THE INFLUENCE OF THE FLUID ACCELERATION TERM ON THE SIMULATION OF A PARTICLE-LADEN COMPRESSIBLE JET WITH SHOCK WAVES

Flavia C. Miranda[1], Arne Heinrich [1] & Jorn Sesterhenn[1]
[1]*Institute of Fluid Mechanics and Engineering Acoustics, Berlin University of Technology, Germany*

Abstract A particle-laden compressible round jet is studied using direct numerical simulation and an Eulerian-Lagrangian point-particle approach is used to model fluid and particle phase. For this purpose, the particle motion is calculated by Newton's law and a two-way coupling mechanism describes the interaction between the two-phases. Our specific objective is to investigate the importance of the fluid acceleration term on the particle equations when strong gradients are present. Expectedly, the results indicate the importance of this term in regions where the shock waves occur. In these regions, the force due to the fluid acceleration shows to be at least three times bigger than the drag force due to steady fluid motion. We performed two simulations: one with the fluid acceleration presented in the particle equation and another where this term is omitted. The results show that the particles computed without this term substantially lag the changes of the properties of the fluid, than the simulation where this term is present.

INTRODUCTION

The study of turbulent flows with particles is important for numerous industrial application and natural processes. As examples on the industrial field think of pneumatic transport of powders, air-brushes, fluidized beds and cyclone separators. In the nature it occurs in sand storms, volcanoes, avalanches and sediment transport of sand in rivers and sea. In order to better understand the physics evolving these problems, studies of such flows are highly important and necessary.
In the context of incompressible jet with particle, a relative rich bibliography can be found. Recently, Fan & Cen [1] investigated investigated numerically the particle dispersion in an axisymmetric incompressible jet. They found out that the particles tend to accumulate preferentially in the regions that present stream wise velocity larger than the mean velocity. But the field of compressible flows is still poorly explored. We have e.g. the work from Vitturi &al. [3]. They studied pyroclastic density currents using Eulerian models to simulated the particle phase. In this context, we propose the study of the particle-laden compressible round jet. We focused our attention in investigation of the importance of the forces due to the fluid pressure gradient and viscous stresses on the particle equation in order to ensure proper modelling of the particles.

NUMERICAL METHOD

The fluid-phase is calculated employing the compressible Navier-Stokes equation based on a characteristic-type formulation of Sesterhenn, [2]. The spatial discretization was done using a sixth order compact finite-difference scheme with spectral-like resolution of wavenumber and for the time integration a low storage Runge-Kutta of fourth order was employed.
In the Eulerian-Lagrangian approach, the fluid is calculated in an Eulerian mesh and the particle in a Lagrangian fashion. A source term on the momentum equation is responsible to make the coupling between the two phases.
In this work, the particle are assumed to be rigid spheres and the fluid is considered dilute, so the particle-particle interaction is negligible. We considered that the forces acting on a particle are the drag force and the force due to the fluid acceleration:

$$\frac{\mathrm{d}\mathbf{U}}{\mathrm{d}t} = \frac{1}{\rho_p}\left(-\mathbf{F}_d + \frac{\mathrm{D}\mathbf{u}_i}{\mathrm{D}t}\right), \tag{1}$$

where the drag force $\mathbf{F}_d$ is computed as $\mathbf{F}_d = \frac{\rho_p}{\tau_p}(\mathbf{U} - \mathbf{u}_i)$, the variable $\mathbf{U}$ corresponds the vector of the particle velocity, ρ_p is the density of the particle phase, the variable $\mathbf{u}_i$ correspond to the fluid velocity interpolated to the Lagrangian points. The fluid acceleration term ($\frac{\mathrm{D}\mathbf{u}_i}{\mathrm{D}t}$) corresponds to the fluid pressure gradient and viscous stresses by means of the momentum equation. Finally, the term τ_p corresponds to the particle response time.

RESULTS

The domain employed on the simulation is $L_x = 22D$, $L_y = 15D$ and $L_z = 15D$, where D is jet diameter. The Reynolds number based on the inlet velocity is $Re_{in} = 1000$ and the Mach number for the developed jet is $Ma_{in} = 1.5$. The resolution employed is $512 \times 256 \times 256$. The starting location of each particle at the jet outlet is set randomly and the particles are injected with the same velocity as the fluid. The volume fraction for the simulations is approximately 10^{-4} and the Stokes number based on the jet diameter and inlet velocity ($St_D = \tau_p * U_{in}/D$) is 0.3.

Figure 1 shows the density gradient of the fluid phase and the particles position. The color of the particles represents the relation between the horizontal component of the force due the fluid acceleration and horizontal component of the drag force, $\frac{\mathrm{D}(u_x)_i}{\mathrm{D}t}/(F_d)_x$. The red particles represent regions where the fluid acceleration term is at least three times bigger than the drag term. As expected, the regions where the shock waves occur correspond the to the regions where the red particles are located. Ocasionally red particles also occur in regions where the eddies are detached from the shear layer.

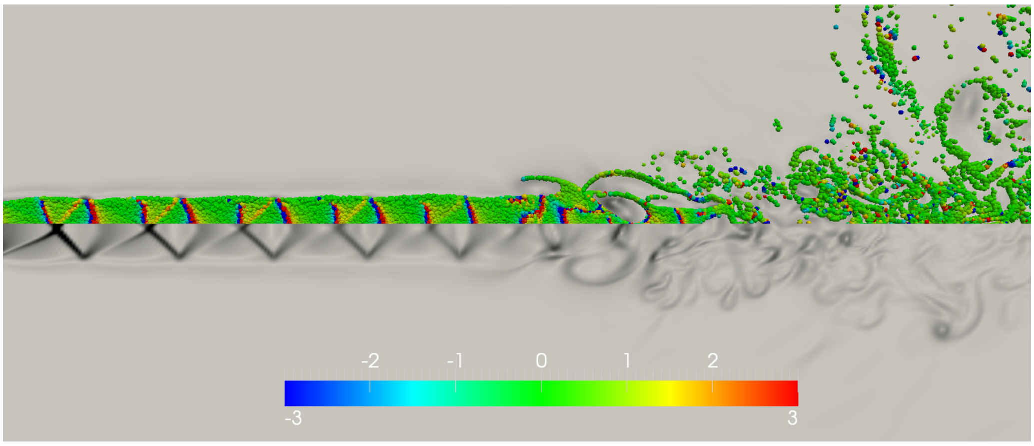
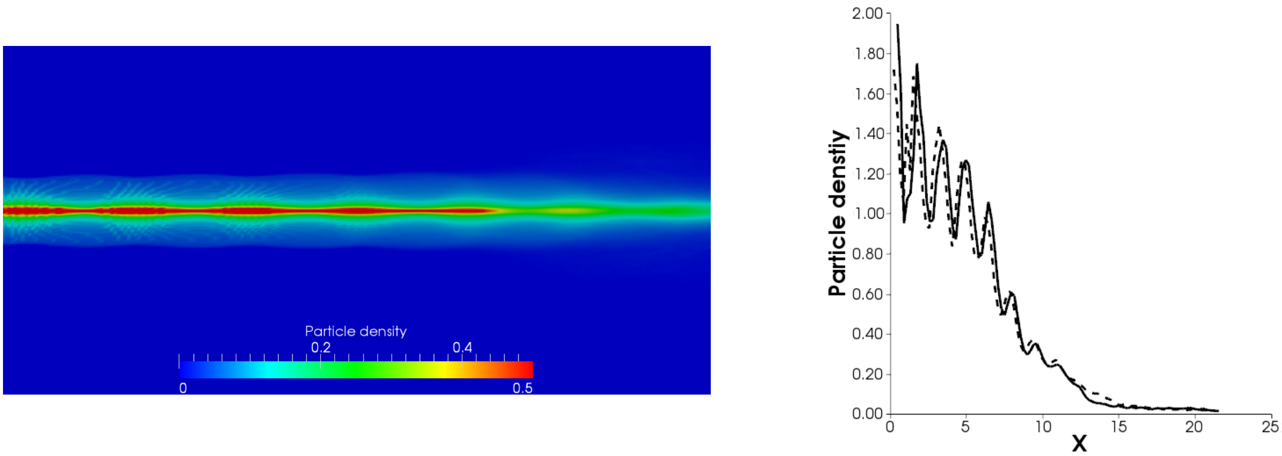


Figure 1. Fluid density gradient and particle position. The particle color represents the relation $\frac{\mathrm{D}(u_x)_i}{\mathrm{D}t}/(F_d)_x{}^{-1}$.

Figure 2 (a) shows the particle density for the simulation with the fluid acceleration term. It can be seen that the particle density resembles the structure of the shock cells. In the graphic 2 (b) the particle density for a central line of the jet is plotted as a function of the horizontal coordinate (x). The dashed line represents the results for the case with the fluid acceleration term computed and the continuous line for the case where this term is ignored. The difference between the two curves suggests that the particles computed without this term feel the changes on the properties of the substantially fluid later than the simulation where this term is present.



(a) (b)

Figure 2. (a) Particle density for the simulation with the fluid acceleration term, (b) Particle density for a line in the centre of the jet. Dashed line corresponds to the results computed with the fluid acceleration term and the continuous line the results computed without this term.

References

[1] D. L. Jianren Fan and K. L. Kefa Cen. Direct numerical simulation of particle-laden low reynolds number turbulent round jet. *International Journal of Multiphase Flow*, **37**:539–554, 2011.

[2] J. Sesterhenn. A characteristic-type formulation of the navier-stokes equations for high order upwind schemes. *Computers & Fluids*, **30**(1):37 – 67, 2001.

[3] M. M. Vitturi, T. E. Ongaro, A. Neri, M. V. Salvetti, and F. Beux. An immersed boundary method for compressible multiphase flows: Apllication to the dynamics of pyroclastic density currents. *Computers & Geosciences*, **11**:183–198, 2007.