a comprehensive 3D numerical investigation of hydrodynamics of particles flowing through in a horizontal pipe loop consists of four bends has been modeled. The multiphase mixture model available in Fluent 6.1 (2001) is used in this study. In this numerical simulation five different particles have been used as secondary phases to calculate real multiphase effect 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. Particle concentration is seen high at the bottom wall in the pipe flow before entering the bends but for the downstream of bend the deposition is 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 is more pronounced at higher velocity. This numerical investigation showed good agreement with the experimental results conducted by CSIRO team (Grainger et al. 2003).

¹ School of Engineering and Science, Swinburne University of Technology, Hawthorn, VIC 3122, Australia.

² Senior Lecturer, School of Engineering and Science, Swinburne University of Technology, Hawthorn, VIC 3122, Australia.

³ Director, Infrastructure Service Management Research, School of Engineering and Science, Swinburne University of Technology, Hawthorn, VIC 3122, Australia.

Key Words: Multiphase, turbulent flow, particle deposition, horizontal pipe flow, bends, CFD investigation