Large eddy simulation of particle-laden flow in a duct with a 90° bend

DO Njobuenwu and M Fairweather
Institute of Particle Science and Engineering, School of Process, Environmental and Materials Engineering, University of Leeds, Leeds LS2 9JT, UK
Email: d.o.njobuenwu@leeds.ac.uk

Abstract. Large eddy simulation (LES) of particle-laden turbulent flow is studied for a square duct with a 90° bend and a radius of curvature of 1.5 times the duct width, and for a Reynolds number based on the bulk flow velocity of 100,000. A Lagrangian particle tracking technique is used to study the motion of particles experiencing drag, shear lift, buoyancy and gravitational forces in the flow. LES predictions capture important physical aspects of these flows known to occur in practice, unlike alternative Reynolds-averaged Navier-Stokes (RANS) approaches, such as flow separation in the boundary layers around the bend entrance on the concave wall of the bend, and around the bend exit on the convex wall. The LES predicted flow and particle statistics are generally in good agreement with both experimental data used for validation purposes and RANS solutions, with r.m.s. fluctuating velocity predictions from the LES in particular being superior to values derived using the RANS technique.

1. Introduction
Pneumatic conveying as a means of transporting solid materials is used extensively in industry. In practice, conveying systems consist of straight lengths of pipe, bends, valves, fittings and associated changes in the cross-sectional area experienced by a flow. Bends, comprising concave and convex walls, are employed to change the direction of a flow, and to allow more compact piping networks. The presence of these walls is responsible for many complex phenomena that occur when transporting particles, including high pressure drop, product segregation and degradation, the formation and dispersion of solid particle ropes, and wall erosion. A number of experimental and numerical modelling approaches have been used to understand flows in ducts with bends. However, there remains an interest in developing a better understanding of the behaviour of such flows, and this work is aimed at providing a numerical approach that can be adopted for optimisation of the design, operation and maintenance of these transport systems, and to provide improved understanding. Reynolds-averaged Navier-Stokes (RANS) modelling coupled with Lagrangian particle tracking (LPT) has been applied previously to study flows in square ducts with 90° bends, e.g. Njobuenwu et al. (2011). In contrast, large eddy simulation (LES) has not as yet been applied to such flows, although it has been used to predict particle flows in straight ducts, e.g. Yao & Fairweather (2010), and in pipe bends with circular cross-sections, e.g. Rutten et al. (2001). Due to the practical importance of these flows, and the fact that LES is recognised as an effective tool for predicting industrial flows in complex geometries, extending LES to examine particulate flows in 90° bends with non-circular cross-sections is of value in improving our ability to predict the behaviour of such flows.
In this study, a combined LES-LPT approach to simulating particulate flows in square ducts with bends is applied to a duct with a radius of curvature of 1.5 times the duct width. The latter value is small enough to produce flow separation on the outer and inner walls of the bend (Kobt et al. 1988, Rutten et al. 2001). However, earlier RANS results (Njobuenwu et al. 2011) and the experimental data (Kuan et al. 2007) used herein for comparison purposes do not show flow separation. The present investigation was therefore also undertaken to assess the performance of LES in predicting separation-prone bend flows, as well as allowing comparison of the results with previous RANS predictions (Njobuenwu et al. 2011).

2. Mathematical model

In LES, the governing equations of motion are separated into resolved (large-scale) and sub-grid (small-scale) fields by a spatial filtering operation. The present work applied a top hat filter, as this fits naturally into a finite-volume formulation, to the continuity and Navier-Stokes equations to obtain the LES equations. The sub-grid scale (SGS) stress arising from the filtering operation was modelled using the Germano dynamic model (Germano et al. 1991), which represents the SGS stress as the product of a SGS viscosity and the resolved part of the strain tensor, and is based on the possibility of allowing different values of the Smagorinsky constant at different filter levels. Test-filtering was performed in all space directions, with no averaging of the computed model parameter field.

Computations were performed using the computer program BOFFIN, developed by Jones and co-workers (Jones et al. 2002, di Mare & Jones 2003), which has been validated extensively, e.g. Yao & Fairweather (2010). The code implements an implicit finite-volume incompressible flow solver using Cartesian co-ordinates and a general curvilinear co-ordinate system, with a co-located variable storage arrangement. Because of this arrangement, fourth-order pressure smoothing is applied to prevent spurious oscillations in the pressure field. Time advancement is performed using an implicit Gear method for all transport terms, and the overall procedure is second-order accurate in space and time. The time step is chosen by requiring that the maximum Courant number lies between 0.1 and 0.3, enforced for reasons of accuracy. Further details of the mathematical model employed, and the numerical solution algorithm and its implementation, may be found elsewhere (Jones et al. 2002, di Mare & Jones 2003, Yao & Fairweather, 2010).

The trajectories of the solid rigid particles as they interact with the flow field and duct walls were tracked by numerical integration of the equation of motion for particulate flow. In line with experimental measurements (Kuan et al. 2007) considered further below, it was assumed that the particles were spherical, rigid and inert, and that the particle suspension was dilute, hence the effect of particle-particle interactions and particle momentum transfer to the fluid were considered negligible. Drag, gravitational, buoyancy and shear lift forces acting on the particles, as well as particle-wall collisions, were considered. A particle collides with a wall when its centre is one radius from the wall, and it loses a fraction of its momentum before being introduced back into the bulk flow. The change in particle momentum due to particle-wall collisions was modelled using experimental measurements of restitution coefficients (Grant & Tabakoff 1975), with wall roughness modelled using a stochastic approach (Sommerfeld & Huber 1999). Here, the particle impact angle is assumed to be composed of the particle trajectory angle to a smooth wall, and a stochastic contribution due to wall roughness. A roughness angle of 5.3°, corresponding to 100 μm glass particles impacting Plexiglas, was employed (Sommerfeld & Huber 1999). The LPT algorithm used has been verified and validated elsewhere, with details given in Njobuenwu et al. (2011). Time-averaged flow field variables were computed from running averages during the computations for both the gas and particle phases. The total number of particles within that domain was sufficient to ensure statistical independence of the results.

A sketch of the computational domain is shown in figure 1(a). The streamwise direction is represented by the (z,θ,x) directions in the straight duct section upstream of the bend, through the bend, and in the downstream section of the duct, respectively, with the wall-normal direction as (x,r,z) in the corresponding sections, and the third spanwise direction corresponding to y throughout the computational domain. A curvilinear, boundary-fitted coordinate system was employed for the 90°
bend consisting of a total of approximately 1.5 million nodes with a non-uniform mesh distribution along the three coordinate directions. The number of nodes in the wall-normal, spanwise and streamwise directions was 121×61×200 respectively, with a symmetry plane applied along the vertical plane bisecting the duct. The number of nodes given in the spanwise direction is therefore for half of the width of the duct only, with a full representation of the duct requiring twice as many nodes in this direction. Expansion of the mesh within the duct cross-section was performed using a power law function. Along the streamwise direction, the geometry was split into three parts: an upstream section, a bend section, and a downstream section. The mesh contracted from the inlet towards the bend in the upstream section, was uniformly dense in the curved bend section, and expanded again in the section downstream of the bend. At the wall boundaries, the standard law of the wall approach for a smooth wall was applied, while a convective boundary condition was imposed at the outflow boundary. The inlet boundary conditions were generated using a separate inflow turbulence generator based on digital filters (Klein et al. 2003). This technique generates turbulence structures, correlated in time and space, with specified turbulence length and time scales, and was applied together with the time-averaged inlet profiles used as the basis of previous RANS calculations (Njobuenwu et al. 2011).

3. Results and discussion
To assess the LES predictions, results are compared with an experimental study (Kuan et al. 2003, 2007) and the previous RANS study (Njobuenwu et al. 2011) noted above. Kuan et al. (2003, 2007) obtained measurements of gas flows containing poly-dispersed particles with diameters between 4 and 160 μm in a horizontal-to-vertical 90° bend with the ratio of the radius of curvature (R) to duct diameter (D), R/D=1.5. The authors considered un-laden air flowing at Reₐ=100,000, with Reₐ=WₐD/μ, and poly-dispersed particles with a density of 2500 kg m⁻³. Measurements were made for the mean and fluctuating gas and particle velocities in both the streamwise and transverse directions within the bend and straight duct sections. Njobuenwu et al. (2011) and Kuan et al. (2003, 2007) independently performed RANS studies of the same flow using second-moment turbulence closures.

In agreement with previous RANS solutions (Njobuenwu et al. 2011) and experimental data (Kuan et al. 2003, 2007), the LES captured the bulk features that characterise turbulent flows in curved ducts. One of these features is that the curvature of the bend causes a non-uniform pressure and velocity distribution around the bend, in contrast to the symmetric profiles found in the straight duct sections away from the bend, as shown in figures 1(b) and (c). However, in contrast to the earlier RANS results (Njobuenwu et al. 2011), flow separation was observed in the LES predictions around the outer concave wall at the bend entrance, with the flow reattaching again shortly downstream. The velocity vectors in figure 1(d) show this flow separation and reattachment zone which can be attributed to the strong adverse pressure gradient near the beginning of the bend that drives the flow towards the convex wall, as shown in figure 1(c). This fast moving fluid is subsequently displaced back towards the concave wall as it transverses through the bend section. Downstream of the first half of the bend, the flow is again seen, figure 1(e), to separate from the inner convex wall, and to reattach just before the exit to the bend. It should be noted that neither of these separations was observed in the experimental study of Kuan et al. (2003, 2007). However, the tendency for separation to occur on both the outer and inner walls of a bend as a function of R/D has been reported in both experimental (Tunstall and Harvey 1968) and numerical (Rutten et al. 2001, Kuan et al. 2007, Kotb et al. 1988) studies. These studies show that the flow is more likely to separate as the ratio R/D tends to unity (corresponding to an “L” bend), and less likely to separate as this ratio tends to infinity (corresponding to a straight duct). On the average, separation was shown to disappear on the outer wall for bends with R/D≥3.0, and to vanish on the inner wall for R/D≥3.5. This work therefore points to the occurrence of such zones in the R/D=1.5 case considered, despite them not being observed by Kuan et al. (2003, 2007).
Figure 1. Along the plane of symmetry of the duct: (a) schematic of the duct geometry, (b) instantaneous streamwise velocity \( (w/w_b) \), (c) streamwise mean velocity vectors, and close up of vectors (d) near concave wall at bend entrance, and (e) near convex wall near bend exit.

There is good agreement between LES predictions and experimental data, as well as with previous RANS predictions, for the mean and fluctuating velocity profiles in the duct. Both predictive techniques are therefore seen to be in general accord with the streamwise mean gas velocity data within the bend, as shown in figure 2(a), and in the straight sections of the duct, as shown in figure 2(b). In these and subsequent figures, the ordinate used is the dimensionless distance from the concave wall, \( r^*=(r-r_o)/(r_i-r_o) \), with velocities for the locations given on the abscissa of a magnitude according to the scale at the top of each figure. Also, despite the prediction of flow separation by the LES and the impact this has near the outer wall at \( \theta=0^\circ \) and the inner wall at \( \theta\geq60^\circ \), the LES results are in general superior to those from the RANS, especially in the prediction of the gas velocity fluctuations, as shown in figures 2(c) and (d). It is also worthy of note that RANS techniques generally have difficulties in predicting unsteady separation from continuous surfaces.

LES predictions of the associated streamwise mean and r.m.s. fluctuating particle velocities are given, respectively, in figures 3(a)-(b) and 3(c)-(d), with results generally in line with data, and the LES-based predictions again superior to the RANS results, apart from close to the outer wall of the bend at \( \theta=0^\circ \) due to the flow separation noted in figures 1(b)-(d). Other slight discrepancies in the near-wall regions are also apparent amongst the two sets of predictions, although at the inner wall the LES clearly performs better than the RANS in predicting the near-wall mean particle velocity. Similar comparisons for the straight duct sections before and after the bend, and for the mean and fluctuating transverse velocities for both the gas and particle phases (not shown), confirm the superiority of the LES-based predictions.
Figure 2. Streamwise mean gas velocity profiles ($\overline{u}/w_b$) plotted against normal distance from concave wall $r^*$ in (a) bend and (b) straight duct sections, and r.m.s. of gas velocity fluctuations ($u_{rms}/w_b$) in (c) bend and (d) straight duct sections (o data, — LES, ----- RANS).

Figure 3. Streamwise mean particle velocity profiles ($\overline{v}_p/w_b$) plotted against normal distance from the concave wall $r^*$ in (a) bend and (b) straight duct sections, and r.m.s. of particle velocity fluctuations ($v_{p,rms}/w_b$) in (c) bend and (d) straight duct sections (o data, — LES, ----- RANS).
4. Conclusions

Large eddy simulations have been performed for particulate flows in square cross-sectioned ducts with 90° bends at $R/D=1.5$ and $Re_b=100k$. Time-averaged velocities from the LES computations for both the gas and particle phases have been compared with experimental measurements (Kuan et al. 2003, 2007) and numerical results from RANS calculations (Njobuenwu et al. 2011), with the LES predicted flow and particle statistics generally in good agreement with experimental data and superior to RANS solutions. LES is also found to predict flow separation on the outer concave wall at the bend entrance, and a larger separation on the inner convex bend wall that starts at approximately 60° around the bend. The bend studied had a radius of curvature small enough, from previous work, to produce separation at the outer and inner walls of the bend, although these separations were not noted in the experimental study of Kuan et al. (2003, 2007). The issue of whether the prediction by the LES of flow separation within the bend flow is realistic is therefore worthy of further study through both simulation and experiment. In particular, an experimental study that focuses on the flow regimes and bend geometries where such phenomena do and do not occur would be extremely useful.

Acknowledgments

The authors wish to acknowledge the financial support from the Commonwealth Scholarship Commission in the United Kingdom.

References

DI MARE, L. & JONES, W. P. 2003, LES of turbulent flow past a swept fence, *Int. J. Heat Fluid Flow* **24**, 606-15.

GERMANO, M., PIOMELLI, U., MOIN, P. & CABOT, W.H. 1991, A dynamic subgrid-scale eddy viscosity model, *Phys. Fluids A* **3**, 1760-5.

GRANT, G. & TABAKOFF, W. 1975, Erosion prediction in turbomachinery resulting from environmental solid particles, *J. Aircraft* **12**, 471-8.

JONES, W.P., DI MARE, F. & MARQUIS, A.J. 2002, LES-BOFFIN: Users guide. Dept. Mech. Eng., Imperial College of Science, Technology and Medicine.

KLEIN, M., SADIKI, A. & JANICKA, J. 2003, A digital filter based generation of inflow data for spatially developing direct numerical or large eddy simulations, *J. Comput. Phys.* **186**, 652-65.

KOTB, N.A.E., MOKHTARZADEH-DEHGHAN, M.R. & WARD-SMITH, A.J. 1988, Numerical study of laminar and turbulent flows in a two-dimensional bend with or without a guide vane, *Int. J. Numer. Methods Eng.* **26**, 245-62.

KUAN, B., WILLIAM, Y. & SOLNORDAL, 2003, CFD simulation and experimental validation of dilute particulate turbulent flow in 90° duct bend, In *Proc. 3rd Int. Conf. on CFD in Minerals and Process Industries*, pp. 531-6, CSIRO.

KUAN, B., YANG, W. & SCHWARZ, M. P. 2007, Dilute gas-solid two-phase flows in a curved 90° duct bend: CFD simulation with experimental validation, *Chem. Eng. Sci.* **62**, 2068-88.

NJOBUENWU, D.O., FAIRWEATHER, M. & YAO, J. 2011, Prediction of turbulent gas-solid flow in a duct with a 90° bend using an Eulerian-Lagrangian approach, *AIChE J.*, dx.doi.org/10.1002/aic.12572.

RUTTEN, F., MEINKE, M. & SCHRODER, W. 2001, Large-eddy simulations of 90 degrees pipe bend flows, *J. Turbul.* **2**, 1-14.

SOMMERFELD, M. & HUBER, N. 1999, Experimental analysis and modelling of particle-wall collisions, *Int. J. Multiphase Flow* **25**, 1457-89.

TUNSTALL, M.J. & HARVEY, J.K. 1968, On the effect of a sharp bend in a fully developed turbulent pipe-flow, *J. Fluid Mech.* **34**, 595-608.

YAO, J. & FAIRWEATHER, M. 2010, Inertial particle resuspension in a turbulent, square duct flow, *Phys. Fluids* **22**, 033303.