

CFD Investigation of Particle Deposition Around Bends in a Turbulent Flow

By **A. Hossain and J. Naser**

School of Engineering and Science, Swinburne University of Technology
PO Box 218, Hawthorn, VIC 3122, AUSTRALIA

Abstract

A comprehensive 3D numerical investigation of hydrodynamics of particles flowing through a horizontal pipe loop consisting of four bends has been modeled. The multiphase mixture model available in Fluent 6.1 [6] 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 accounted. The deposition of particles, along the periphery of the wall around bends has been investigated. The effect of bend and fluid velocity on particle deposition 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 of the bends. 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 [9].

For details please contact ahossain@swin.edu.au