Powder Technology 283 (2015) 593–606

Contents lists available at ScienceDirect

Powder Technology

journal homepage: www.elsevier.com/locate/powtec

Numerical investigation of mass loading effects on elbow erosion

Carlos Antonio Ribeiro Duarte, Francisco José de Souza *, Vinicius Fagundes dos Santos

School of Mechanical Engineering, Federal University of Uberlândia, Av. João Naves de Ávila, 2121 Bloco 5P, 38400-902 Uberlândia, Minas Gerais, Brazil

ARTICLE INFO

Article history:
Received 9 December 2014
Received in revised form 1 June 2015
Accepted 3 June 2015
Available online 11 June 2015

Keywords:
Elbow erosion
Mass loading
Four-way coupling
Inter-particle collisions
Cushioning effect

ABSTRACT

Wear due to particles is often the key factor for pipeline failure. In this work, the effects of different sand particle concentrations on the erosion of an elbow pipe are investigated numerically. In order to assess the quality of the numerical predictions of the erosion rate, experimental data were first used to validate the erosion and restitution models at low concentration. The input parameters for the empirical erosion correlation were obtained from accurate CFD models for the gas–solid flow within the bend. One, two and four-way couplings were evaluated at different mass loadings. In general, it was found that even at low to moderate mass loadings, the effects of inter-particle collisions on the penetration ratio cannot be neglected. Another important finding is that the maximum penetration ratio gradually diminishes as the mass loading increases. As counterintuitive as it may appear, this phenomenon has actually been observed in experiments and is named cushioning effect. Based on the analysis of the simulation results, it can be concluded that a layer of particles builds up adjacent to the elbow wall, protecting it from direct particle collisions. Conversely, the inter-particle collisions damp the particle impact to the surface, therefore reducing the penetration peak.

1. Introduction

Abrasive wear is a commonly faced issue in many industrial processes. When small, solid particles carried by a fluid stream hit a surface, they can cause severe damage to it. This type of surface damage is classified as erosive wear, and is frequently observed in process industries.

Erosion is a particularly serious problem in piping systems, especially in the conveying of oil and gas. Its intensity is closely related to poor process efficiency and reduced component lifetime. Conveying systems usually require special attention regarding piping fittings, which normally cause abrupt changes in flow direction. Elbows, tees and cross plugs are some examples of vulnerable conveying system components in erosive environments. Erosion in these parts can cause result in fluid leakage, contamination and production downtime, among other problems. Particularly, in pneumatic conveying systems, the high velocities required for transporting particulate materials represent a potential issue. Erosive wear is probably the main reason why the industry is often reluctant to install pneumatic conveying systems, particularly when abrasive materials must be handled [1]. On the other hand, erosion has interesting practical uses and advantages, such as in manufacturing processes which employ waterjet cutting, drilling and surface cleaning.

Saving maintenance time and resources are factors that drove industry and research centers to focus on developing tools for erosion prediction. Accurate tools to predict erosion rates and eroded material lifetime can help designers propose solutions to mitigate the consequences of the wear process, or at least anticipate where erosion is most likely to occur. However, developing a method that quantitatively describes the erosion process is not trivial, because of the great number of variables associated to the physical mechanism. Also, a detailed analysis of both eroded and erodent materials as well as the fluid behavior is necessary.

Progress in understanding the erosion due to particles has been achieved by the utilization of CFD models that can accurately simulate the fluid flow and particle motion through pipelines and bends [2–4]. Once the impact velocity and angle of the particles colliding against a surface are accurately calculated, it is possible to apply empirical correlations to quantify the erosion rate or the material mass loss.

There have been a number of numerical investigations on bend erosion, such as [5–8]. Although in general good agreement with experiments was obtained in these previous works, none of them investigated the effect of the particle mass loading. Essentially, very low mass loading was simulated, under which the effects of the particles on the carrying fluid and interparticle collisions were assumed not to be relevant.

The purpose of this work is to carry out an investigation of the mass loading effects on the erosion of a ninety-degree-elbow. Several empirical correlations and mechanistic models developed to calculate erosion in elbows for gas/solid and gas/liquid/solid flows exist [9–11], but recent studies from Pereira et al. [4] showed that the Oka model [12] appears to be the most accurate, robust approach, as it is based on measurable properties of both eroded and erodent materials. For the purpose of validating the CFD model, numerical results are compared to the experiments by Mazumder et al. [13]. Based on four-way-

---

* Corresponding author. Tel.: +55 34 3239 4040 615.
E-mail address: fjsouza@mecanica.ufu.br (F.J. de Souza).

http://dx.doi.org/10.1016/j.powtec.2015.06.021

coupled simulations of the gas–solid flow in the same geometry, a detailed analysis of the mass loading influence on particle motion and penetration rate was carried out. Unfortunately, a limited number of studies [14,15] have been conducted to understand the erosion rate as a function of mass loading and did not provide detailed information about particle–particle and particle–fluid interactions and their effects on erosive wear. In the present work, the finite-volume, unstructured code UNSCYFL3D, was used to solve the gas flow combined with a Lagrangian point–particle model for the particulate phase (Eulerian–Lagrangian approach). The two-layer *k*–epsilon model was used to account for turbulence effects. Interestingly, the results show that even at mass loadings as low as 0.25, the effects of inter-particle collisions on the flow and on the penetration ratio cannot be neglected. Another important finding is that the penetration ratio actually diminishes as the mass loading increases. As counterintuitive as it may appear, this phenomenon has actually been observed in experiments (cushioning effect). A physical mechanism for such an effect is proposed based on numerical analyses of the coupled gas–solid flow.

2. Mathematical models

The Euler–Lagrange approach is employed in this investigation. The modeling of both phases is described below.

2.1. Gas phase model

A Reynolds-Averaged Navier–Stokes (RANS) approach is adopted in this investigation. For a general, steady-state flow, the above-mentioned equations can be written in tensor notation as:

$$\frac{\partial(\rho u_i)}{\partial(x_i)} = 0 \tag{1}$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu+\mu_t)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right] + Su_{ip} + \rho g_i. \tag{2}$$

The numerical solution of the conservation equations for the momentum and turbulence is accomplished by the computational code UNSCYFL3D [16]. This in-house tool is based on the finite volume method in unstructured three-dimensional grids. The SIMPLE (semi-implicit method for pressure-linked equations) algorithm is used to couple the velocity and pressure fields. The collocated arrangement is used for all variables, with the conventional Rhie–Chow interpolation scheme for the computation of the mass flow rate through each element face. The discretization procedure described above generates a linear system of equations for each variable at each element center. The biconjugate gradient and the algebraic multigrid (AMG) methods are used to efficiently solve the linear system resulting from the discretization of the conserved and turbulence properties. The main advantage of this approach is that it does not depend on the element shape, as the data structure is based on element faces. For storing the coefficients of the linear systems for the velocity components, pressure correction and turbulence variables, the CSR (compressed sparse row) format is used. More details of the solution method can be found in [17].

In all the simulations carried out in this work only the steady-state solution for fluid was sought. The second-order upwind scheme was employed for the advective term, whereas the centered differencing scheme was used for the diffusive terms of the momentum equations and turbulence model equations.

The standard *k*–epsilon model is the most widely known and extensively used two-equation eddy viscosity model [18]. It was originally developed to improve the mixing-length model and to avoid the algebraic prescription of the turbulence length scale in complex flows. Transport equations are solved for two scalar properties of turbulence, the turbulence kinetic energy, *k*, and its dissipation rate, epsilon:

$$\frac{\partial(\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j} + P - \rho\epsilon\right] \tag{3}$$

$$\frac{\partial(\rho u_j \epsilon)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j} + C_{\epsilon 1}\frac{\epsilon}{k}P - C_{\epsilon 2}\rho\frac{\epsilon^2}{k}\right] \tag{4}$$

where *P* is the production term, given by:

$$P = (\mu_t+\mu)\left[\left(\frac{\partial u_i}{\partial u_j}+\frac{\partial u_j}{\partial u_i}\right)\frac{\partial u_i}{\partial u_j}\right]. \tag{5}$$

The eddy-viscosity in the standard *k*–epsilon model is defined as a function of the turbulent kinetic energy and the turbulent dissipation rate as:

$$\mu_{t,standard} = C_\mu \rho \frac{k^2}{\epsilon}. \tag{6}$$

Although widely used, the standard *k*–epsilon displays some weaknesses, such as the assumption that the flow is fully turbulent. To circumvent this issue, the 2-layer *k*–epsilon model was employed, as it can handle well both the core flow and the near wall region. Essentially, it consists in solving the standard model for the turbulent flow region and a one-equation model for the region affected by the viscosity. In the one-equation *k*–epsilon model, the conservation equation for *k* is retained, whereas epsilon is computed from:

$$\epsilon = \frac{k^{3/2}}{l_\epsilon}. \tag{7}$$

The length scale that appears in Eq. (7) is computed from:

$$l_\epsilon = yC_l\left(1 - e^{-Re_y/A_\epsilon}\right). \tag{8}$$

In Eq. (8), Re_y is the turbulent Reynolds number, defined as:

$$Re_y = \frac{\rho y \sqrt{k}}{\mu} \tag{9}$$

where y is the distance from the wall to the element centers. This number is the demarcation of the two regions, fully turbulent if $Re_y > Re_y^*$, $Re_y^* = 200$ and viscosity-affected, $Re_y < 200$. For the one-equation model, the turbulent viscosity is computed from:

$$\mu_{t,2layer} = \rho C_\mu l_\mu \sqrt{k}. \tag{10}$$

The length scale in the equation above is computed as below:

$$l_\mu = yC_l\left(1 - e^{-Re_y/A_\mu}\right). \tag{11}$$

In UNSCYFL3D, both the standard *k*–epsilon and the one-equation model described above are solved over the whole domain, and the solutions for the turbulent viscosity and the turbulence kinetic energy dissipation rate provided by both models are smoothly blended:

$$\mu_t = \lambda_\epsilon \mu_{t,standard} + (1-\lambda_\epsilon)\mu_{t,2layer}. \tag{12}$$