WIT CONFERENCES
This paper is part of the Proceedings of the 11th International Conference on Engineering Sciences (AFM 2016)
www.witconferences.com

Effect of radius ratio on pressure drop across a 90° bend for high concentration coal ash slurries

A. Rawat, S. N. Singh & V. Seshadri
Department of Applied Mechanics, Indian Institute of Technology, India

Abstract

High concentration slurry pipelines are now being increasingly preferred for the transportation of coal ash slurries in thermal power plants. The focus of the present paper is to establish the effect of radius ratio on the pressure drop across pipe bends at high concentrations ($C_w \geq$ 60% by weight) where the distribution of solids is expected to be homogeneous and hence to optimize the bend geometry for such flows. Fly ash slurries at concentrations above 60% (by weight) have a strong non-Newtonian character with Bingham fluid type of behaviour. Numerical simulation at high concentrations (60% to 68% by weight) through a pipe bend has been carried out using the commercially available CFD software FLUENT. For the laminar regime, the measured values of yield stress and Bingham viscosity are used as input, whereas for turbulent flow several turbulence models have been tried to establish the optimum turbulence model. To achieve this objective, the predicted results are compared with the experimental data obtained by earlier investigators at IIT Delhi on coal ash slurries with concentrations in the range 50 to 65% by weight. It was seen that predictions made using K-ω (SST) turbulence model were in good agreement with experimental data. Hence, using the K-ω (SST) model for turbulent flow and the Bingham plastic model for laminar flow, pressure drops for eight different 90° bends with different radius ratios (R/r) in the range 1 to 6 and sharp cornered bend (R/r=0) have been computed for different concentrations in the range of 60 to 68% by weight. On the basis of the study, the optimum value of radius ratio is found to lie between 3 and 4.
Keywords: HCSD systems, non-Newtonian fluid, Bingham plastic model, pipe bend, bend radius ratio, K-ω SST turbulence model.

www.witpress.com, ISSN 1743-3533 (on-line)
doi:10.2495/AFM160061

1 Introduction

Disposal of fly ash through pipelines in the form of slurry within thermal power plants and from thermal power plants to ash ponds is done at higher concentrations (solid concentration $C_w \geq 60\%$ by weight) to save water and reduce energy consumption. Bends are an integral part of any pipeline network and cause extra pressure drop. This study is undertaken to optimise the bend geometry using CFD which will result in minimum pressure drop for the flow of high concentration slurries through it. This will help designers of HCSD systems to select optimum bend for their applications without going into costly experimentation. Another major objective of the study is to establish the most suitable turbulence model for the flow of high concentration ash slurries.

Several investigators have established that fly ash slurries at high concentrations (above 60% by weight) behave like homogeneous suspensions during the flow through pipelines [1, 2]. Further, slurries of fly ash at concentrations above 60% by weight have been observed to exhibit non-Newtonian behavior [3]. Bingham plastic model has been found to be adequate for describing the rheology of such slurries at moderate shear rates [4].

Ito [5] probably was the first person to carry out a systematic study for flow through pipe bends using water. He used smooth pipe bends having radius ratios in a wide range (1.25–21.6). He was the one who established that for determining permanent pressure drop across bends both upstream and downstream pipe lengths are required. Mishra *et al.* [6] have done experimental and numerical investigation to understand the flow characteristics in long radius constant area as well as diverging-converging bends for single phase fluid. For radius ratio related studies they considered radius ratios of 2.0 to 5.0. They observed that radius ratio and area ratio considerably affect the flow characteristics of single phase fluid through bends. They also found that increasing radius ratio results in decreasing bend loss coefficient. Mishra *et al.* [7] have also done experimental studies for pressure drop across conventional bends and varying area ratio bends. In this paper they varied solid concentrations from 10% to 45% with a velocity variation of 1 to 3 m/s. Kumar *et al.* [8] also conducted experimental studies in rough 90° bends of different radius ratio for single phase flow and established that bend loss coefficient is minimum for a bend having radius ratio of 5.6 and also re-established that the flow disturbances caused by the bend persist up to 40–50 pipe diameters downstream of the bend and no significant disturbance is seen upstream of the bend. Verma *et al.* [4], whose work has been used here for validation of CFD methodology, also carried out experimental studies for the slurry flow through pipe bends. They selected optimum bend of radius ratio 5.6 by doing experiments on the flow of water through pipe bends and later on conducted pressure drop studies across this bend for the flow of high concentration coal ash slurries (see fig. 1). Csizmadia and Csaba [9] also tried to identify the optimum radius ratio bend for Bingham plastic fluid flow but their studies are based on CFD simulations only without substantiation from experimental data.

2 CFD methodology with mathematical modelling

2.1 Governing equations

The governing equations for this type of fluids are the ones which are mentioned in the user manual of FLUENT (Version 6.3.26) [10]. The continuity equation for conservation of mass is given as:

$$\nabla.(\rho\vec{v}) = 0 \tag{1}$$

Conservation of momentum equation is given as:

$$\nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) \tag{2}$$

where p is static pressure and $\bar{\bar{\tau}}$ is the Stress tensor and for laminar flow of Bingham plastic fluids, this is given as:

$$\bar{\bar{\tau}} = \bar{\tau}_0 + \mu_B \bar{\bar{D}} \tag{3}$$

where $\bar{\bar{D}}$ is given as:

$$\bar{\bar{D}} = \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) \tag{4}$$

μ_B is Bingham viscosity and $\bar{\tau}_0$ is the yield stress.

2.2 Turbulence models

For turbulent slurry flow through pipes, appropriate turbulence model needs to be identified .For the present investigation, the following two equation models were considered.

k-ω (standard and SST)

k-ε (standard, RNG and realizable)

The details of the models are as in Dewan [11].

2.3 Geometry

The computational domain consists of 3-D bends of different radius ratios (Sharp Cornered Bend (R/r=0) and R/r=1,2,3,4,5,5.6 and 6) developed in GAMBIT (Version 2.4.6) along with pipe lengths 4.6 m and 4 m upstream and downstream respectively for each bend. Here bend with R/r=0 is the elbow having sharp corners and bend with R/r=1 is the common 90° elbow. The diameter of the pipeline has been taken as 50 mm NB. Sample mesh across the pipe cross section for radius ratio 5.6 is shown in fig. 2. After grid independence tests the sizes of mesh volumes selected are shown in table 1.

2.4 Boundary layer meshing

The boundary layer meshing has been used along the wall surfaces to numerically model the large gradients of all parameters close to the wall. The boundary layer comprises of ten rows. Each row consists of an equal number of cells. The first layer is at the distance of 0.1 mm from the wall. The growth factor is selected as 1.1.

Table 1: Mesh volumes selected for various radius ratio bends.

Radius ratio (R/r))	Total mesh volumes	Radius ratio (R/r))	Total mesh volumes
0.0 (Sharp corner bend)	210638	4.0	439200
1.0 (Elbow)	684608	5.0	439200
2.0	439200	5.6	439200
3.0	439200	6.0	439200

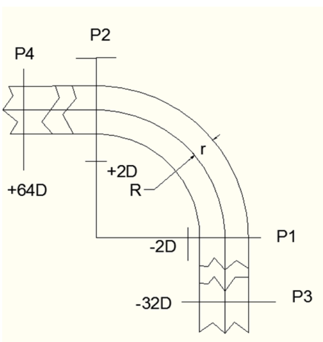

Figure 1: Geometric details of 90° horizontal bend used by Verma *et al.* [4].

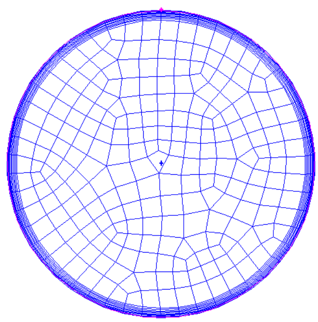

Figure 2: Mesh across the pipe cross-section for bend with radius ratio 5.6.

2.5 Boundary conditions

The calculation domain is bounded by the three boundaries: inlet boundary, outlet boundary and wall. At the inlet a uniform velocity is fed and sufficient upstream length was provided to get fully developed flow at the bend inlet. The outlet was given as outflow. Wall was treated as stationery and with no slip. Average roughness height was calculated using water flow data and it was found to be of the order of 10^{-4} m for all cases. The roughness constant was taken as 0.5.

2.6 Solution control and convergence

A first order upwind discretization scheme was used for the momentum equation, turbulent kinetic energy and specific dissipation energy. The convergence criterion based on the residual value of the calculated variables, i.e., mass, velocity components, turbulent kinetic energies, Specific dissipation rate was set to 10^{-6} times the residual value for each variable. For pressure–velocity coupling, the SIMPLE algorithm has been used. The additional solution strategies adopted are; the reduction of under relaxation factor of momentum to 0.7, turbulence kinetic energy and specific dissipation rate to 0.8.

2.7 Material selection

For high concentration fly ash slurries, distribution of solids is taken as homogeneous and behaves like Bingham fluid [1, 2]. In FLUENT, the Herschel Bulkley model has been modified to represent the Bingham plastic model. The Herschel Bulkley model is given as:

$$\tau = \mu_a \dot{\gamma} = \tau_0 + K\left[\dot{\gamma}^n - \left({}^{\tau_0}/_{\mu_0}\right)^n\right] \tag{5}$$

where μ_a=apparent viscosity, k=consistency index, n=power law index, $\dot{\gamma}$ =shear rate, τ_0=yield stress and μ_0=yielding viscosity. For modification k=μ, n=1, and $(\tau_0/\mu_0) \rightarrow 0$ has been entered into FLUENT. In particular here μ_0 is given as 10^{10}. This modification converts the Herschel Bulkley model of FLUENT to the Bingham model as below:

$$\tau = \tau_0 + \mu\dot{\gamma} \tag{6}$$

where μ=constant plastic or Bingham viscosity.

3 Experimental data of Verma *et al.* [4]

Verma *et al.* [4] performed experiments in the Fluid Mechanics Laboratory of IIT Delhi. They used fly ash sample collected from, NCPC, Dadri (India) having weighted mean diameter (d_{wm})=0.35mm.The pressure drop for five overall concentrations (50%, 55%, 60%, 62% and 65% by weight) across the bend of radius ratio 5.6 was measured by varying the velocity up to 3 m/s for each sample. Here pressure drop across bend means pressure drop between the locations 2-D Upstream and 2-D downstream of the bend. The diameter (D) of the upstream and downstream pipes was 0.05 m. Geometrical details of the bend chosen by them are as given in fig. 1 and table 2 below.

Table 2: Details of the 90° horizontal bend used by Verma *et al.* [4].

Pipe diameter	50 mm NB
Radius of curvature of bend	148.4 mm
Radius ratio R/r	5.6
Length of pipe bend	4.4D

www.witpress.com, ISSN 1743-3533 (on-line)

4 Prediction of pressure drop in high concentration slurry system across a 90° bend using CFD

For the present work, numerical simulation of pipeline slurry flow of a homogeneous mixture at high concentration (60% to 68% by weight) has been carried out to predict the pressure drop across the bend of radius ratio 5.6. Here also the pressure drop across bend means pressure drop between the locations 2-D upstream and 2-D downstream of the bend. For the simulation, laminar model for laminar flow and k-ε (standard, RNG and reliable) and k-ω (standard and SST) turbulence models of the CFD package FLUENT for modeling turbulent flow are used. After establishing grid independence, optimum mesh sizes as mentioned in table 1 are used for all predictions. For validation, the predicted results are compared with the experimental data reported by Verma *et al*. [4] as shown in fig. 3.

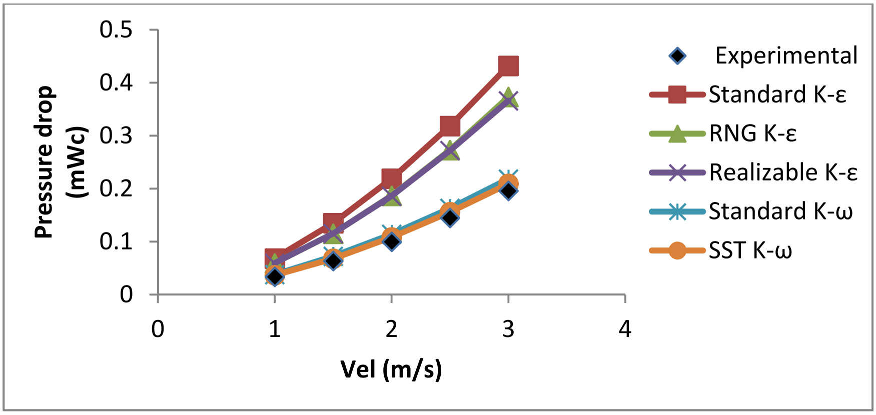

Figure 3: Pressure drop vs. velocity for different turbulence models (Cw = 60%, R/r=5.6).

Hanks and Pratt [12] have given a correlation for a critical Bingham Reynolds no. (R_{eBC}). The flow remains laminar if $R_{eB}<R_{eBC}$ otherwise it is assumed to be turbulent. Hence for the fly ash sample taken, at 60% solid concentration, for velocities from 1 to 3 m/s, flow is in turbulent regime. Considering this fact pressure drop has been calculated after processing the mesh files in FLUENT, for all the above mentioned k-ε and k-ω models one by one. The pressure drops are computed in meters of water column (mWc). Fig. 3 clearly shows that the pressure drop obtained with the k-ω SST turbulence model is much closer to the experimental values obtained by Verma *et al*. [4].

As per the criteria of Hanks and Pratt [12] the flow of the slurry is in a laminar regime up to a critical velocity of 0.5 m/s (C_w=60% and 62%).The values of critical velocities for the other two higher concentrations (C_w=65% and 68%) are 1.5 m/s and 4 m/s respectively. Thus the flow is laminar throughout the velocity range considered (1–3 m/s) at the highest concentration.

Now using k-ω SST model for turbulent flow and laminar model for laminar flow, pressure drop across the bends of various radius ratios has been calculated at C_w values of 60%, 62%, 65% and 68%.The radius ratios of the bends considered in the study are 0, 1, 2, 3, 4, 5, 5.6 and 6. The range of velocity selected for the studies is the same as that exists in prototype high concentration slurry disposal pipelines (HCSD) i.e. from 1 m/s to 3 m/s. Figs 4 to 7 show the values of pressure drops across the bends, computed using CFD at various mentioned C_w values.

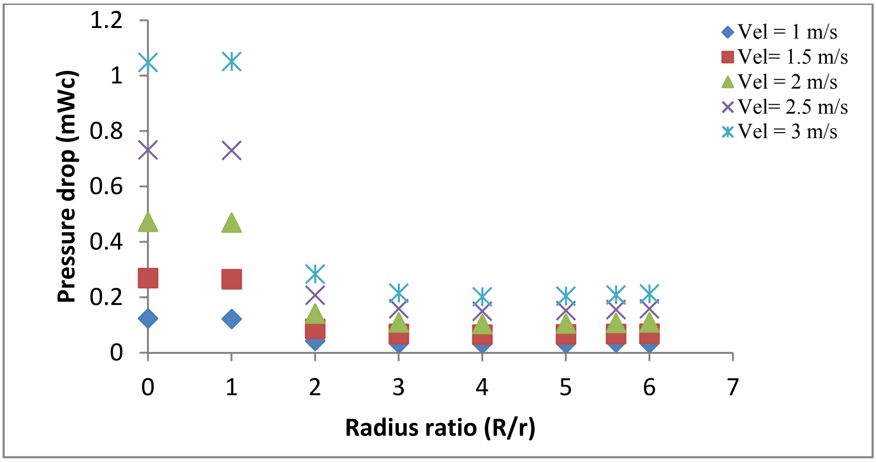

Figure 4: Pressure drop across the bend for different R/r values at various velocities (Cw=60%, D=50mm).

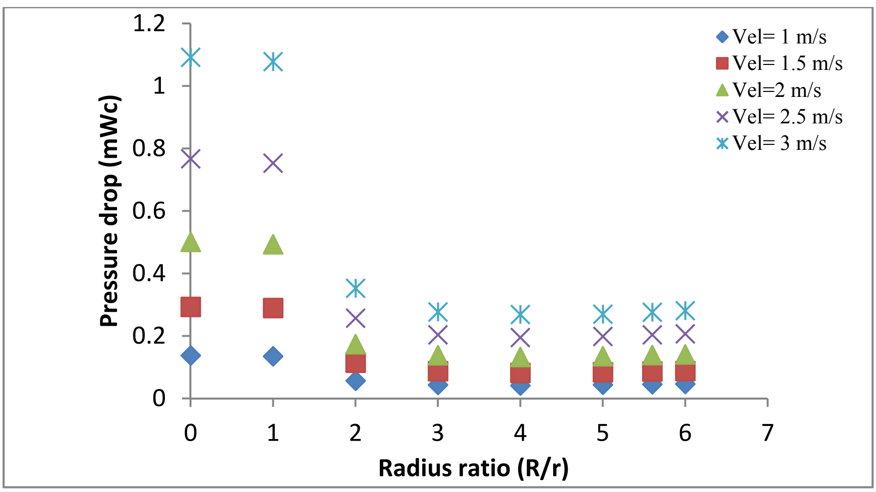

Figure 5: Pressure drop across the bend for different R/r values at various velocities (Cw=62%, D=50mm).

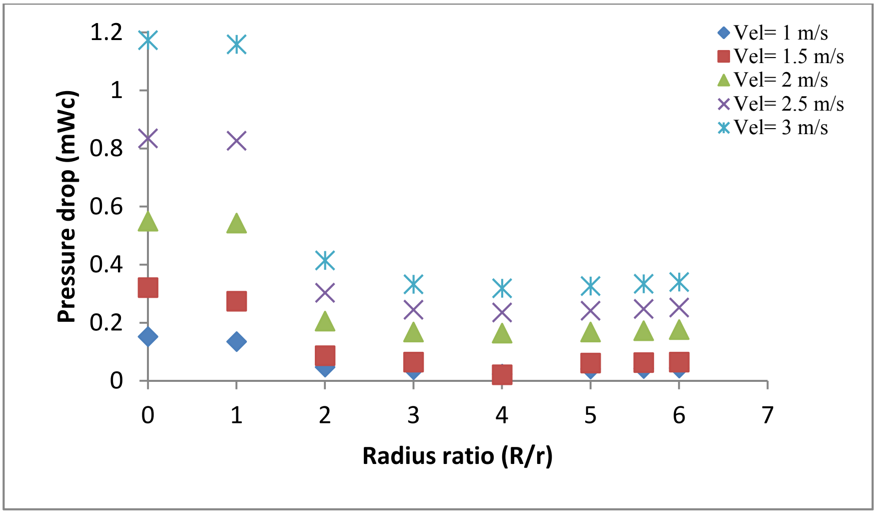

Figure 6: Pressure drop across the bend for different R/r values at various velocities (Cw=65%, D=50mm).

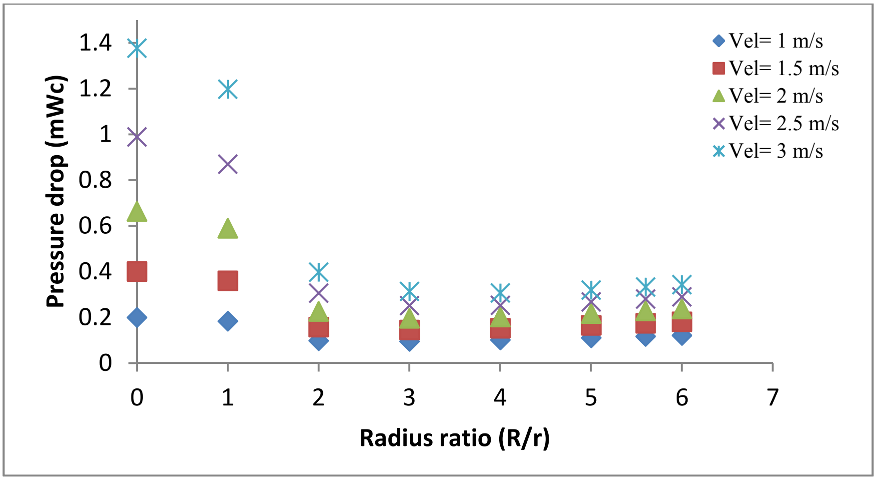

Figure 7: Pressure drop across bend for different R/r values at various velocities (Cw=68%, D=50mm).

In figs 4 and 5 as per the criterion of Hanks and Pratt [12] the values of pressure drops obtained at Cw=60% and 62%, are the values purely related to turbulent flow. From these figures it can be clearly seen that the optimum radius ratio, which gives minimum pressure drop at all velocities in the range of 1 m/s to 3m/s, is 4.0. Further in fig. 6, as per the criterion of Hanks and Pratt [12] the values of pressure drop up to the velocity of 1.5 m/s are corresponding to a laminar regime but at higher velocities the flow is in turbulent regime. From fig. 6 for the values corresponding to the laminar regime, the optimum bend is of radius ratio 3.0, which gives minimum pressure drop, whereas for turbulent flow regime again the optimum bend geometry is of radius ratio 4.0. In fig. 7 all the pressure drops correspond to the values of a laminar regime at all velocities from 1 m/s to 3m/s. From fig. 7, the minimum pressure drop corresponds to the radius ratio of 3.0.

It is observed from these figures that for a sharp corner bend (R/r=0) and R/r=1.0 bend, the values are nearly same and then there is a sudden drop in the pressure value at the highest velocity for R/r=2.0. Beyond this value of radius ratio, the reduction is marginal up to R/r=4.0 for turbulent flows and up to R/r=3.0 for laminar flows and then the values further increase marginally. The sudden drop could be attributed to suppression of separation towards the exit due to increase in radius ratio and also reduction in the intensity of secondary flows. Beyond R/r=4.0 in turbulent flows and R/r=3.0 in laminar flows, the increase in pressure drop could be attributed to the increase in the length of the bend.

These conclusions are somewhat in variance with those of Csizmadia and Hõs [9]. According to them the optimum radius ratio (R/D) was 2.0, which in fact as per our notations is R/r=1.The reason for the deviation of the results may be due to the differences in the choice of turbulence model and range of parameters. However, in the present study, the CFD methodology has been first thoroughly validated using the experimental data. Thus, the present results are more relevant to HCSD pipelines of coal ash slurries.

5 Conclusions

The following conclusions can be drawn out of the work done:

- The Bingham plastic model is the appropriate model for the high concentration slurry ($C_w \geq 60\%$) flow through pipe bends.
- k-ω SST turbulence model gives fairly accurate pressure drop values in the turbulent regime at all $C_w \geq 60\%$, for velocities up to 3m/s for the flow through pipe bends.
- For the flow of high concentration coal ash slurries ($C_w \geq 60\%$) bend with radius ratio 4.0 is the optimum bend which causes minimum pressure drop across it in turbulent regime, whereas for laminar regime flow in the HCSD systems, bend with radius ratio 3.0 is the optimum bend which gives minimum pressure drop across it.

References

[1] Elliot, D. E., Hydraulic transport of coal ash at high concentration, *Proceedings of Hydrotransport-1*. Cranfield, Bedford, U.K.: British Hydromechanics Research Association, pp. 25–56, 1970.

[2] Wilson, K. C., A dense phase option for coarse coal pipelining. *J. Pipelines* **5**, pp. 251–257, 1982.

[3] Bunn, T.F. and Chambers, A.J. and Goh, C.M., Rheology of some fly ash slurries, In: *International Coal Engineering Conference* (1990: Sydney, N.S.W.). 1990 International Coal Engineering Conference: Preprints of Papers. Barton, ACT: Institution of Engineers, Australia, pp. 7–14, 1990.

[4] Verma, A. K., Singh, S. N., Seshadri, V., Pressure drop for the flow of high concentration solid-liquid mixture across 90° horizontal conventional circular pipe bend, *Indian Journal of Engineering & Material Sciences*, **Vol. 13**, pp. 477–483, December 2006.

[5] Ito, H., Pressure losses in smooth pipe bends, Trans. ASME, *J. of Basic Engg.*, **Vol. 82**, pp. 131–143, 1960.

[6] Mishra, R., Patil, S.A., Singh, S.N., Seshadri, V., Flow characteristics of diverging-converging bends using CFD, *IE(I) Journal-CV*, **Vol. 82**, pp. 186–192, February 2002.

[7] Mishra Rakesh, Singh, S.N., Seshadri, V., Pressure drop across conventional and diverging-converging pipe bends in the flow of multi-sized particulate slurries, *Indian Journal of Engineering & Materials Sciences*, **Vol. 5**, pp. 9–14, February 1998.

[8] Kumar, U., Singh, S. N., Seshadri, V., Pressure losses in rough 90° bends of different radii of curvature, *Proceeding of 26th National Conference on FMFP*, IIT Kharagpur, 15–17 December 1999.

[9] Csizmadia, P. and Hõs, Csaba, CFD-based estimation and experiments on the loss coefficient for Bingham and power-law fluids through diffusers and elbows, *Computers & Fluids*, *Elsevier*, **Vol. 99**, pp. 116–123, 22 July 2014.

[10] Fluent, Fluent 6.3 user's guide. Lebanon: New Hampshire (USA) Fluent Inc., 2006.

[11] Dewan, A., k-ε and other two equations models (Chapter 6), *Tackling turbulent flows in engineering*, Springer Heidelberg, Dordrecht, London, New York, pp. 59–80, 2011.

[12] Hanks, R.W. and Pratt, D.R., On the flow of Bingham plastic slurries in pipes and between parallel plates, Soc. Petroleum Engineers J., **Vol. 7**, pp. 342–346, 1967.