EFFECTS OF GRAVITY ON SHEARED TURBULENCE LADEN WITH BUBBLES OR DROPLETS

Said Elghobashi and Juan Lasheras**
Mechanical and Aerospace Engineering Department
University of California, Irvine, California 92717

** Applied Mechanics and Engineering Sciences Department
University of California, La Jolla, California 92093

ABSTRACT

This is a new project which started in May 1996. The main objective of the experimental/numerical study is to improve the understanding of the physics of *two-way coupling* between the dispersed phase and turbulence in a prototypical turbulent shear flow – **homogeneous shear**, laden with small liquid droplets (in gas) or gaseous bubbles (in liquid). The method of direct numerical simulation (DNS) is used to solve the full three-dimensional, time-dependent Navier-Stokes equations including the terms describing the two-way coupling between the dispersed phase and the carrier flow. The results include the temporal evolution of the three-dimensional energy and dissipation spectra and the rate of energy transfer across the energy spectrum to understand the fundamental physics of turbulence modulation, especially the effects of varying the magnitude of gravitational acceleration. The mean-square displacement and diffusivity of the droplets (or bubbles) of a given size and the preferential accumulation of droplets in low vorticity regions and bubbles in high vorticity regions will be examined in detail for different magnitudes of gravitational acceleration. These numerical results which will be compared with their corresponding measured data will provide a data base from which a subgrid-scale (SGS) model can be developed and validated for use in large-eddy simulation (LES) of particle-laden shear flows. Two parallel sets of experiments will be conducted: bubbles in an immiscible liquid and droplets in air. In both experiments homogeneous shear will be imposed on the turbulent carrier flow. The instantaneous velocities of the fluid and polydispersed-size particles (droplets or bubbles) will be measured simultaneously using a two-component Phase-Doppler Particle Analyzer (PDPA). Also, the velocity statistics and energy spectra for the carrier flow will be measured.

INTRODUCTION

Turbulent, dispersed, two-phase flows occur in a wide range of manufacturing and processing applications. Often, the efficiency of the process depends directly on the degree of uniformity of dispersion of one phase into the other, e.g. the dispersion of droplets in a gas, or gaseous bubbles in a liquid. Thus, when the influence of gravity is reduced or eliminated, it is expected that the efficiency of many of these manufacturing or chemical processes could be greatly enhanced. However, at present, no existing two-phase mathematical model is accurate enough to be used for the design, operation and control of processes involving turbulent, dispersed two-phase flows. The keystone in the characterization of these flows still remains to be the modeling of the two-way interaction between the dispersed-phase and the turbulent carrier flow.

The **main objective** of our experimental/numerical study is to improve the understanding of the fundamental physics of *two-way coupling* between small particles (droplets or bubbles) and a turbulent carrier flow. This two-way coupling encompasses the dispersion of the discrete particles (droplets or bubbles) by the turbulent fluid motion and the simultaneous modulation of the turbulence structure of the carrier flow by the dispersed-phase.

Our **approach** is to conduct a well-controlled set of ground-based and reduced-gravity experiments to measure the instantaneous velocities of the carrier flow and polydispersed-size droplets (or bubbles) using a two-component Phase-Doppler Particle Analyzer (PDPA) in a homogeneous-shear-turbulence wind tunnel (or water channel) where the air (or water) is laden with small droplets (or bubbles) of a known size-distribution. Although a flow laden with polydispersed-size particles (drops or bubbles) represents an added complexity to the experimental study, we have intentionally selected these conditions for two reasons. First, these conditions closely resemble those in practical applications. Second, this flow enables us to perform a full parametric study of the particle size effects on the turbulence kinetic energy exchange between the two phases ***with a single experiment.*** The parallel numerical study employs direct numerical simulation (**DNS**) to solve, for the same flow, the full three-dimensional, time-dependent Navier-Stokes equations including the two-way coupling terms. The choice of this uniform-shear flow avoids the unnecessary complexities resulting from inhomogeneities of the mean velocity or pressure as in jets and mixing layers.

The results of our study will directly impact the development of ***subgrid scale***, SGS, models for ***large eddy simulation***, LES, of particle-laden turbulent shear flows. At present, **there is no SGS model for particle-laden flows with two-way coupling**. It is important to note that an SGS model represents a ***local spatial average of the instantaneous small scale turbulence.*** This ***instantaneous spatial average*** is quite different from what is modeled in the conventional ***time-averaged*** transport equations (e.g. in the $k - \varepsilon$ model) where there is no distinction between large- and small-scale motions, and, by definition, the instantaneous motion cannot be retrieved. Therefore, all existing closure models of particle-laden turbulent shear flows provide little or no assistance in developing SGS models for LES of the same flows. The needed information can be obtained ***only*** from well-controlled experiments and DNS studies like those considered here.

It is true that DNS is not expected to predict engineering flows in complex geometries in the next few decades. However, DNS is the ***only method that can provide new physical insights at all scales of turbulence*** . Furthermore, DNS has provided excellent agreement with measurements in wind tunnels at comparable values of R_λ [3].

The **scientific merit** of the present study is that it will enhance the understanding of the basic physics of particle-laden turbulent shear flows and especially the effects of gravity on the two-way interaction between the dispersed phase and the carrier flow. Furthermore, the newly acquired experimental and numerical results will be used to improve the current mathematical models of these flows.

This project started in May 96 and thus this paper presents only a description of the numerical method, the experimental facility and the quantities to be measured.

NUMERICAL STUDY

The three-dimensional Eulerian instantaneous velocity field for the fluid will be obtained by the method of direct numerical simulation (DNS) described by Elghobashi et al [2], Gerz et al. [5],

Elghobashi and Truesdell [3] [4] and thus only the main features are outlined here.
We consider a cubical domain with side-length L which is usually about eight times the initial integral length scale of turbulence (ℓ_o). Gravity acts downward in the negative x_3 ($or\ z$) direction. The two other coordinates x_1 ($or\ x$) and x_2 ($or\ y$) are in the horizontal plane. The fluid is incompressible and has a constant kinematic viscosity, ν. The equations governing fluid motion are the instantaneous, three-dimensional Navier-Stokes, and continuity equations which can be written in a dimensionless form as :

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j}(u_j u_i) + S x_3 \frac{\partial u_i}{\partial x_1} + S u_3 \delta_{i1} = \frac{1}{Re}\frac{\partial^2 u_i}{\partial {x_j}^2} - \frac{\partial p}{\partial x_i} - f_i \ , \quad (1)$$

$$\frac{\partial u_j}{\partial x_j} = 0 \ . \quad (2)$$

In the above equations, u_i are the deviations of the instantaneous velocity U_i from their respective reference profiles: $u_i(x,y,z,t) = U_i(x,y,z,t) - U_{ref}(z)\delta_{i1}$, where $U_{ref}(z) = z\,(dU_1/dz)_{ref}$ and $(dU_1/dz)_{ref}$ is a fixed reference velocity gradient. The nondimensional shear S is defined as $S = (L/\Delta U)(dU_1/dz)_{ref}$, where ΔU is the difference in the mean velocity at the top and bottom of the domain. The characteristic quantities used in nondimensionalizing eqs. (1)-(2) are $L, \Delta U$ and a reference density ρ_o. The dimensionless time, t, is defined as $t = (\tau\,\Delta U)/L$, where τ is the real time. The last term in eq.(1) is the force exerted on the fluid by N particles (droplets or bubbles) per unit mass of fluid, in the x_i- direction, and calculated from : $f_i = \sum_{m=1}^{N} f_{m,i}$, where $f_{m,i}$ is the instantaneous local sum, in the x_i- direction, of the forces which act on one particle m. These are represented by the first four forces on the RHS of the equation (3) of particle motion described in the next subsection. Normalization of f_i is consistent with the other terms in eq. (1). N is the instantaneous number of particles, $[N = N(x_1, x_2, x_3, t)]$, within the control volume at which eq. (1) is integrated. The direct effect of the particles presence on the continuity equation of the fluid, eq. (2), is assumed negligible since the volume fraction of the particles is less than 10^{-3}. It should also be emphasized that if the volume fraction of the particles exceeds 10^{-3}, particle-particle collision may occur, thus resulting in *four-way coupling* [1] which is not considered in the present study.

The instantaneous velocity of a spherical particle, v_i, in the x_i direction, is obtained by time integration of the following Lagrangian equation of particle motion :

$$\begin{aligned} m_p\,(dv_i/dt_p) = \ & m_p\,(u_i - v_i)/\tau_p + m_f\,(Du_i/Dt) + \tfrac{1}{2}\,m_f\,(Du_i/Dt - dv_i/dt_p) \\ + \ & 6a^2\,(\pi\,\rho\,\mu)^{1/2} \int_{t_{p_o}}^{t_p} \frac{d/d\tau(u_i - v_i)}{(t_p - \tau)^{1/2}}\,d\tau + (m_p - m_f)\,g_i \ . \end{aligned} \quad (3)$$

Equation (3) describes the balance of forces acting on the particle as it moves along its trajectory. The term on the left hand side is the inertia force acting on the particle due to its acceleration. The terms on the right side are respectively the forces due to viscous and pressure drag, fluid pressure gradient and viscous stresses, inertia of virtual mass, viscous drag due to unsteady relative acceleration (Basset) and buoyancy. As mentioned in the previous section, the negative of the sum of the first four terms on the RHS of (3) is the force exerted by one particle on its surrounding fluid. The response time τ_p is the time for momentum transfer due to drag and is given, for the case of Stokes flow, as : $\tau_p = (2a^2\rho_p)/(9\nu\rho)$.
The quantities, a, m_p, ρ_p, are, respectively, the particle radius, mass and material density. The fluid density and dynamic viscosity are ρ and μ. m_f is the mass of the fluid displaced by the particle and equals $m_p(\rho/\rho_p)$. The derivative, d/dt_p, is with respect to time following the moving particle, whereas Du_i/Dt is the total acceleration of the fluid as seen by the particle, evaluated at the particle position $\vec{x}_p$. The integration of (3), via a second order Adams-Bashforth scheme

provides the new velocity, $v_i(t)$, in the x_i- direction for each droplet as a function of time. The new position, $x_{p,i}(t_n)$, along the particle trajectory is calculated from:

$$x_{p,i}(t_n) = x_{p,i}(t_{n-1}) + \Delta t[v_i(t_n) + v_i(t_{n-1})]/2 + \Delta t[U_{ref}(x_{p,i}, t_{n-1})]\delta_{i1}, \quad (4)$$

where t_n and t_{n-1} are the times at the current and previous time-steps , and $\Delta t = t_n - t_{n-1}$. The last term in (4) is the displacement of a particle due to the imposed mean velocity $U_{ref}(x_3)$ of the carrier flow in the x_1 direction.

The fluid velocity $u_i[x_{p,i}(t)]$ at the particle location, which is needed to integrate eq. (3), is obtained by a fourth-order, two-dimensional, four-point Hermitian polynomial interpolation scheme between the adjacent Eulerian fluid velocity values. This scheme is applied in the three coordinate directions at the particle location. It should be mentioned that eq. (3) must be solved simultaneously with the Navier-Stokes equ.(1) and the continuity equ.(2) in order to evaluate the coupling terms f_i.

EXPERIMENTAL STUDY

In order to examine the complex interactions between the particles and the turbulent flow, simultaneous measurement of the velocity fields of the two phases is required. For this purpose, detailed phase Doppler velocity and size measurements will be conducted at locations downstream along the evolving turbulence using an Aerometrics phase Doppler particle analyzer (PDPA) system. Due to the relatively large concentration of particles (void fraction) required for our particle-turbulence two-way interaction study, these measurements represent a particularly challenging problem. Therefore we will to conduct each experimental study in two stages. The first stage considers the modification of decaying homogeneous (grid-like) turbulence in a uniform flow by the presence of particles of a known size distribution. The second stage concerns the more general case of homogeneous turbulence in a uniform-mean-gradient shear flow. The following sections describe the experimental set-ups for the studies of both inertial particles (droplets in air) and particles with negligible inertia (bubbles in water).

Uniform mean-velocity facilities

a) Wind tunnel

The blow-down wind tunnel to be used in the first part of experiments (water droplets in air) already exists in our laboratory at UCSD. The basic facility is described in detail by Lázaro and Lasheras [6] [7]. By extending the splitter plate an additional 4m and by building both top and bottom surfaces of the test sections out of movable belts, this facility permits the study of the decay of the particle-laden turbulent flow over extended distances. The facility is rather versatile since it allows not only for the variation of the mean velocity and rms of the air flow, but also the systematic variation of both the droplet size distribution, and the droplet volumetric flow rate.

b) Water channel

The water channel used in the second set of experiments also exists in our laboratory at UCSD. Conceptually this facility is identical to the above described wind tunnel with the exception of the bubble generation section.

Droplet and bubble generation

a) Droplet generation

A polydispersed size spray of droplets generated through an array of air blast atomizers is used in our wind tunnel to inject droplets into the shear flow and control their size and void fraction. The humidity of the carrier gas prior to the droplet injection section is always kept equal to the

equilibrium saturation value. Thus, evaporation effects in the test section are quite negligible [6] [7].
Each atomizer consists of two 22G (0.406mm inner diameter) stainless steel tubes placed together in an inverted U configuration [6]. The bent tube is fed with compressed air whereas the straight one is fed with distilled water from a pressurized tank. The high velocity air jet issuing from the bent tube impinges and breaks the capillary water jet emanating from the straight needle. This design allows, within a certain range, control over the mean diameter and droplet density. The diameter is controlled by varying the air supply pressure while the density can be regulated by adjusting the water flow rate. The atomizer is then introduced in the air stream with the compressed air jet issuing in the streamwise direction. The resulting droplet volume-size probability density function shows that 99% of the number pdf is contained within the range $2\mu m < D < 100\mu m$.
To ensure uniform particle concentration in the cross-section of the free stream, several hundred atomizers are distributed along the atomization region, forming an equidistant triangular mesh in the plane perpendicular to the streamwise direction. Every atomizer can be separated from its neighboring ones at a preset distance. The overlapping of the resulting array of sprays in the mesh results in the desired uniform free-stream droplet size and density (void fraction) distribution.
Our experimental set-up and flow conditions ensure that there is negligible droplet collision and evaporation and thus are ideal for the study of particle/turbulence interaction.

b) Bubble generation
The bubbles are created using the "novel" injection scheme. Water, saturated with CO_2 at 90 psi by a water carbonator, is expanded as a jet through small holes drilled into a grid of brass tubes. The resulting negative pressure step experienced by the carbonated water causes homogeneous nucleation to occur in the jet. Due to the rapid mixing with non-carbonated water flow in the water channel , the bubble growth is limited, resulting in a small polydispersed bubble size distribution. Bubble production is not uniform in the vertical direction due to varying pressure steps along the small brass tubing, however, the mixing induced by the jets homogenizes the bubble sizes throughout the stream. Any bubbles produced that are too large rise rapidly and are removed by the free surface just downstream of the injection location. Bubbles that accumulate on the underside of the splitter plate in the slow moving flow prior to the contraction are removed by suction.

Uniform-mean-shear flow facilities

Recently, Piccirillo [8] has successfully extended Rohr's multi- layer concept [9] to air flow. This new facility is currently used in our laboratory at UCSD to study the evolution of turbulence in a uniform-mean-shear, thermally stratified flow. The facility consists of a blow-down, open-loop, multi-layered wind tunnel. The facility is supplied by an array of ten independent blowers connected to individual diffusers and turbulent management sections, and thus allows for the generation of ten layers of air at a pre-determined velocity and temperature. The merging of the ten parallel layers results in a uniform-mean-shear flow at the beginning of the test section. This multi-layered design allows for a relatively simple way to generate a large variety of uniformly-sheared turbulent flows in which water droplets can be added to produce a flow of uniform Φ_v.
For our droplet-laden air flow turbulence experiments and for the bubble-laden water flow, both multilayer facilities will be modified by simply placing at the end section of each layer an array of atomizers or micronozzles in a similar way as is currently done in the uniform-velocity facilities described above. To minimize the wake effects resulting from the boundary layers on the plates separating each layer, a set of screens are placed downstream of the merging section. To produce uniform volumetric flow rate of droplet loading (Φ_v) throughout the whole cross-stream length of the test section, the number of individual atomizers placed in each array will be adjusted depending

on the mean air velocity in each layer.

Quantities to be measured

The polydispersed drop size distribution will be discretized into five size families. Similarly, for the bubble case, the bubble size distribution will be discretized in the same number of families. For each drop size, we will measure the three components of their velocity along the centerline (u_d,v_d,w_d). The size range between 1 μm and 2 μm will be used as markers to extract the velocity of the carrier gas (u_a,v_a,w_a). We will conduct a parametric study of the effect of Φ_v and the strain parameter $\frac{1}{U_a}\frac{dU_a}{dz}$ for a fixed droplet size distribution.
The following turbulence quantities will be extracted from the measurements.
Carrier flow (Air or Water): The turbulent stresses (normal and shear) , the production rate of turbulence kinetic energy, the streamwise dissipation rate of turbulence energy, the energy spectrum, the turbulence length scales (Taylor microscale, the integral length scale by integrating the corresponding autocorrelation coefficient).
Dispersed phase (Droplets and Bubbles): For each family D_i of the discrete size distribution we will compute: the probability density distribution function of the three components of the particle velocity, the turbulence kinetic energy of each particle family, $\overline{q^2_{D_i}} = \frac{1}{2}\left(\overline{u'^2_{D_i}} + \overline{v'^2_{D_i}} + \overline{w'^2_{D_i}}\right)$, the mean volume flow rate of each droplet size, $\Phi_{v_{D_i}}$, and the total mean volume flow rate of droplets, $\Phi_v = \sum_{i=1}^{5} \Phi_{v_{D_i}}$.

REFERENCES

[1] S.E. Elghobashi. On predicting particle-laden turbulent flows. *Appl. Sci. Res.*, 52, 1994.

[2] S.E. Elghobashi, T. Gerz, and U. Schumann. Direct simulation of turbulent homogeneous shear flow with buoyancy. *Fifth International Symposium on Turbulent Shear Flows, Cornell Univ.*, pages 227–233, 1985.

[3] S.E. Elghobashi and G.C. Truesdell. Direct simulation of particle dispersion in decaying isotropic turbulence. *J. Fluid Mech.*, 242:655–700, 1992.

[4] S.E. Elghobashi and G.C. Truesdell. On the two-way interaction between homogeneous turbulence and dispersed solid particles, part 1 : turbulence modification. *Phys. Fluids*, A5:1790–1801, 1993.

[5] T. Gerz, U. Schumann, and S. Elghobashi. Direct simulation of stably stratified homogeneous turbulent shear flows. *J. Fluid Mech.*, 200:563–594, 1989.

[6] B.J. Lazaro and J.C. Lasheras. Particle dispersion in the developing free shear layer. part 1. unforced flow. *J. Fluid Mech.*, 235:143–178, 1992.

[7] B.J. Lazaro and J.C. Lasheras. Particle dispersion in the developing free shear layer. part 2. forced flow. *J. Fluid Mech.*, 235:179–221, 1992.

[8] P.S. Piccirillo. Evolution of stratified shear turbulence. *Ph.D. Thesis*, University of California, San Diego, AMES Dept., 1993.

[9] J. J. Rohr, E. C. Itsweire, K. N. Helland, and C. W. Van Atta. An investigation of the growth of turbulence in a uniform-mean-shear flow. *J. Fluid Mech.*, 187:1, 1988.