A Numerical Method for the Calculation of the Resistivity Coefficient in the Flow Around a Sphere

R. Bedogni(1)A. Di Fazio(2)

(1) *Osservatorio Astronomico di Bologna, Via Zamboni 33, I-40126, Bologna*

(2) *Osservatorio Astronomico di Roma, Viale del Parco Mellini 84, I-00136, Roma*

© Società Italiana di Fisica

Summary. — A PPM finite difference scheme is used in a numerical program, adapted in order to evaluate the resistivity force exerted on a sphere by a fluid generally in a turbulized state. The problem has been investigated in the Eulerian limit including the numerical dissipation effects of the PPM method. The program is designed to be able to compute the resistivity force, the turbulent energy in the fluid and the main scale of turbulence in the wake. The experimental determination of the drag coefficient have been reproduced within a max error of ≈ 25 % and a mean error of ≈ 6 %. The so-called "crisis of resistivity" is actually reproduced with dip positions (Reynolds number) affected by displacement errors roughly around a few percent. This is a fairly good simulation, considering also that the Reynolds numbers that can be simulated using the PPM numerical viscosity are partly limited by the computing times and the costs induced by the chosen grid zoning. The efficiency of generation of turbulence produced by the work of the resistivity force is also studied, as well as the distribution, along the wake, of the turbulent r.m.s. pulsation and of the local main scale of turbulence.

PACS 95.30.Lz – Hydrodynamics.

PACS 47.27.Eq – Turbulence simulation and modeling.

PACS 47.15.Ki – Inviscid flow with vorticity.

1. – Introduction

The scientific problem of evaluating the resistivity force exerted on an object moving in a fluid in a turbulent regime, and sometimes even in a supersonic turbulent regime, is very fierce, and it challenges theoretical physics and engineering already from the second decade of this century. The most known applied cases of this scientific problem are the predictions of the flight speeds of airplanes and jets, the cruising speeds of ocean liners and cargo ships, the maximization of the speeds of racing cars, the design of the foils of rockets and re-entry vehicles from space, the design of the most efficient sail plans and hulls of the modern ocean racers and America's cup racers, and many others. As it can be easily understood, the main difficulties arise when the predictivity of a model is very important, and when the empirical heuristic experiments cannot solve a particular problem. Unfortunately, this is very frequent when we deal with *turbulence* and its related phenomena, understanding which in depth can be reasonably ranked as one of the nowadays hardest problems of theoretical physics.

The scientific problematic that motivated us to build a suitable instrument to calculate the resistivity force are of astrophysical nature. In fact, we work in the general line of investigation of the evolution of self-gravitating objects in their primordial state, such as protogalaxies and protoglobular clusters, as well as the formation of open clusters, of stars inside them, and of molecular clouds, but also of the great scales: primordial clusters of galaxies. In all these cases, although with different (and non-linearly scaling) physical and chemical conditions, a self-gravitating cloud undergoes fragmentation via gravitational, thermal, and Rayleigh-Taylor instabilities, while moving under the effect of its own gravity. The state of the gas is determined by the radiative and chemical processes. The rate of fragmentation, in its turn, depends on the state variables, and controls the birth of the "fragment phase" at the expense of the environment gas (see Di Fazio [1]). The fragments undergo internal self-gravitating evolution, with collapse and subfragmentation (except those which become stars), and so on [1]. A part from an initial, transient collision activity, their cross sections eventually reduce themselves, to yield a collisionless system of fragments that orbits in the cloud's gravitational field, embedded in the environment gas left over from the fragmentation process (see Capuzzo-Dolcetta, Di Fazio, and Menshchikov, [2]). One of the macroscopically important heating mechanisms for the gas is the dissipation of the turbulent supersonic wakes that are generated by the fragment's supersonic motions in the gas (see Battinelli *et al.* [3]). In making a global model of evolution of our protocloud, it is important to take into account another physical phenomenon: the generation of turbulence itself, as one of the phenomena connected to the resistivity force, due to the motions of fragments (see [3] and Di Fazio *et al.* [4]). In fact, before decaying into brownian motion, the turbulent motions "freeze" some amount of internal energy, and modify the thermodynamic evolution of the gas, which is essential to be able to calculate the absorption and emission functions of the gas. Moreover, the presence of diffused turbulence in the gas alters the dispersion relation for the various instabilities (in first place, the gravitational one), thus influencing the critical masses for fragmentation (e.g. the Jeans mass [3]).

In order to complete the picture regarding the need to calculate the resistivity force in our global models of evolution, we should account for the explosion of supernovae, which exerts an action which can be, from case to case, of compression, destruction, fragmentation enhancement, or wiping out of the outer parts of the shocked fragments.

In order to build a starting model of the gas flow-cloud interaction, we chose to study the simplified case of a spherical gaseous cloud which is hit at the fly-by of an interstellar shock (Bedogni and Woodward [5]). After the passage of the shock, a more quiescent phase takes place, with the formation of a turbulent wake. In this phase we evaluate and study the resistivity force, as well as other effects, like those due to the mass-loss which the cloud undergoes. The importance and the role of the resistivity force in generating turbulence was first proposed and investigated in Battinelli *et al.* [3], using an approximate approach to its velocity dependence in a complex global evolution program. In this work, we set up a numerical tool suitable to calculate the resistivity force quantitatively in different environment conditions.

As discussed by McKee [6], the resistivity force ("physical drag") plays an important role in the shock-cloud interaction. The value suggested by McKee [6] for the drag coefficient ≈ 2 is too large compared with the experimental results for rigid spheres, and is it also too large for very high Mach numbers. A numerical approach seems the only way to measure the correct drag and to understand its role in compressing and accelerating the interstellar clouds.

In this work, we test our hydrodynamical tool, by reproducing the experiments made on rigid spheres. Of course, this will not guarantee that the method is valid for the compressible case, but at least it will be valid in the limiting cases of the experiments that are available. Indeed, we should not forget that absolutely no laboratory experiment will reproduce the conditions of a supernova blast or the action of a supersonic motion with a Reynolds number of $10^6 \div 10^9$ and Mach numbers (at the shock front) over 500.

The first classical experiments for the measurement of the resistivity force exerted on a rigid sphere moving in a viscous flow, from laminar to highly turbulent regime (Reynolds number $\mathcal{R}e \in [10^6, 10^7]$) can be found in the literature of the first three decades of this century (see, e.g. the cumulative graph reported by Sedov [7]; Sedov [8] on the measurements by Allan, Goettingen, Libster, Schiller-Schmidel). In the data shown by Sedov, one can see the gradual, slow deviation from the so-called Stokes regime (laminar) starting at $\mathcal{R}e \gtrsim 1$. Up to that point, the resistivity force is linear with the Reynolds number. The dependence becomes more and more than linear with increasing $\mathcal{R}e$, and eventually becomes approximately quadratic in $\mathcal{R}e$ in the interval $\mathcal{R}e \in [10^3, 10^5]$. A conspicuous feature shows up after that interval, and lasting for about a dex in $\mathcal{R}e$, in that the resistivity force sharply drops and then rises again. This is what is well known as the so-called "crisis of resistivity" in the subsonic case.

Even the measurements for the simplest objects (rigid spheres, cylinders, cones and wedges) posed severe problems, e.g. how to hold these objects in the flow without interfering heavily and uncontrollably with the measure itself. The interaction of the formed train of waves with the wind tunnel or liquid flow conveyor also interferes with the measured force, not to speak about the train of waves formed by the device holding the object in the flow (e.g. several kinds of dynamometers). Moreover, soon it was evident that the dependence of the resistivity force and of the main parameters of turbulence in the wake of the object on the Reynolds number was also a quite variable function of the Mach number.

The visualization of the flow in the experiments also turned out to be a delicate matter, and the different experimental methods adopted showed different characteristics and parameters of the flow in the wakes of the objects. In this context, for example, consider the particle suspension methods and the electrolytic hydrogen bubble methods in liquids, the electric spark tracing via high voltage electrodes (visualizing fluid particles path lines in their flow), and the photographic "schlieren" and shadowgraph methods, being respectively sensitive to the first and second spatial derivative of the density.

The main parameters of the flow that determine the resistivity force and other relevant quantities are the Reynolds number and the Mach number. The surface roughness of the chosen objects also influences the drag, and so does the tunnel blockage effect, in the cases where the motion takes place in a flow tunnel.

Among the main experiments in this problematic, we mention the measures of Wegener and Ashkenas [9] of the dependence of the resistivity force acting on a sphere at supersonic speeds on Mach number, for different Reynolds numbers. The experimental apparatus consisted in a sphere held by a wire and with a damping mechanism (see [9], p.553). They took into account the wire ("transducer") resistivity, and they could measure an increasing resistivity force with increasing Mach numbers in a small interval $\mathcal{M}_s \in [3.87, 4.33]$. The different Reynolds numbers were obtained using spheres of different diameters.

Resistivity force measurements (for spheres) using various types of transducers (or suspension systems) were compared by Bailey [10] in the case of subsonic speeds for Reynolds numbers ranging from 400 to $\sim 10^7$. Bailey showed that the different experimental methods affected the results from $\approx 2\%$ to more that $\approx 100\%$ (see [10] p.408), and proposed a carefully discussed derived "standard" drag curve which is very similar to that shown by Sedov ([7], p. 49).

Achenbach (11) set up experiments in order to determine the influence of surface roughness and tunnel blockage on the resistivity force. Achenbach shows that the crisis of resistivity moves to the left (i.e., towards lower Reynolds numbers) while the roughness ratio increases (the latter ratio being the average micro-roughness linear size divided by the diameter of the sphere). Moreover, the minimum of the drag, reached in the crisis of resistivity dip, tends to increase its attained value with increasing roughness ratio (see Achenbach [11] p. 116). These results are of fundamental importance for our present paper, in that our calculations give for the boundary of the simulated object not a smooth sphere, of course, but a stepwise-rough sphere, due to the computational grid.

This, in its turn, will induce some sort of a "numerical surface roughness effect", of which we will have to be aware in the numerical simulations. Achenbach also showed that the resistivity force increases with increasing blockage ratios (tending to equal the so called "ram pressure" force $\propto \rho u^2$ when the blockage ratio tends to 1). Moreover, with higher blockage ratios, the crisis of resistivity takes place at higher Reynolds numbers, the attainment of such crisis is smoother (the slope in the dip is lower), and the minimum of the drag reached in the crisis tends to be higher (see Achenbach [11] p. 121).

Miller and Bailey [12] show the drag coefficient (i.e., the resistivity force divided by the ram pressure, ρu^2 and by the geometrical cross section) versus the Reynolds number across the crisis zone, for spheres at various Mach numbers, from 0.3 way up to 1.8 (see [12], fig.4 p.456). The drag coefficient is shown to be ever increasing with Mach number, and the crisis of resistivity moves at higher $\mathcal{R}e$ with increasing Mach numbers. For Mach numbers greater than 0.8, the presented graph remarkably does not show any critical dip.

Bryson and Gross [13] studied the interaction of strong shocks with several types of objects embedded in supersonic flows.

A three dimensional numerical calculation on incompressible stratified flow past a sphere was performed at low Reynolds number ($\approx$ 200) by Hanazaki [14]. In his work, the drag coefficient is calculated, and its behaviour versus the Froude number is compared with the experiments; the pressure distribution on the surface of the sphere is also computed. Fornberg [15] presents a study of the wake vorticity for steady incompressible flow past a sphere with Reynolds numbers $\lesssim 5000$. No calculations on the main parameters of turbulence or on the drag coefficient are presented.

In the present paper, sect.**2** is dedicated to the description of the numerical method, the setup and calculation of the so called edit quantities (i.e. those physical quantities important for the description of the flow, but not primary unknown variables of the computation), and the various algorithms adopted. Section **3** displays the numerical results of the present work. Section **4** contains a discussion and compares our numerical results with the available experiments. The conclusions are in sect.**5**.

2. – Numerical setup

2˙1. *Numerical Method.* – The Piecewise Parabolic Method (PPM) developed by Colella and Woodward [16] is one of most powerful tools to simulate the unsteady flows containing strong shocks in astrophysics, such as supernova explosions [5]. Details of the characteristics and the performance of PPM in comparison with Godunov's , Mc Cormack's and FCT's methods are presented and widely discussed by Woodward and Colella [17].

The Euler equations describing the conservation of mass, momentum, and energy for an inviscid gas may be written as:

$$\partial_t \rho + \nabla\cdot(\mathbf{u}\rho) = 0 \tag{1}$$

$$\partial_t(\rho\mathbf{u}) + \nabla\cdot(\mathbf{u}\cdot\rho\mathbf{u}) = -\nabla p \tag{2}$$

$$\partial_t(\rho E) + \nabla\cdot(\mathbf{u}\rho E) = -\nabla\cdot(\mathbf{u}p) \tag{3}$$

where ρ is the density of the fluid, E is the total energy per unit mass : $E = 1/2u^2 + e$ (e=internal energy per unit mass), p is the pressure, and $\mathbf{u}$ the vector velocity. The pressure p is related to the density ρ and internal energy e by the equation of state $p = (\gamma - 1)\rho e$. The ratio of the specific heats is γ =5/3 for a monoatomic gas, and we choose $\gamma \simeq 7/5$ for air (the approximation is of course explained by the predominance of biatomic molecules in air).

2˙2. *Initial and Boundary Conditions.* – Cylindrical (z, r) coordinates are introduced to treat the case of a planar shock interacting with a spherical obstacle centered in z=2 and r=0 and a radius R normalized to 1. The rigid body is located in front of the incident shock and its radius R is a free parameter that selects the length scale of every single model. Our units are such that the density and the sound velocity in the ambient medium are set equal to 1 (i.e. $c_a \equiv 1 \equiv \rho_a$) so that pressure becomes $p_a = 1/\gamma$. The mass and time scales can be easily deduced from the physical quantities previously defined.

The incident planar shock ,in cylindrical coordinates, is perpendicular to the z axis and is moving from $(z = 0,\ r = 0)$ (at the initial time $t_{ini} = 0$) towards the positive z direction. For all cases, we actually compute using different Mach numbers. We stop the computation after a time $t_{fin} = f\tau$, where $\tau = 2R/u_s$ is the crossing time of the rigid body by the incident shock, and u_s is the shock velocity. We chose, for the factor f, $f = 20$ and $f = 40$, so that the final time is reached for all computations after approximately 18 and 36 crossing times.

The Mach number of the incident shock is defined as $\mathcal{M}_s = u_s/c_a$, where u_s is the shock velocity. The shocked quantities ρ_s,p_s,and u_{zs} are fixed by the Rankine-Hugoniot conditions:

$$\rho_s = \frac{(\gamma+1)}{((\gamma-1)+2/{\mathcal{M}_s}^2)} \tag{4}$$

$$p_s = \frac{{\mathcal{M}_s}^2(2\gamma+(1-\gamma)/{\mathcal{M}_s}^2)}{\gamma(\gamma+1)} \tag{5}$$

$$u_{zs} = \frac{\mathcal{M}_s(2+2/{\mathcal{M}_s}^2)}{(\gamma+1)} \tag{6}$$

and the r-component of the shock velocity is equal to zero $(u_{rs} = 0)$.

The calculation around the rigid body is made using a uniform grid. this approach has several advantages: *a*) the mesh generation is straightforward and easily computed , *b*) the grid refinement is easy to do and *c*) the grid effects arising from non-uniformities in the mesh are eliminated.

The numerical grid is set up as a rectangular domain with a uniform "active" part, surrounded by an "expanded" one, in which the mesh points are not equidistant, but designed to permit the free development of the bow shock through the upper side of the active grid. The active part of the grid extends from $z = 0$ to $z = 9$ and from $r = 0$ to $r = 3$, and selects the region of the numerical computation with uniform mesh of size $\triangle r = \triangle z$, depending on the resolution of the problem. Non uniform expanding zones, from -10 to 0 in z and from 3 to 20 in r, define the expanded region.

The primary difficulty with this kind of grid is, of course, represented by the boundary conditions.

Flow-in ($z = -10$) boundary conditions are fixed on the left side of the $z - r$ grid,in order to make sure that the shocked values are incoming. Flow-out ($z = 9$) and ($r = 20$) boundary conditions are imposed on the right and top sides of the $z - r$ grid, in order to allow the outflow of the fluid quantities. An outflow condition is accomplished by maintaining a "zero gradient" in all fluid variables (i.e. $\nabla \cdot \mathbf{u} = 0$, $\nabla p = 0$ and $\nabla \rho = 0$). In the next sections, we will take care only of the numerical results in the active (uniform) part of the grid.

We set reflecting boundary conditions at the bottom ($r = 0$) of the $z - r$ grid, enforcing an axial symmetry with respect to the plane containing the z axis and perpendicular to the r axis. This is of course obvious in a 2D calculation.

We paid particular attention to the calculations on the boundary wall between the object and the gas (see fig. 1), represented as an impenetrable series of "stair steps". The numerical flux at the boundary of the sphere is computed by simply enforcing the "wall" boundary conditions : $\mathbf{u} \cdot \mathbf{n} = 0$ (where $\mathbf{n}$ is the unit vectors normal to the sphere) and the continuity of pressure p.

As the resolution increases, the uniform grid approaches very well the "natural" boundary of the body. For the cases of low resolution, a new treatment has been used in order to determine better approximated fluxes at the zone interfaces. In order to do this, like in Edgar and Woodward [18] we allow "fractional" steps, producing a smoother boundary for a given computational mesh. This correction technique yields a sensible improvement in the comparison with the experimental data because tends to enforce a no-slip boundary condition near the wall of the sphere [18].

2˙3. *Edit quantities.* – In this subsection, we introduce and set up the so-called *edit quantities*, i.e. several useful variables that will be computed and displayed. They are called "edit" in contrast to the "fundamental" variables, i.e. the unknowns of the differential equations. The edit quantities, functions of the unknowns, are -of course- not needed to solve the system of partial differential equations, but they are needed for a better and detailed physical description of the numerically studied phenomena.

2˙3.1. Resistivity Coefficient and Drag force. Let us define the resistivity coefficient (or drag coefficient) (Landau and Lifshitz [19]; Batchelor [20])

$$C_D \equiv F_D/(0.5\rho_\infty {u_\infty}^2 A_s),$$

where:

$$F_D = \int_S p(\mathbf{n}\cdot\mathbf{i})dS \tag{7}$$

(dS being the elementary vector area) is the resistivity force (or "drag") exerted on the object by the relative flow, ρ_∞ and u_∞ are, respectively, the density and velocity at infinity while A_s is the area of the projection of the body onto a plane normal to the flow where $\mathbf{n}$ and $\mathbf{i}$=$\mathbf{u}_\infty/u_\infty$ are unit vectors, respectively, normal to the elementary surface dS and in the direction of the flow. Depending on the chosen geometry, this area is : $A_s = \pi R^2$ for a sphere .

Alternatively, the force can be also computed using the momentum conservation in a fluid volume containing the flow, surrounded by the closed surface Σ :

$$F_D = \oint_\Sigma (p\mathbf{n} + \rho\mathbf{u}(\mathbf{u}\cdot\mathbf{n}))\cdot\mathbf{i}d\Sigma \tag{8}$$

In practice, the integral can be performed between two infinite planes at z=constant: one of them in front of the body and the other far behind it (Landau and Lifshitz [19]). Due to the fact that the integral performed over the lateral surface vanishes at infinity, one can write the force F_D exerted on the body as the difference between the total momentum flux entering in front of the calculation flux tube and that leaving from the end of said tube. We actually use expression (7) in the calculations.

2˙3.2. Viscosity and Reynolds number. Here we compute the fluid flow around a rigid body, for relatively weak shocks, with different resolutions. The PPM solves the Euler equations of inviscid flow. Therefore, the method's viscosity is actually due to the approximation of calculating the equations as finite differences performed on a computational grid. In particular, the behaviour of the numerical viscosity is very similar to a true viscosity and dissipates the kinetic energy of small-scale fluid motions into heat. This dissipation is discussed by Porter and Woodward [21] for thermal convection and by Edgar and Woodward [18] for the case of a shock hitting a wedge. For some well representative case, the effective kinematic viscosity ν_{eff} was found to be approximately :

$$\nu_{eff}/c_s D \propto (\mathcal{M} + 0.25)^{3.15} (D/\Delta x)^{-3} \tag{9}$$

where c_s is the local sound speed, $\mathcal{M} = u_\infty/c_s$ is the adiabatic Mach number of the flow, Δx the resolution of the computational mesh and D the scale length - i.e. the diameter $D = 2R$ - of the rigid body.

The expression (9) was obtained by Porter and Woodward [21] by studying the evolution of a sinusoidal wave perturbation in a hydrodynamical flow at different Mach numbers, using the PPM hydro scheme. Given the statistical shape of the pulsations in a developed turbulence medium, the perturbation studied by Porter and Woodward [21] is very reasonably applicable to a wide manifold of flows in the conditions studied by our paper. We carefully checked the results of Porter and Woodward [21], reproducing exactly the same behaviour. Furthermore, we applied the same perturbation to other hydro schemes (Van Leer upwind scheme with linear and parabolic approximation – e.g. the "ZEUS-2D" code (by U.I.-NCSA, Urbana-Champaign) – the first order Godunov scheme and the second order TVD scheme of

Harten "UNO"), obtaining the same qualitative dependence of ν_{eff} on $(D/\Delta x)^{-\alpha}$ at a given Mach number, but with exponents α less than 3, ranging in the interval $[3, 1]$. Here $\alpha = 3$ is the exponent obtained for the PPM method.

We chose the latter, because it is much less viscous than the other methods tested and is certainly among the less numerically viscous methods in the literature for 2D calculations. This characteristic of having a substantially lower numerical viscosity is very important, as we are forced to compute runs with lower and lower minimum mesh size, in order to be able to calculate cases with higher and higher Reynolds numbers. Of course, if the chosen method's numerical viscosity were higher, one could obtain only lower Reynolds numbers per given computation time costs.

The Reynolds number $\mathcal{R}e$ is defined as :

$$\mathcal{R}e \equiv (u_\infty D)/\nu \tag{10}$$

where ν –in general– is the kinematic viscosity, while in our case it is the numerical viscosity. The parameter $\mathcal{R}e$, of course, is a very important one for the flow. In terms of the Mach number, we get: $\mathcal{R}e = (\mathcal{M} c_s D)/\nu_{eff}$ where $c_s = (\gamma p_\infty / \rho_\infty)^{1/2}$ is the local adiabatic sound velocity far from the object, so that $\mathcal{M} = u_\infty / c_s$. The Mach number of the local perturbed flow can be expressed [19] in function of the Mach number of the incident shock $\mathcal{M}_s$, for $\mathcal{M}_s \geq 1$, as :

$$\mathcal{M} = \frac{2({\mathcal{M}_s}^2 - 1)}{\left[(2\gamma {\mathcal{M}_s}^2 + (1-\gamma))((\gamma - 1){\mathcal{M}_s}^2 + 2)\right]^{1/2}} \tag{11}$$

Therefore this quantity, for very strong shocks with $\mathcal{M}_s \rightarrow \infty$, is limited by its maximum value $\mathcal{M} = 2/\left[2\gamma(\gamma - 1)\right]^{1/2}$.

We compute several models with different Mach numbers. In Table 1 we show the Mach numbers of the incident shock $\mathcal{M}_s$ together with the local Mach numbers $\mathcal{M}$ for $\mathcal{M}_s \geq 1$.

Table1: Mach Numbers		
$\gamma = 7/5$	$\mathcal{M}_s$	$\mathcal{M}$
A	1.20	0.287
B	1.50	0.604
C	2.00	0.962
D	2.81	1.304
E	3.00	1.357
F	4.00	1.553
G	∞	1.889

Using the thermal r.m.s. velocity

$$u_{th} = (3K_bT/(\mu m_H))^{1/2}$$

we obtain

$$\mathcal{R}e = (\mathcal{M}c_sD)/(\lambda u_{th}) = [\mathcal{M}(\gamma/3)^{1/2}]/\lambda$$

introducing the temperature T, the Boltzmann constant K_b, the mean-free path of the particles λ, the mass of the hydrogen atom m_H, and the average molecular weight, μ. In this work, however, we actually use the numerical viscosity in place of the physical one. We will now analyze what this implies. In our method, the expression for the numerical viscosity given above (9) can be expressed also by :

$$(12) \qquad \nu_{eff}/c_s L \approx 0.462 \times (\mathcal{M}+0.25)^{3.15}(L/\Delta x)^{-3}$$

This expression was derived by Edgar and Woodward [18]. In (12), $2^{1/2}L$ is the wavelength of sinusoidal velocity variations; here we assume $2^{1/2}L$ to be $\approx D$.

Using this numerical viscosity, we calculate what we call the "numerical" Reynolds number, redefined by using (10) and (12) as :

$$(13) \qquad \mathcal{R}e_{eff} \approx 1.082\mathcal{M}(D/\Delta x)^3/(\mathcal{M}+0.25)^{3.15}$$

We now note the following. In the case we want to study different viscous flows without changing neither the dimension of the object nor the Mach number, our scheme implies that we have to resort to vary the size of the mesh intervals. Furthermore, the functional dependence of this numerical Reynolds number on the relevant quantities is different from that of the usual physical Reynolds number.

In fact, only for infrasonic flows it has about the same $\mathcal{M}$-dependence as the physical $\mathcal{R}e$, while for sonic and supersonic flows it decreases with increasing $\mathcal{M}$ (with power ≈ 2.15), whereas the physical $\mathcal{R}e$ increases linearly. The numerical $\mathcal{R}e$ increases with increasing linear dimension D (with power 3), whereas the physical one increases only linearly. For the sake of comparison, we consider the "physical" Reynolds number $\mathcal{R}e = (u_\infty D)/\nu_{phys}$ in two cases : a) for a neutral monoatomic hydrogen gas ($\gamma = 5/3$) we have $\nu_{phys} = \nu_{hyd} \approx (2 \times 10^{-15}T^{5/2})/(D ln\Lambda)$ where $ln\Lambda$ is the so called Coulomb logarithm (or Gaunt factor); b) for air ($\gamma = 7/5$),using the Sutherland law $\nu_{phys} = \nu_{air} \approx (\rho_{\nu_0}/\rho)(1+119.5/T_0)/(1+119.5/T) \times (T/T_0)^{1/2}$ where $T_0 = 273.1\ C^\circ$ and $\rho_{\nu_0} = 1.293 \times 10^{-3}$ are the ambient temperature and density.

2˙3.3. Scale of Turbulence and Pulsation. We also define the vector pulsation $\triangle\mathbf{u} \equiv \mathbf{u}(z,r) - \mathbf{u}_\infty$ as the vector difference between the local fluid velocity and the general flow velocity far from the interaction region. Let us consider the scalar product $\triangle\mathbf{u} \cdot \triangle\mathbf{u}'$ where $\triangle\mathbf{u}'$ is the pulsation at a point $\mathbf{P}'$ on a sphere of fixed radius r centered in the point $\mathbf{P}$ whose pulsation is $\triangle\mathbf{u}$.

Let us define the quantity W as the average of the absolute value of said scalar product : $W \equiv \langle | \triangle\mathbf{u} \cdot \triangle\mathbf{u}' | \rangle_r$, where the subscript r stands to remind that the average is done on the mentioned sphere's surface, and the angular brackets indicate the operator of the average. We now perform said surface average on a sphere of radius r:

$$W = \int_{S_{(r)}} (dS' \mid \triangle\mathbf{u} \cdot \triangle\mathbf{u}' \mid)/A = W(\mathbf{P}, r) \tag{14}$$

where $A = 4\pi r^2$. We can explicit the scalar product in terms of the components u_z and u_r of the velocity at point $\mathbf{P}$ at the center of the sphere of radius r and with the components $u_z{}'$ and $u_r{}'$ of the velocity $\mathbf{u}'$ of a point $\mathbf{P}'$ running on the sphere.

Considering that only the primed quantities vary with r we obtain :

$$W = \mid u_\infty{}^2 - u_z u_\infty - 1/A[(u_z - u_\infty)\int_{S_{(r)}} u_z{}' dS' + u_r \int_{S_{(r)}} u_r{}' dS'] \mid \tag{15}$$

Let us now consider the function $W(r)$. Let the quantity W_{max} indicate the maximum on the function $W(r)$, i.e. the maximum of the average of the square pulsation. We then get the main scale of turbulence L_{tur} as the semi-width of the distribution of $W(r)$ taken at half-height. The quantity $W_{tur} = 0.5 \times W_{max}$ is the square of what can be called the "main pulsation", being referred to a point at distance L_{tur} form the center of the distribution. Since W_{tur} is an average square pulsation, we actually choose to plot the r.m.s. of the main pulsation, Δu_{rms}, defined as :

$$\Delta u_{rms} \equiv {W_{tur}}^{1/2} \tag{16}$$

It will be useful to compare the main scale of turbulence with the transverse kinetic linear scale, defined as:

$$K_{tur} \equiv [\langle u_z - u_\infty \rangle^2 / \langle \partial u_z / \partial y \rangle^2]^{1/2} \tag{17}$$

2˙4. *Selected Parameters.* – We have chosen to integrate models with the following values of the local Mach numbers $\mathcal{M} = 0.28, 0.60, 0.96, 1.30, 1.35, 1.55$ see table 1 for the corresponding Mach numbers of the incident shock. We select the above set of Mach numbers so that, while having an economically reasonable quantity of cases, we also have an acceptably "dense" set. The first two values were chosen in order to scan the infrasonic cases, and the value $\mathcal{M} = 1.30$ was selected in order to simulate some experiments by Bryson and Gross [13]. In almost all the integrated cases, we have chosen to follow a biatomic gas, due to the fact that this paper deals mostly with the simulation of laboratory experiments. For this reason, the value for the specific heat ratio $\gamma = 7/5$ is almost never indicated. On the contrary, when we choose to integrate a monoatomic case, the figures have the $\gamma = 5/3$ specification.

3. – Numerical Results: flow around spheres

3˙1. *Subsonic Case.* – Throughout the present paper, the figures displaying 4 snapshots of the various variables are set so that the first snapshot is in the upper left corner, the second in the upper-right, the third in the lower left, etc.. The corresponding sequential times are: 5, 10, 15, 20 in the sphere-crossing time units

τ (see sect.**2**). Hereafter, we will call the quantity $\mathcal{H} = ln(p/\rho^{\gamma})$, proportional to the local entropy, simply "the entropy". Moreover, hereafter, what has been defined in sect.**2** as the "numerical Reynolds number" will be simply called the Reynolds number. The reason why we begin to present the first snapshot at $\tau = 5$ is that, during this interval, the flows settle down to a statistically steady condition. In this period, the bow shock forms and dissipates (for $\mathcal{M}_s \leq 2$, i.e. $\mathcal{M} \leq 0.96$) into a subsonic wave.

We first fix our attention on an infrasonic case with $\mathcal{M}_s = 1.2$ - or $\mathcal{M} = 0.28$- grid resolution $\triangle x = 0.02$ and Reynolds number $\mathcal{R}e \approx 2.2 \times 10^5$.

Let us look at fig. 2 (*a*-*d*) and fig. 3 (*a*-*d*). Figure 2 shows the level curves of the variable $log_{10}\rho$. We can see the conspicuous formation and evolution of the characteristic Karman vortexes, with the secondary ones all along the wake. Along the body of the sphere, just after the detachment of the limit layer, we can see the initial formation of the vortexes. Figure 3 shows the level curves of the "entropy", taken at the same times as above. The boundary of the limit layer is very well marked and the structure of the wake is clearly seen in its development. A weak contact discontinuity can be seen going past the sphere in the first and second frame. Moreover, the choice of plotting the "entropy" allows to evidence contemporaneously all the structures associated with entropy, i.e. the vortexes and the various discontinuities in the flow, thus also tracing very well the formation of the wake. A minimum in density is reached inside the main vortex (see fig. 2) corresponding to the maximum dissipation in entropy.

More details on the dynamical evolution of the shock-sphere interaction can be better enhanced looking at an animation-presented evolution, made using lower grid resolution $\triangle x = 0.04$ and Reynolds number $\mathcal{R}e \approx 2.7 \times 10^4$. The animation clearly shows a transient phase lasting some sphere-crossing times (i.e. $t \lesssim 5\tau$,see above) in which the only visible feature is the dissipation of the reflected subsonic wave. After this phase, the main shock is gone beyond the grid and a vortex (see fig. 2 *a* and 3 *a*) forms near the upper side of the sphere, moving slowly towards the increasing z. A simulation with higher resolution - $\triangle x = 0.01$ - and Reynolds number $\mathcal{R}e \approx 1.7 \times 10^6$ (not shown here) reveals that the vortex sheet splits in a number of eddies, even if the main vortex is already present with smaller size. This effect is clearly due to the lower numerical viscosity, which increases the turbulence in the shocked fluid.

3·2. *Supersonic Case.* – We now examine a model with $\mathcal{M}_s = 3$ (or $\mathcal{M} = 1.35$), in the case of the sphere, without changing the resolution of the numerical grid but for a Reynolds number (depending on $\mathcal{M}$ see 2.9) $\mathcal{R}e \approx 3.3 \times 10^4$.

This is a substantially supersonic case, and thus we have the formation of a single vortex (as always in our models, the second, symmetrical one is only a specular reflection). The vortex keeps rather close to the sphere and moves inappreciably with respect to it (see fig. 4 *a-d* and fig. 5*a-d*, showing the level curves, respectively, of $log_{10}\rho$ and $\mathcal{H}$). From fig. 4 we can see the evolution of the bow and tail shocks. The wake gets thinner with distance from the sphere. From fig. 5 we can see the high degree of concentration of the entropy in the wake for this case. We remark that, in the case of the flow at $\mathcal{M}_s \gtrsim 2.0$ (i.e. $\mathcal{M} \gtrsim 0.96$, not shown here) in which the bow shock begins to appear, the wake begins to be more appreciably collimated around the symmetry axis. At higher resolution - $\triangle x = 0.01$ - and Reynolds number $\mathcal{R}e \approx 2.6 \times 10^5$ (not shown here) we get more details of the structure of the vortex but the other features remain unchanged.

4. – Comparison with Experiments and Discussion

4.1. *The Resistivity curve.* – The models that have been run cover the following range for the Reynolds number: $\mathcal{R}e \in [50, 3\times 10^6]$. In this section we shall speak of Mach numbers, actually meaning local (or "post-shock") Mach numbers, using the previously introduced symbol $\mathcal{M}$. Moreover, we throughout use $\gamma = 7/5$. The values of the numerical drag C_D (see sect.**2.3**) presented below have been obtained performing the mean of the single values computed in a time interval $\triangle t \in [8, 20]\tau$, and $\triangle t \in [8, 40]\tau$ (in order to check to time behaviour of C_D for longer times) τ being the crossing time. The Reynolds numbers (see (2.9)) are corrected by a factor ≈ 0.1 in order to calibrate the resistivity curve with the experimental one, given by Sedov [8], in the low values of $log_{10}\mathcal{R}e$.

In fig. 6 we can see the behaviour of the resistivity coefficient C_D versus $log_{10}\mathcal{R}e$ for some of the Mach numbers defined in sect.bf2.6 ($\mathcal{M}$= 0.28, 0.60, 0.96, 1.553). Said behaviour can be compared with that corresponding to the experiments reported by Miller and Bailey [12] (p. 458, fig. 4, see fig. 7 of the present paper) taking into account that their C_D versus $\mathcal{R}e$ curves are labelled by the previously mentioned post-shock Mach numbers. There is a good agreement (max error ≈ 5 %) for Mach numbers $\mathcal{M} \gtrsim 0.96$, practically on all the part of Miller and Bailey's $\mathcal{R}e$

range covered by our runs: $10^4, 10^5, 10^6$.

For the lower Mach numbers ($\mathcal{M} = 0.28, 0.60$) the theoretical values of C_D fit very well the "plateau" of the experimental curves. This result, expecially for the subsonic case, has been obtained introducing a new treatment to calculate better approximated fluxes at the zone interfaces near the object. As the resolution decreases, in fact, the grid does not well follow the natural boundary. In this context, we adopt Edgar and Woodward's [18] method, allowing for "fractional" steps which produce a smoother boundary for a given computational mesh. This correction algorithm yields a conspicuous improvement (see Appendix A) in comparison of the calculations without the corrected fluxes.

As pointed out by Edgar and Woodward [18] this improved boundary treatment has the benefit of a mesh refinement in the computation of the flow around a sphere. As a consequence of the flux correction, a boundary layer is generated next to the wall enforcing a no-slip boundary condition. The existence of this boundary forces the solution toward the limit of "viscous" solutions rather than the inviscid solution of Euler's equations. The average error on C_D is lowered from $\lesssim 35\%$ down to $\lesssim 7 \div 8\%$.

However this improvement is small near the critical regions. The displacement of the resistivity crisis to the right, i.e. to greater Reynolds numbers, and the discrepancy with the experimental data in the critical region 2×10^5 are both partly due to the so-called surface roughness effect.

In fact, our sphere is actually sort of rough, due to the fact that its boundary is not continuous like a real sphere, but step-wise shaped, depending on the mesh interval size. The effect of surface roughness on the resistivity force has been investigated through experiments. For the case of the sphere, see e.g. [11] p.116, fig. 2 i.e. (fig. 8 of the present paper). The experiments show that for $\mathcal{R}e \lesssim 10^5$ the resistivity coefficient is practically unaffected by the surface roughness. For $\mathcal{R}e \gtrsim 10^5$, instead, the effect is conspicuous, and it consists in displacing to the left (i.e. to lower Reynolds numbers) the crisis of resistivity. In other words, the greater the surface roughness, the lower the critical Reynolds number marking the dip due to the resistivity crisis.

Let us now discuss how we simulate, in our theoretical numerical experiments, the different Reynolds numbers. In order to obtain them, we exploit the dependence of the PPM artificial viscosity on the mesh interval size (see (13)) and thus we obtain different Reynolds numbers using different mesh intervals. By doing so, the numerical representation of the sphere will have a smoother or rougher surface, and it will induce different behaviours of the resistivity force, due to the surface roughness effect. In particular, from fig. 6 we understand that, in our numerical simulations, the crisis of resistivity will be moved somewhat to the right.

Let us consider the curve corresponding to the lower Mach number ($\mathcal{M} = 0.28$) (see fig. 9). We see that each point to the right of $\mathcal{R}e = 10^5$ (i.e. the point to the right of which the surface roughness effect is active) is obtained by lowering $\triangle x$, thus with a lower surface roughness. This moves to the right the resistivity crisis, as fig. 8 shows that, as we lower the surface roughness, the resistivity crisis moves to the right (i.e. to higher $\mathcal{R}e$). We note that the surface roughness effect (see our fig. 9 and Appendix B) greatly reduces the error in estimating the resistivity crisis drop in C_D. The error, in average, diminishes by a factor $\approx 2 \div 2.5$ in C_D, and by a factor ≈ 1.3 in the estimate of the Reynolds number corresponding to the minimum in C_D.

In fig. 10 we show the resistivity coefficient versus time in crossing times . We note that, for intermediate resolutions and Reynold's numbers, C_D settles down to a quasi-constant value, as required by the stationary solution.

The data displayed in the following are calculated at a time $t = 20 \times \tau$. In fig. 11 we plot the resistivity coefficient versus Mach number, for different resolutions. We can see how the resistivity force tends to be proportional to the so-called ram pressure (ρu^2, corresponding to a coefficient $C_D = 1$) when the Mach number increases. At a given Mach number the points representing higher resolutions (lower $\triangle x$) correspond to more turbulent cases, since the higher the resolution the higher the Reynolds number. The difference in behaviour with different resolutions tends to zero as the Mach number increases. On the contrary, for low Mach numbers ($\mathcal{M} \lesssim 0.96$) the most turbulent cases tend to have lower resistivity coefficients. Note the minimum reached around $\mathcal{M} = 0.60$ for the intermediate resolution $\triangle x = 0.015$.

4·2. *Turbulent energy, pulsation, and the main scale of turbulence.* – We now discuss some of the other (previously introduced in sect.**2.3** *edit quantities* of the model).

An interesting "edit" quantity is the ratio $E_{tur}/\mathcal{L}_{tur}$ (energy in turbulence in the wake behind the sphere / work done by the resistivity force, hereafter "turbulent energy ratio", indicated by $E_{tur}/\mathcal{L}_{tur}$). So we define $E_{tur} \equiv \int_V (1/2)\rho \triangle u^2 dV$ and

$\mathcal{L}_{tur} \equiv \int_{\triangle t} F_D \mathbf{i} \cdot \mathbf{u} dt$ where E_{tur} is computed at time $t = \triangle t = 20\tau$, V is the volume of the wake, and $\mathbf{i}$ is a unit vector in the direction of the flow. Figure 12 shows the mentioned ratio versus Reynolds number, for different Mach numbers. We can see that said ratio tends - roughly speaking - to accumulate to the value ≈ 0.5 for low Mach numbers, while keeping as low as $C_D \lesssim 0.05$ for the highest Mach number considered ($\mathcal{M} = 1.55$). This means that the efficiency in generating turbulence - in the considered range of $\mathcal{M}$ - is a decreasing function of the Mach number. Moreover, while at low Mach numbers (infrasonic regimes) the turbulent energy ratio is an increasing function of the Reynolds number, at the sonic and supersonic regimes the turbulent energy ratio is roughly constant, or a slightly decreasing function of $\mathcal{R}e$. This circumstance confirms the higher efficiency in generating turbulence for infrasonic motions.

Figure 13 (*a*-*c*) shows the linear transverse kinetic scale, K_{tur} (see (17)) versus coordinate z (parallel to the flow) for different Reynolds numbers, and for different Mach numbers. The variable K_{tur} actually measures the vertical scale of the velocity variations (i.e. transverse with respect to the flow direction). The upper plot of fig. 13 (*a*-*c*) refers to an infrasonic case ($\mathcal{M} = 0.28$), the central one to a roughly sonic flow ($\mathcal{M} = 0.96$), and the lower one to a supersonic flow with $\mathcal{M} = 1.55$. As expected from the discussion of fig. 12, at higher and higher Mach numbers correspond lower and lower transverse irregularities. Moreover, these tend to spread all over the wake for the sonic and supersonic cases, with the latter being the most spread-out. The effect of the Reynolds number is lower, tending practically only to anticipate the point in the wake where the greatest transverse irregularities are met.

In other words, the largest transverse irregularities move closer to the sphere, in its wake, when the Reynolds number increases. An effect of an increasingly chaotic behaviour with $\mathcal{R}e$ in the direction transverse to the wake can be seen only in the supersonic case, while the magnitude of the irregularities, on the contrary, decreases with $\mathcal{R}e$ for $\mathcal{M} = 1.55$.

Figure 14 (a-c) displays the main scale of turbulence, L_{tur} versus z for the same $\mathcal{R}e$ and $\mathcal{M}$ of fig. 13 (a-c). Roughly the same behaviour of K_{tur} can be seen for the main scale of turbulence. The latter is essentially different from the previously examined variable K_{tur} in that it is the scale of irregularities in the pulsation vector $\triangle \mathbf{u}$, statistically averaged *in all directions*. Said irregularities in the flow tend to decrease with increasing Mach numbers. The behaviour of L_{tur} with Reynolds number is rather chaotic for the infrasonic case, while tending to be smoother for the sonic and supersonic case. The lower plot of fig. 14 shows that it is not true that the irregularities in the motion in all directions diminish for increasing $\mathcal{R}e$, opposite to what happens for the transverse irregularities. This means that, for the supersonic case, the irregularities of the flow tend to be more along the direction of the flow rather than perpendicular to it.

In fig. 15 (a-c) we can see the behaviour of the r.m.s. of the pulsation of the flow, along the axis of the flow, z. The dependence of $\triangle u_{rms}$ on $\mathcal{R}e$ is rather chaotic for the infrasonic case, while tending to become monotonically decreasing for the sonic and supersonic cases.

Figures 16 ($a - c$) display the r.m.s. pulsation of the flow ($\triangle u_{rms}$, "main pulsation") versus the main scale of turbulence L_{tur}. Each point ($\triangle u_{rms}$, L_{tur}) is taken in a different location in the flow. Taking into account the "decay law" for turbulent irregularities (the *law of Kolmogorov-Oboukhov*, see e.g. [19]), fig. 16 is an approximate representation of the spectrum of the turbulent pulsation in the flow. Using the relation $(2E_{tur}/\rho)^{1/2} \approx \triangle u_{rms}$, i.e. knowing that the pulsation is proportional, in order of magnitude, to the square root of the energy in turbulent motions, fig. 16 actually gives us an approximate glance to the turbulent energy spectrum in the wake of the turbulent flow behind the sphere. Although the general aspect of the graphs is somewhat chaotic, there are indeed several discernible interesting characteristics. To start with, a visible effect to be remarked is the displacement of "power" to the lower scales when Mach number increases, with the exception of the high Reynolds case of fig. 16 *a*, which refers to a case on the right of the resistivity crisis dip. The dependence on Reynolds number is more chaotic, even though it shows the rough tendency to have higher pulsations for the lower Reynolds numbers, except when peaks –or spectral "lines" – are formed. The latter – for all Mach numbers – tend to form at the high Reynolds numbers considered. In particular, the spectrum appears to be peaked for the Reynolds numbers characteristic of the resistivity crisis or –more precisely speaking– of the plateau zone located just before the resistivity crisis. For example, looking at fig. 16 *b* (transonic case), and therein at the curves with Reynolds numbers $\mathcal{R}e = 5.7 \times 10^4$ and $\mathcal{R}e = 1.3 \times 10^5$, the spectrum peaks -kind of sharply- around 0.8 L_{tur}). For the lowest and highest Mach numbers (infrasonic and supersonic cases), fig. 16 *a* and 16*c* show the tendency of the spectrum to peak around $0.4 \div 0.5$ L_{tur}) for the higher Reynolds numbers, i.e. the tendency of having more "power" in the lower scales for the highest Reynolds numbers.

Figures 17 $(a-c)$ show the energy density in turbulent motions, $\epsilon_{tur} \equiv 0.5\rho \triangle {u_{rms}}^2$, plotted vs. the main local scale of turbulence, L_{tur}, for various values of the Reynolds number $\mathcal{R}e$ and of the Mach number $\mathcal{M}$, as in fig. 16. The same behaviour applies for the turbulent energy density as for the r.m.s. turbulent pulsation, except that: i) the lines are more pronounced, and ii) more fine structure can be seen, due to the fact that we used 30 radius points (instead of 20) for the calculation of the average used in $\triangle u_{rms}$ (see sect.**2.3.3**). The conspicuous line around $L_{tur} \approx 0.9$ tends to move to the left (i.e. at lower L_{tur}) as the Reynolds number $\mathcal{R}e$ increases. Contemporaneously, said line tends to spread out.

Figures 12-17 would suggest the realization of experiments in order to check the behaviours of the "turbulent energy ratio", the turbulent linear scale, and the turbulent pulsation, that we obtained from our hydrodynamical numerical calculations.

5. – Conclusions

The results reported in this paper, compared to the experimental data, show that the PPM is a good method in simulating the evolution of the flow around a sphere, generally in a supersonical, turbulized state. The PPM numerical viscosity, together with boundary correction and roughness effect correction, allows us to fit reasonably well the resistivity curves determined experimentally, both in the subsonic and in the supersonic cases. The numerical errors in simulating the experimental value of the drag coefficient, after accounting for boundary correction and surface roughness effect, the error regarding the determination of the position of the resistivity crisis (i.e. the error on Reynolds number corresponding to the dip) has become relatively small: a few percent for the sphere.

Section **2** describes the numerical setup, with method and edit quantities. Section **3** shows the numerical outputs for a typical subsonic and supersonic case for the sphere. In sect.**4** we discussed in detail the ability of the code to compute the numerical drag, ranging from low to high Reynolds numbers.

The large computer time required to reproduce the "transcritical" part of the resistivity curve (i.e. for very high resolution: $\triangle x < 0.01$) does not allow to reproduce more precisely this interesting part of the drag curve. Some interesting characteristics of the flow were found in our calculations, regarding: i) the efficiency in producing turbulence via the work of the resistivity force; ii) a– the correlation between the main r.m.s. pulsation and the main scale of turbulence; ii) b–in the subsonic and supersonic regimes a characteristic "line" is formed in the spectrum for high Reynolds numbers in correspondence with $L_{tur} \approx 0.4 \div 0.5$, and in the transonic regime at $L_{tur} \approx 0.8$; iii) the distribution of the pulsation along the wake. All these would suggest the realization of experiments in order to check the behaviours seen in fig. 13-17 *(a-c)*.

The present method will be used -in a forthcoming paper- to calculate 1) the resistivity force, 2) the efficiency in generating turbulence by the work of the resistivity force, and 3) the pulsation, in the case of the gaseous flow around a compressible, more dense gaseous object. This will be aimed to astrophysical investigations, as described in the Introduction. In that case, the mass-loss from the gaseous object, as well as the induced fragmentation inside it, will be computed. The fairly good reproduction of the experiments in the case of the flow around rigid bodies, performed in the present work, makes it possible to assume –with relative confidence– that the future calculations for a compressible object, using the present algorithms, will also be reasonably good.

Appendix A.

Introducing the boundary correction

In sect.**2**, we introduced the problem of the roughly approximated flux at the zone interfaces. As the resolution decreases, in fact, the grid does not well follow the natural boundary. Thus, for the cases of low resolution, a new treatment has been introduced, in order to calculate better approximated fluxes at the zone interfaces near the object. In this context, we adopt Edgar and Woodward's [18] method, allowing "fractional" steps which produce a smoother boundary for a given computational mesh. This correction algorithm, as shown in fig. A1, yields a conspicuous improvement. In fact, the corrected flows allow us to calculate drag coefficients that are affected by much smaller errors with respect to those of the C_D calculated without boundary correction. Figure A1 shows, for the spherical, infrasonic case ($\mathcal{M} = 0.28$) the drag coefficient vs. Reynolds number. The experimental data are displayed together with our computed data. The improvement, in the laminar and in the critical regions, is small. On the contrary, a very pronounced improvement takes place in the "plateau" region: the average error on C_D is lowered from $\lesssim 35\%$ down to $\lesssim 7 \div 8\%$. The critical region (crisis of resistivity) is improved greatly by the roughness correction.

Appendix B.

Accounting for the surface roughness effect

We show the influence on the resistivity coefficient, C_D, of the inclusion in the computations of the surface roughness effect. As we read in ∮4.1, for $\mathcal{R}e \gtrsim 10^5$ the experiments show that the effect consists in displacing to lower Reynolds numbers the crisis of resistivity. In order to take into account this effect, we use fig. 8, taken from Achenbach [11] that plots C_D vs. $\mathcal{R}e$ for various surface roughness ratios, for the spherical case. Of course, the roughness parameter in our case is $(\triangle x/D)$, where $\triangle x$ is the mesh size and D is the diameter, respectively, of the sphere or of the circular cylinder considered. We then correct our results by computing the effect of roughness and compare the results with and without roughness effect, as can be seen in fig. 9. The improvement in the range $\mathcal{R}e \in [2 \cdot 10^5, 2 \cdot 10^6]$ is dramatic. The error $\triangle C_D/C_D$ is reduced up to a factor ≈ 8. As we see from Achenbach [11] (fig. 2) the resistivity coefficient C_D is practically unaffected by the surface roughness for $\mathcal{R}e \lesssim 10^5$, and we can in fact see this fulfilled in our figures in the mentioned range. To summarize, including the surface roughness effect (induced by the "stepwise" numerical simulation of the surface of the cylinder and of the sphere) the experimental determinations of the drag coefficient have been reproduced within a max error of ≈ 25 % and an average error of ≈ 6 % for the sphere. The position of the resistivity dip, as seen from fig. 9 as determined from the experimental curves, has been reproduced within displacement errors - in Reynolds number - roughly around ≈ 25 % for the sphere. Considering the boundary correction reduces said errors to roughly around a few percent for the spherical case as well.

* * *

This work was supported by C.N.R., the research funds of the Astronomical Observatory of Bologna and by a Grant n. 92-3-102-6 of the C.I.N.E.C.A center in Casalecchio (Bologna). A smaller contribution came from the research funds of the Astronomical Observatory of Rome.

REFERENCES

[1] A. Di Fazio, *Astron. Astrophys.*, **159**, (1986) 49.

[2] R. Capuzzo-Dolcetta, A. Di Fazio, and A.B. Men'shchikov, *Physical Processes in Fragmentation and Star Formation*, edited by R. Capuzzo-Dolcetta A. Di Fazio, C. Chiosi (Heilderberg Publ. 1990) p. 303.

[3] P. Battinelli, R. Capuzzo-Dolcetta, A. Di Fazio, V.A. Urpin and S.V. Vereshchagin, *Astron. Astrophys.*, **254**, (1990) 331.

[4] A. Di Fazio, A.V. Soloviev, V.A. Urpin and S.V. Vereshchagin, *Astronomicheskij Zhurnal*, **72**, (1993) 145.

[5] R. Bedogni and P.R. Woodward, *Astron. Astrophys.*, **231**, (1990) 481.

[6] C.F. Mc Kee, *Supernova Remnants and the Interstellar Medium IAU Colloqium 101*, edited by R.S. Roger, T.L. Landecker, (Cambridge University Press 1988) p. 205.

[7] N. Sedov, *Fizika Zhidkostej*, Nauka, Moscow (1954).

[8] N. Sedov, *Similitude et Dimensions en Mecanique*, MIR, Moscow (1965).

[9] P.P. Wegener and H. Ashkenas, *J. Fluid. Mech.*, **10**, (1961) 550.

[10] A.B. Bailey, *J. Fluid. Mech.*, **65**, (1974) 401.

[11] E. Achenbach, *J. Fluid. Mech.*, **65**, (1974) 113.

[12] D.G. Miller and A.B. Bailey, *J. Fluid. Mech.*, **93**, (1979) 449.

[13] A.E. Bryson and R.W.F. Gross, *J. Fluid. Mech.*, **10**, (1961) 1.

[14] H. Hanazaki, *J. Fluid. Mech.*, **192**, (1988) 393.

[15] B. Fornberg, *J. Fluid. Mech.*, **190**, (1988) 471.

[16] P. Colella and P.R. Woodward, *J. Comp. Phys.*, **54**, (1984) 174.

[17] P.R. Woodward and P. Colella, *J. Comp. Phys.*, **54**, (1984) 115.

[18] B.K. Edgar and P.R. Woodward, *UMSI Report 92/234*(1992).

[19] L.D. Landau and E.M. Lifshitz, *Fluid Mechanics*, Pergamon Press (1987).

[20] G.K. Batchelor, *An Introduction to Fluid Dynamics*, Cambridge University Press (1991).

[21] D.H. Porter and P.R. Woodward, *Astrophys. J. Suppl. Series*, **93**, (1994) 309.

Figure Captions

Figure 1 The numerical grid near the sphere in cylindrical coordinates, where the circles mark the boundary for the x sweep, while the crosses mark the boundary for the y (or r) sweep.

FIGURE 2A-D Grey level-shaded contour plots of $log_{10}\rho$ for the sphere : $\mathcal{M}_s = 1.2, \mathcal{M} = 0.28$ and $\Delta x = 0.02$. Range : 15 contours from 0 to 0.148; time sequence: a) t=8.33, b) t=16.6, c) t=24.9, d) t=33.3.

FIGURE 3A-D Grey level-shaded contour plots of the entropy $\mathcal{H} = ln(p/\rho^\gamma)$ for the sphere : $\mathcal{M}_s = 1.2, \mathcal{M} = 0.28$ and $\Delta x = 0.02$. Range : 15 contours from -0.336 to -0.275; same time sequence of figure 2.

FIGURE 4A-D Grey level-shaded contour plots of $log_{10}\rho$ for the sphere : $\mathcal{M}_s = 3, \mathcal{M} = 1.35$ and $\Delta x = 0.02$. Range : 15 contours from -0.303 to 0.914; time sequence a) t=3.33, b) t=6.66, c) t=9.99, d) t=13.3.

FIGURE 5A-D Grey level-shaded contour plots of the entropy $\mathcal{H} = ln(p/\rho^\gamma)$ for the sphere : $\mathcal{M}_s = 3, \mathcal{M} = 1.35$ and $\Delta x = 0.02$. Range : 15 contours from -0.336 to 1.204; same time sequence of figure 4.

FIGURE 6 The drag coefficient C_D versus $log_{10}\mathcal{R}e$ for the sphere at ($\mathcal{M}_s = 1.2, \mathcal{M} = 0.28$), ($\mathcal{M}_s = 1.5, \mathcal{M} = 0.60$), ($\mathcal{M}_s = 2.0, \mathcal{M} = 0.96$), ($\mathcal{M}_s = 4.0, \mathcal{M} = 1.55$). The solid and dashed curves represent results obtained in two different wind tunnels (see Sedov, 1954, 1965 and Bailey, 1974) with $\mathcal{M} \simeq 0.3$. From Miller and Bailey (1979) $- \cdots -$ $\mathcal{M} = 0.6$, $\cdots$ $\mathcal{M} = 1.0$, $- \cdot - \cdot -\cdot$ $\mathcal{M} = 1.4$.

FIGURE 7 Adapted from figure 4 from Miller and Bailey (1979) page 456. The symbol $\mathcal{M}_\infty$ of the original figure corresponds to our local Mach number $\mathcal{M}$.

FIGURE 8 This is the reproduction of figure 2 from Achenbach (1974) page 116. Summary of sphere drag measurements at high Reynolds numbers $0.2 \lesssim \mathcal{M} \lesssim 1.8$. Parameter : surface roughness. Solid line, smooth ; $\times$ $k/d_s = 25 \cdot 10^{-5}$; $\diamondsuit$ $k/d_s = 150 \cdot 10^{-5}$; $\bigcirc$ $k/d_s = 250 \cdot 10^{-5}$; $\triangle$ $k/d_s = 500 \cdot 10^{-5}$; $\square$ $k/d_s = 1250 \cdot 10^{-5}$.

FIGURE 9 The drag coefficient C_D versus $log_{10}\mathcal{R}e$ for the sphere at $\mathcal{M}_s = 1.2, \mathcal{M} = 0.28$. Comparison with roughness corrected C_D.

FIGURE 10 The drag coefficient C_D versus time normalized to the τ crossing time for increasing grid resolutions: $\triangle x = 0.2$; $\triangle x = 0.058$; $\triangle x = 0.04$; $\triangle x = 0.02$; $\triangle x = 0.015$; $\triangle x = 0.01$.

FIGURE 11 The drag coefficient C_D for the sphere versus the local Mach number $\mathcal{M}$ for resolutions $\triangle x = 0.018$ (circle), 0.015 (triangle) , 0.012 (cross), 0.010 (diamond).

FIGURE 12 The ratio $E_{tur}/\mathcal{L}_{tur}$ versus $log_{10}\mathcal{R}e$ for the sphere at ($\mathcal{M}_s = 1.2$,$\mathcal{M} = 0.28$), ($\mathcal{M}_s = 1.5$,$\mathcal{M} = 0.60$), ($\mathcal{M}_s = 2.0$,$\mathcal{M} = 0.96$), ($\mathcal{M}_s = 4.0$,$\mathcal{M} = 1.55$). Same coding of figure 6 for the symbols.

FIGURE 13A-C The transverse kinetic linear scale K_{tur} in the wake of the sphere versus z for ($\mathcal{M}_s = 1.2, \mathcal{M} = 0.28$) and $\mathcal{R}e = 2.7\times10^4, 2.1\times10^5, 5.2\times10^5, 1.7\times10^6$, ($\mathcal{M}_s = 2.0, \mathcal{M} = 0.96$), and $\mathcal{R}e = 7.1\times10^3, 5.7\times10^4, 1.3\times10^5, 4.5\times10^6$ ($\mathcal{M}_s = 4.0, \mathcal{M} = 1.55$). and $\mathcal{R}e = 3.3\times10^3, 2.6\times10^4, 6.2\times10^4, 2.1\times10^5$.

FIGURE 14A-C The main scale of turbulence L_{tur} in the wake of the sphere versus z for the same Mach and Reynolds numbers of figure 15.

FIGURE 15A-C The main pulsation $\triangle u_{rms}$ (in units u_∞) versus z in the wake of the sphere for the same Mach and Reynolds numbers of figure 15.

FIGURE 16A-C The main pulsation $\triangle u_{rms}$ (in units u_∞) versus the main scale of turbulence L_{tur} in the wake of the sphere for the same Mach and Reynolds numbers of figure 15.

FIGURE 17A-C The turbulent energy density ϵ_{tur} (in units ϵ_∞) versus the main scale of turbulence L_{tur} in the wake of the sphere for the same Mach and Reynolds numbers of figure 15.

FIGURE A1 The drag coefficient C_D in the neighborhood of the "critical" zone, versus $log_{10}\mathcal{R}e$, for the sphere at $(\mathcal{M}_s = 1.2, \mathcal{M} = 0.28)$ compared with the roughness correction.