Two–phase flow predictions of the turbulent flow in a combustion chamber including particle–particle interactions

Michael Breuer and Michael Alletto
Department of Fluid Mechanics, Helmut–Schmidt–University Hamburg, Holstenhofweg 85, D–22043 Hamburg, Germany
E-mail: breuer/alletto@hsu-hh.de

Abstract. Relying on large–eddy simulation (LES) and an efficient algorithm to track a huge number of Lagrangian particles through turbulent flow fields in general complex 3D domains, the flow in a pipe and a model combustion chamber is tackled. The influence of particle–fluid (two–way coupling) as well as particle–particle interactions (four–way coupling) is investigated. The latter is modeled based on deterministic collision detection. First, the LES results of a particle–laden vertical pipe flow with a specular wall and a mass loading of 110% are evaluated based on DNS data from the literature. Second, the predicted LES data of a ring combustion chamber at two different mass loadings (22% and 110%) are analyzed and compared with experimental measurements.

1. Introduction

Turbulent dispersed multiphase flows play an important role in many technical as well as medical applications such as cyclones, combustion chamber or inhalators. Up to now, most of the numerical investigations were based on the Reynolds–Averaged Navier–Stokes equations (RANS) combined with statistical turbulence models (see, e.g., review of Sommerfeld et al. (2008)). Since in the majority of flows considered complex phenomena such as curved streamlines, secondary flow regions and transition are involved, the continuous phase is not predicted reliably by RANS. Consequently, the prediction of particle motion including dispersion and deposition using a Lagrangian random–walk eddy–interaction model or similar methods to track the particles in the flow field was often found to be not accurate enough.

Therefore, during the last years huge effort was directed towards a methodology which is much more appropriate for the simulation of the continuous phase of turbulent flows including complex flow phenomena, i.e., the large–eddy simulation (LES) technique. However, the computational resources required for LES are very large. Predicting the particulate phase by a Lagrangian particle tracking further increases the computational requirements. Thus a highly efficient tracking algorithm is a must. Furthermore, since dense two–phase flows are of particular interest, physical effects such as the fluid–particle interaction (two–way coupling) as well as particle–particle collisions (four–way coupling) have to be taken into account leading to an enormous computational burden if not tackled reasonably. Both issues are briefly considered in this paper. A four–way coupled simulation technique in complex geometries using LES and a Lagrangian
particle tracking with deterministic collision detection results. The procedure is validated for high mass loadings based on two-phase pipe flow using DNS data (Vreman, 2007) for comparison. Furthermore, it is applied to the particle-laden flow in a model combustion chamber, for which experimental data are available by Boree et al. (2001).

2. Computational Methodology

The method relies on the Euler–Lagrange approach, where the continuous phase is simulated in the Eulerian frame of reference using LES, whereas for the particulate phase a huge number of individual particles are tracked throughout the computational domain in a Lagrangian frame of reference.

2.1. Continuous Phase

For the prediction of the fluid phase based on LES the filtered Navier–Stokes equations are taken into account. They are discretized by a standard 3-D finite-volume method for arbitrary non-orthogonal and block-structured grids within the LES code LESOCC (Breuer, 1998, 2002). The spatial discretization of all fluxes is based on central differences of second-order accuracy. A low-storage multi-stage Runge–Kutta method (second-order accurate) is applied for time-marching. In order to ensure the coupling of pressure and velocity fields on non-staggered grids, the momentum interpolation technique is used. For modeling the non-resolvable subgrid scales, two different models are applied, namely the well-known Smagorinsky (1963) model with Van Driest damping near solid walls ($C_s = 0.065$) and the dynamic approach with a Smagorinsky base model proposed by Germano et al. (1991) and modified by Lilly (1992). The code is highly vectorized and additionally parallelized by domain decomposition using MPI.

2.2. General Procedure for the Dispersed Phase

The particulate phase is computed based on Newton’s second law taking only drag, lift, gravity and buoyancy into account. Owing to high density ratios ($\rho_p/\rho_f \gg 1$) considered, all other contributions can be neglected. The particles are assumed to be rigid and spherical. Thus the drag force on the particle is based on the Stokes flow around a sphere improved by a correction factor defined by Schiller & Naumann (1933) in order to extent the validity of the relation for the drag coefficient towards higher particle Reynolds numbers ($0 < Re_p \leq 800$). The lift force acting on particles in shear flows is modeled by a formulation provided by McLaughlin (1991) for unbounded flows. If the particle lags behind the fluid, the resulting lift force experienced by the particle drives it in the direction of positive velocity gradient. The presence of a wall further affects the lift force, which can be accounted for by an extended model. However, since the implementation of this model in a general-purpose curvilinear code is not trivial, it was presently not taken into account.

The numerical procedure for the solution of the governing equations is as follows. The ordinary differential equation for the particle velocity is integrated by a fourth-order Runge–Kutta scheme in physical space. In order to enable efficient tracking of millions of particles on a block-structured curvilinear grid, the second integration required to get the new location of the particle is done in the computational space ($\xi, \eta, \zeta$; $c$-space) rather than in the physical space ($x, y, z$; $p$-space), see Breuer et al. (2006, 2007). In $c$-space schemes the particle traces are integrated in a coordinate system, in which the curvilinear physical space grid is orthonormal. Point location within the $c$-space grid is as trivial as for a Cartesian grid, since there is an explicit relationship between the $c$-space coordinates of a particle location and the grid cell containing it. Thus the $c$-space method has the advantage that no search of the particle’s new position is required as for algorithms working in $p$-space and typically spending the majority of CPU time for global and local search algorithms. With respect to the application of high-performance
computers, it is highly beneficial that c-space methods do not require such CPU time-killing search algorithms, which are moreover difficult to parallelize and vectorize.

For tracking the particles through the continuous flow field, the local instantaneous (filtered) fluid velocities at the particle position are required. These are interpolated using a Taylor series expansion around the cell center next to the particle (Marchioli et al., 2007). This interpolation scheme was shown to possess a weaker filtering effect on the fluid velocity than a trilinear interpolation used before (Breuer et al., 2006, 2007) leading to better results for particles with small relaxation times.

If for tiny particles their relaxation time is of the same order as the smallest fluid time scales, the unresolved scales in LES become important for the particle motion. To consider the effect of the subgrid scales a simple stochastic model by Pozorski & Sourabh (2009) is applied. It requires the estimation of the subgrid-scale kinetic energy carried out with the help of the scale similarity approach of Bardina et al. (1980).

Restricting the interaction of fluid and particles to these effects is called one-way coupling and only valid for volume fractions below about $10^{-6}$ (Sommerfeld et al., 2008). Above this limit the influence of the particles on the fluid motion has to be taken into account, leading to a two-way coupled simulation. For that purpose the particle-source-in-cell (PISC) method by Crowe et al. (1977) is used accounting for the exchange of momentum between the dispersed and the continuous phase. This leads to a modified filtered Navier–Stokes equation with an additional source term representing the forces exerted by the particles onto the fluid. A smooth source term distribution is achieved by trilinear distributing the contribution of the particle to the 8 cell centers surrounding the particle. When the volume fraction becomes larger than $10^{-3}$ (Sommerfeld et al., 2008) the regime of dilute dispersed two-phase flows is left. Then particle–particle collisions play an important role and have to be taken into account which is denoted four-way coupling.

2.3. Deterministic Collision Model

As mentioned above, for four-way coupling the method how particle–particle collisions are handled is a very critical issue. Nevertheless, a deterministic collision model is taken into account instead of a stochastic collision model often used before (see, e.g., Sommerfeld (2001)). Solely binary collisions are considered here. Following the technique of uncoupling developed by Bird (1976), the calculation of particle trajectories is split into two stages:

1. particles are moved based on the equation of motion without inter-particle interactions,
2. the occurrence of collisions during the first stage is examined for all particles. If a collision is found, the velocities of the collision pair are replaced by the post-collision ones without changing their position which is also advantageous for parallelization.

The collision handling itself is carried out in two steps (A and B):

(A) In the first step likely collision partners are identified. Since for small time steps only collisions between neighboring particles are likely, substantial computational savings are achieved by dividing the computational domain into virtual cells. Choosing the cell size in such a way that the particles per cell are sufficiently low, the cost of checking collisions is reduced from the order $O(N_p^2)$ to $O(N_p)$, which is crucial for large numbers of particles, e.g. $N_p = O(10^7-10^8)$, at high mass loadings. Furthermore, to avoid overlapping cells or the necessity to take the 26 surrounding cells into account during the first step, the search and collision detection procedure is carried out a second time with slightly different cell sizes.

(B) The second step solely takes the particles in one virtual cell into account. Following a suggestion of Chen et al. (1998) the algorithm relies on the assumption of constant velocity within a time step, which is reasonable for the small time step sizes applied in LES. Based on the assumption of linear displacements during a time step, it is possible to detect the collision
of two particles by purely kinematic conditions, i.e., (i) the two particles have to approach each other and (ii) their minimum separation within a time step has to be less than the sum of their radii. If a collision is detected, the velocities of the colliding particles are changed according to a hard sphere inelastic collision. The measure of inelasticity is generally expressed by a restitution coefficient currently set to unity. For the results presented here, friction is not taken into account for the collisions. Thus, solely the velocity components in collision–normal direction are changed by the collision. For this purpose, the Cartesian velocity components are transformed to the collision–normal direction prior to collision and re-transformed after collision.

For the interaction of particles with rigid walls, currently two different boundary conditions can be applied: (i) The particle sticks at the wall and is consequently removed from the computational domain. That is a reasonable assumption in case of a wetted wall as used in Breuer et al. (2006, 2007) for the deposition predictions in a bend or an idealized mouth region. Here the particles adhere to the wetted surface upon contact. (ii) Alternatively in the cases considered here, the particle rebounds fully elastically from a smooth specular wall. This implies that the sign of the velocity component normal to the wall is inverted and all other components are kept. For further physically reasonable situations including inelastic wall collisions taking friction into account or considering rough walls, appropriate models are currently developed.

Simulations with high mass loadings were carried out with this enhanced Euler–Lagrange algorithm including particle–particle collisions based on the deterministic model and working on curvilinear block-structured grids. Nevertheless, the CPU time required for the collision detection and handling took less than 10% of the total time demonstrating that the code is highly efficient.

3. Description of the Test Cases

After the procedure had shown good agreement with the experimental data of Benson et al. (2005) in a simple channel geometry with smooth walls (Breuer & Alletto, 2011), a turbulent pipe flow and the cold flow in a combustion chamber where taken as intermediate test cases to validate the code in fully curvilinear geometries.

3.1. Pipe Flow

The present particle–laden turbulent pipe flow results are compared with recently published DNS data of Vreman (2007). The Reynolds number based on the bulk velocity $U_B$ and the pipe radius $R = 10$ mm is set to $Re = 2006$ ($Re_r = 140$). Two particle sizes of $d_p = 60 \mu m$ and $d_p = 90 \mu m$ are considered, corresponding to a Stokes number $St_{60}^+ = \tau_p \alpha_r^2 / \mu = 82.3$ or $St_{60}^- = \tau_p U_B / R = 8.4$ and to $St_{90}^+ = 185$ or $St_{90}^- = 18.9$, respectively. Heavy particles with a ratio of particle to fluid density of $\rho_p / \rho_f = 2100$ corresponding to glass beads are assumed. The mass loading of the vertical gas–solid pipe flow is $\Phi = 110\%$ for both particle sizes. The gravity vector is pointing in the mean flow direction. A specular reflection boundary condition is applied for the particles at the (smooth) pipe wall. Since Vreman (2007) concluded that wall roughness is the main reason for the discrepancies observed between his DNS data and the experiments, the uncertainty involved by stochastic wall roughness models is excluded in the current study. Thus, the present results are solely compared with the simulations of Vreman (2007) applying the same boundary conditions for the particles. The computational domain is $12R$ long in streamwise direction. Periodic boundary conditions are applied at the streamwise boundaries applying a forcing term in the momentum equation which guarantees a fixed mass flux through the pipe. At the wall the no-slip boundary condition is used. The grid consists of 970,515 cells. In wall–normal direction the first computational node has a distance from the wall of $\Delta y/R = 0.001$, which corresponds to $y^+ = 0.14$. Thus the viscous sublayer is resolved. The time step is made dimensionless with $U_B$ and $R$ and is set to $\Delta t = 0.0028$. 
3.2. Combustion Chamber Flow

In order to test the code using a more challenging case with practical relevance, a cold flow in a model combustion chamber without swirl is considered next. The geometry, depicted in Fig. 1, is chosen to match the configuration investigated by Boree et al. (2001). They gained detailed particle and fluid data in a configuration typical for combustion devices. This experimental setup is an excellent test case to get insight into the complicated physical mechanisms governing dense multiphase flows.

The particle–laden air flow with a mean velocity $U_{\text{jet}} = 3.1 \text{ m/s}$ enters the chamber through a circular pipe ($R_{\text{pipe}} = 10 \text{ mm}$) located on the chamber axis (Fig. 1). The Reynolds number based on the pipe radius and the mean inflow velocity is $Re = 2006$. The particles with a density ratio of $\rho_p/\rho_f = 2100$ have diameters varying in the range of $d_p = 22 \text{ to } 100 \mu\text{m}$. The mass loading is either $\Phi = 22\%$ or $110\%$. Clean air enters through an annular ring ($R_i/R_{\text{pipe}} = 7.5$, $R_a/R_{\text{pipe}} = 15$) with a mean velocity $U_e/U_{\text{jet}} = 1.775$. The gravitational acceleration acts in the main flow direction. The inflow conditions are provided by two additional LES predictions using pipe and annular ring flows with periodic boundary conditions and the same cross-sectional grid. For all computed cases (one-, two- and four–way) with the same mass loading, the same inflow data are used to reduce the considerations solely to the chamber flow. At the outflow a convective boundary condition and the no–slip condition at the walls are prescribed. The chamber with a length of $L/R_{\text{pipe}} = 90$ is discretized by an O–type grid consisting of 13 blocks and about $1.3 \times 10^7$ cells. To additionally save CPU time, particles hitting the wall or passing the plane normal to $z/R_{\text{pipe}} = 45$ were removed from the domain.

4. Results and Discussion

4.1. Pipe Flow

Figure 2 shows the comparison of the results obtained by the present particle–laden LES and the DNS of Vreman (2007). All quantities are made dimensionless by the centerline velocity $U_c$ of the unladen flow. An excellent agreement between the results of both methodologies (LES vs. DNS) is found. That proves to be true for the unladen case and the four–way coupled case for both particle diameters. Figure 2(a) shows the mean streamwise fluid velocity. It is obvious that the mean streamwise flow is drastically altered due to the feedback of the particles with a diameter of $d_p = 60 \mu\text{m}$. The velocity profile shows a more pronounced maximum in this case. Overall, it is closer to a fully laminar velocity profile than the unladen flow or the two–phase flow containing the bigger particles, which hardly influence the mean streamwise velocity distribution. The partial relaminarization especially for $d_p = 60 \mu\text{m}$ is more clearly visible by looking at the fluid velocity fluctuations in streamwise (Fig. 2(c)) and wall–normal (Fig. 2(e)) direction. The particles attenuate the fluid velocity fluctuations by almost an order of magnitude. This phenomenon is found in both, the present LES and the reference DNS data.
The Reynolds shear stress, which is an indicator of the momentum transfer between the mean and the transverse direction, has almost vanished (Fig. 2(g)). The results of Vreman (2007) are not shown for this quantity because it was difficult to extract the small values from the figures. The particle mean velocity shown in Fig. 2(b) reflects the behavior of the mean fluid except in the near–wall region, where particle–particle collisions lead to a slip velocity between the particles and the wall. Again, the LES and DNS data are in close agreement. Particle–particle collisions enhance the streamwise fluctuations (Fig. 2(d)) in the near–wall region and in the pipe center and are reduced in the buffer region with respect to the one–way coupled case. The wall–normal fluctuations and the Reynolds shear stresses are strongly augmented due to the particle–particle interactions (Fig. 2(f) and 2(h); Note that for these quantities DNS data are only available for $d_p = 90 \mu m$). Additionally, a decoupling of the fluid and particle second moments can be observed if particle–particle collisions are taken into account. In the one–way coupled case the second moments of the fluid and the particles are similar in the order of magnitude and the general distribution, while in the four–way coupled case they disagree completely. In conclusion, the four–way coupling leads to strongly decreased Reynolds stresses of the continuous phase, while the dispersed phase shows an enormous increase of all stresses, especially of the wall–normal component and of the Reynolds shear stress.

4.2. Combustion Chamber Flow

In the following the results of three different LES predictions applying one–way, two–way and four–way coupling are discussed and compared with the experimental data of Boree et al. (2001) for mass loadings of $\Phi = 22\%$ and 110%. For that purpose the instantaneous flow and particle fields were averaged in time over a time interval of about 6.5 flow–through times of the chamber for $\Phi = 22\%$ and 4.5 flow–through times for $\Phi = 110\%$ and additionally in circumferential direction. For the particle phase the size class of $d_p = 50 \mu m$ was chosen for the evaluation, which is equivalent to the number averaged diameter of the distribution (Boree et al., 2001).

To provide a first overview of the general flow structure, in Fig. 3 the streamlines for the four–way coupled simulations, both $\Phi = 22\%$ and 110%, are shown for the averaged flow. Since there are no big differences between the one-, two- and four–way coupled flow for the low mass loading and the two- and four–way coupled flow for the high mass loading, these two figures provide an illustration how the momentum additionally injected by the particles changes the flow in the model combustion chamber. As visible in Fig. 3(a) due to the central jet a second counter-clockwise rotating recirculation region is formed inside the wake located behind the bluff body. In the case of the high mass loading (Fig. 3(b)) this recirculation region is elongated compared to the low mass loading case (Fig. 3(a)). Additionally, it is shifted slightly away from the axis, since the central jet penetrates through the recirculation region in contrast to the low mass loading case. Thus, the two stagnation points visible in Fig. 3(a) disappear for $\Phi = 110\%$ depicted in Fig. 3(b). The red lines stand for the measurements planes, where the statistics of the simulations are compared with the measurements of Boree et al. (2001).

4.2.1. Low Mass Flow Rate $\Phi = 22\%$

Figure 4(a) depicts the axial velocity of the flow along the axis. As visible, the flow field of the continuous phase is influenced by the particulate phase already at this low mass flow rate. The jet developing at the exit of the pipe is stopped rapidly in the recirculating flow forming a first stagnation point $S_1$ on the axis (see Figs. 1 and 3(a)). In the unladen case $S_1$ is found at about $z/R_{pipe} = 12.5$, whereas for the four–way coupled case this point is shifted downstream to about $z/R_{pipe} = 14.5$. The same trend even more pronounced can be observed in the experimental data. Contrarily, the deviation between the two–way and four–way coupled case is minor, indicating that the role of particle–particle collisions is of less importance for the low mass loading. However, the influence of the particles on the fluid is of major interest. A
Figure 2: Results of the pipe flow at $Re = 2006$ and $\Phi = 110\%$: Mean streamwise velocity: (a) fluid, (b) particle, fluctuations in streamwise direction: (c) fluid, (d) particle, fluctuations in wall–normal direction: (e) fluid, (f) particle, Reynolds shear stress, (f) fluid, (g) particle.
second stagnation point $S_2$ of the fluid flow is located at the end of the recirculation bubble. Here the differences between the three predicted cases is minor. Figure 4(b) shows the axial evolution of the axial velocity fluctuations. No significant difference between the one–, two– and four–way coupled predictions can be found. The maximum is observed in accordance with the experiments in the region of the maximum axial velocity gradient and is slightly underpredicted and shifted upstream. Figure 4(c) illustrates the radial velocity fluctuations along the axis. These Reynolds stresses are overpredicted in the region of the recirculation region but the maximum is found similar to the experiments at the end of the recirculation region near $S_2$. It is worth noting that, if the influence of the particles on the fluid is taken into account, the maximum is diminished with respect to the one–way coupled case and furthermore shifted downstream. The maximum deviation between the one–way and the four–way coupled cases is observed quite far away from the entrance, i.e., near the second stagnation point $S_2$.

Figures 4(d) to 4(l) display profiles of the mean streamwise velocity and the streamwise and radial velocity fluctuations at three different axial positions ($z/R_{pipe} = 8, 16$ and 24) for the continuous flow. The following observations can be made: 1. Except on the axis, minor deviations are found between the three different simulations at $\Phi = 22\%$. 2. The differences in the measurements of the mean streamwise continuous flow for the single and two–phase flow observed at the outer radii are hard to explain. 3. Except for this region, in general a good agreement between predicted and measured data is found.

Figure 5 shows the same statistics as Fig. 4 but for the particulate phase. As for the continuous phase, noticeable differences between the one–way and the two– or four–way coupled predictions are found only on the axis. Similar observations as for the continuous phase can be made and a good agreement is found between the simulation and the experiment. In contrast to the radial fluid velocity fluctuations on the axis (Fig. 4(c)), there are only minor discrepancies between simulation and experiment for the radial particle velocity fluctuations (Fig. 5(c)) in the region $15 < z/R < 25$, i.e., in the domain where the recirculation is secluded by the outer annular flow. Differences between the simulations and the experiment are found in the locations of the peak of the axial velocity fluctuations (Fig. 5(b)) and in the profile of the axial velocity fluctuations at $z/R = 16$ (Fig. 5(h)). In accordance with the experiment the maximum of the streamwise velocity fluctuations in the predictions coincide with the maximum of the streamwise velocity gradient (see Fig. 5(a)). However, in the simulations this point is slightly shifted upstream with respect to the data of Boree et al. (2001). The plane at $z/R = 16$ is located at the maximum measured axial velocity fluctuations on the axis, while at this axial position the simulated fluctuations are already decreasing (Fig. 5(b)). This explains the difference between the measured and simulated profiles in Fig. 5(h). Nevertheless, overall a good agreement is
Figure 4: Continuous phase of the combustion chamber flow at $\Phi = 22\%$: Mean streamwise velocity at: (a) $z/R =$ along the axis, (d) $z/R = 8$, (g) $z/R = 16$, (j) $z/R = 24$; streamwise velocity fluctuations at: (b) $z/R =$ along the axis, (e) $z/R = 8$, (h) $z/R = 16$, (k) $z/R = 24$; radial velocity fluctuations at: (c) $z/R =$ along the axis, (f) $z/R = 8$, (i) $z/R = 16$, (l) $z/R = 24$.

found, especially for the lower–order statistics such as the mean streamwise particle velocity (see Fig. 5(d), (g), (j)).

4.2.2 High Mass Flow Rate $\Phi = 110\%$

In Figs. 6 and 7 the same profiles are depicted for the high mass loading of $\Phi = 110\%$, which
Figure 5: Dispersed phase of the combustion chamber flow at $\Phi = 22\%$: Mean streamwise velocity at: (a) $z/R =$ along the axis, (d) $z/R = 8$, (g) $z/R = 16$, (j) $z/R = 24$; streamwise velocity fluctuations at: (b) $z/R =$ along the axis, (e) $z/R = 8$, (h) $z/R = 16$, (k) $z/R = 24$; radial velocity fluctuations at: (c) $z/R =$ along the axis, (f) $z/R = 8$, (i) $z/R = 16$, (l) $z/R = 24$.

still corresponds to a maximum local volume ratio of merely about 0.05% at the inlet. Solely the cross-section $z/R_{pipe} = 24$ is replaced by $z/R_{pipe} = 20$, since no experimental data are available for the former. Here the deviations between one-way coupling on the one hand and two-way or four-way coupling on the other hand become more pronounced as visible e.g. in Figs. 6(a) to 6(c) and Figs. 7(a) to 7(c). The injected particles induce so much momentum that the stagnation points on the axis disappear and the jet penetrates through the recirculation zone (Fig. 6(a)). Furthermore, the particles cause a downstream shift of the maximum axial
fluid velocity fluctuations (Fig. 6(b)). Similar to the experimental data, the peak value is not reduced with respect to the single phase flow. Differences in the magnitude compared with the experimental data are observed right before the region where the annular flow mixes with the central jet. Looking at the mean axial velocity distribution along the axis (Fig. 6(a)), it is obvious that in the experiment the jet keeps a constant velocity on the axis over a longer distance from the inlet than in the simulation. However, both reach the minimum at almost the same point. This leads to a steeper experimental velocity profile in the region where the predicted axial velocity fluctuations do not fit well to the experiments and explains the differences in the magnitudes detected. In contrast to the axial velocity fluctuations, the radial velocity fluctuations along the axis depicted in Fig. 6(c) are strongly attenuated by the presence of the particles up to the mixing zone between the annular flow and the central jet. This region coincides with the maximum of the radial velocity fluctuations in the two–phase case. Good agreement is found between the two–way or four–way coupled simulations and the two–phase measurements.

Figures 6(d) to 6(l) display profiles of the mean streamwise velocity and the streamwise and radial velocity fluctuations of the continuous flow at three different axial positions ($z/R_{pipe} = 8, 16$ and $20$). Here, the following conclusions can be drawn: 1. The influence of the particles is noticeable at larger distances from the axis than for the lower mass loading. This sounds quite reasonable since the farther penetrating jet also modifies the recirculation region. 2. Good agreement is found between the numerical results and the experiments except for the velocity fluctuations in the mixing region between the jet and the annular flow, i.e. $15 \leq z/R \leq 20$, (Fig. 6(h), 6(i), 6(k) and 6(l)). The reason for these discrepancies are still under investigation. Figure 7 shows the particle statistics along the axis and on the measurement planes $z/R = 8, 16$ and $20$. Similar observation as for the continuous phase can be made for most of the particle statistics depicted. Interesting is that the particle streamwise velocity fluctuations along the axis (Fig. 7(b)) are attenuated with respect to the one–way coupled prediction, which is in contrast to the observations made for the fluid fluctuations. Major difference to the experiments are solely restricted to the streamwise velocity fluctuations near the axis of the planes $z/R = 16$ and $z/R = 20$. Again, the overall agreement is found to be satisfactory, especially for the mean streamwise particle velocity (see Fig. 7(d), (g), (j)), but this time also for the radial velocity fluctuations (see Fig. 7(f), (i), (l)). Furthermore, it is worth noting that the deviations found in the simulations applying two–way and four–way coupling are found to be minor, which is astonishing at least for this high mass loading case.

5. Conclusions

A methodology for the simulation of dispersed two–phase turbulent flows at high mass loadings is presented. Key features are the simulation of the continuous flow based on LES, an efficient particle tracking scheme applicable on curvilinear block–structured grids, and a deterministic but nevertheless non time–consuming inter–particle collision algorithm. Close agreement of the present LES results with reference DNS data are found for a four–way coupled particle–laden vertical pipe flow assuming specular pipe walls. An overall good agreement is found between two- and four–way coupled simulation and the experiment of Boree et al. (2001) at both measured mass loadings (i.e., $\Phi = 22\%$ and $110\%$). The one–way coupled assumption is only justified at the lower mass loading since the particles substantial alter the flow structure at the higher mass loading. Minor differences are found between two- and four–way coupled simulation at both mass loadings, which indicates a inferior importance of the inter–particle collisions in comparison to the particle–fluid interaction on both fluid and particle statistics at the mass loadings considered. Summarizing, the tool developed reliably predicts the complex physical phenomena involved in turbulent dispersed two–phase flows and the objective is to extend the methodology to even
Figure 6: Continuous phase of the combustion chamber flow at Φ = 110%: Mean streamwise velocity at: (a) $z/R$ = along the axis, (d) $z/R = 8$, (g) $z/R = 16$, (j) $z/R = 20$, streamwise velocity fluctuations at: (b) $z/R$ = along the axis, (e) $z/R = 8$, (h) $z/R = 16$, (k) $z/R = 20$; radial velocity fluctuations at: (c) $z/R$ = along the axis, (f) $z/R = 8$, (i) $z/R = 16$, (l) $z/R = 20$.

higher mass loadings.

Acknowledgments

The time–consuming computations were carried out on the national supercomputer NEC SX-9 at the High Performance Computing Center Stuttgart (grant no.: PARTICLE / pfs 12855),
Figure 7: Dispersed phase of the combustion chamber flow at $\Phi = 110\%$: Mean streamwise velocity at: (a) $z/R$ = along the axis, (d) $z/R = 8$, (g) $z/R = 16$, (j) $z/R = 20$; streamwise velocity fluctuations at: (b) $z/R$ = along the axis, (e) $z/R = 8$, (h) $z/R = 16$, (k) $z/R = 20$; radial velocity fluctuations at: (c) $z/R$ = along the axis, (f) $z/R = 8$, (i) $z/R = 16$, (l) $z/R = 20$.

which is gratefully acknowledged.

References

BARDINA, J., FERZIGER, J.H. & REYNOLDS, W.C. 1980 Improved subgrid–scale models for large–eddy simulations. AIAA Paper, 80–1357.
Benson, M., Tanaka, T. & Eaton, J.K. 2005 Effects of wall roughness on particle velocities in a turbulent channel flow. Trans. ASME J. Fluids Eng. 150, 250–256.

Bird, G.A. 1976 Molecular Gas Dynamics. Claredon.

Boree, J., Ishima, T. & Flour, I. 2001 The effects of mass loading and inter–particle collision on the development of polydispersed two-phase flow downstream of a confined bluff body. J. Fluid Mech. 443, 129–165.

Breuer, M. 1998 Large-eddy simulation of the subcritical flow past a circular cylinder: Numerical and modeling aspects. Int. J. for Numerical Methods in Fluids 28 (9), 1281–1302.

Breuer, M. 2002 Direkte Numerische Simulation und Large–Eddy Simulation turbulenter Strömungen auf Hochleistungsrechnern. Shaker Verlag, Aachen.

Breuer, M. & Alletto, M. 2011 Efficient simulation of particle-laden turbulent flows with high mass loadings using LES. In Seventh Int. Symp. on Turbulence and Shear Flow Phenomena (TSFP-7), July 28–31, 2011. Ottawa, Canada.

Breuer, M., Baytekin, H. T. & Matida, E. A. 2006 Prediction of aerosol deposition in 90 degrees bends using LES and an efficient Lagrangian tracking method. J. of Aerosol Science 37 (11), 1407–1428.

Breuer, M., Matida, E.A. & Delgado, A. 2007 Prediction of aerosol drug deposition using an Eulerian–Lagrangian method based on LES. In Int. Conf. on Multiphase Flow, July 9–13, 2007. Leipzig, Germany.

Chen, M., Kontomaris, K. & McLaughlin, J.B. 1998 Direct numerical simulation of droplet collisions in a turbulent channel flow, part I: Collision algorithm. Int. J. Multiphase Flow 24, 1079–1103.

Crowe, C.T., Sharma, M.P. & Stock, D.E. 1977 The Particle-Source-In-Cell (PSI-CELL) model for gas-droplet flows. Trans. ASME J. Fluids Eng. 99, 325–322.

Germano, M., Piomelli, U., Moin, P. & Cabot, W.H. 1991 A dynamic subgrid–scale eddy viscosity model. Phys. Fluids A 3, 1760–1765.

Lilly, D.K. 1992 A proposed modification of the Germano subgrid–scale closure method. Phys. Fluids A 4, 633–635.

Marchioli, C., Armenio, V. & Soldati, A. 2007 Simple and accurate scheme for fluid velocity interpolation for Eulerian–Lagrangian computation of dispersed flow in 3D curvilinear grids. Computers & Fluids 36, 1187–1198.

McLaughlin, J.B. 1991 Inertial migration of a small sphere in linear shear flows. J. Fluid Mech. 224, 261–274.

Pozorski, J. & Sourabh, V. 2009 Filtered particle tracking in isotropic turbulence and stochastic modeling of subgrid–scale dispersion. Int. J. Multiphase Flow 35, 118–128.

Schiller, L. & Naumann, A. 1933 A drag coefficient correlation. VDI Zeitschrift 77, 318–320.

Smagorinsky, J. 1963 General circulation experiments with the primitive equations, I, The basic experiment. Monthly Weather Review 91, 99–165.

Sommerfeld, M. 2001 Validation of a stochastic Lagrangian modelling approach for inter–particle collisions in homogeneous isotropic turbulence. Int. J. Multiphase Flow 27, 1829–1858.

Sommerfeld, M., Von Wachem, B. & Oliemans, R. (eds.) 2008 Best practice guidelines for computational fluid dynamics of dispersed multiphase flows. In SIAMUF, Swedish Industrial Association for Multiphase Flows, ERCOFTAC.

Vreman, A. W. 2007 Turbulence characteristics of particle-laden pipe flow. J. Fluid Mech. 584, 235–279.