EULER-EULER MODELLING OF THE INTERACTION OF A GAS-PARTICLE MIXTURE WITH A DETACHED SHOCK

G Marois, Philippe Villedieu, Julien Mathiaud

13th World Congress on Computational Mechanics (WCCM XIII) and 2nd Pan American Congress on Computational Mechanics (PANACM II), July 22-27, 2018, New York City, NY, USA

EULER-EULER MODELLING OF THE INTERACTION OF A GAS-PARTICLE MIXTURE WITH A DETACHED SHOCK

G. MAROIS*†, P. VILLEDIEU#†, J. MATHIAUD*

gentien.marois@cea.fr
philippe.villedieu@onera.fr
julien.mathiaud@cea.fr

*CEA CESTA, 14 Avenue des Sablières, Le Barp, France
†INSA-TOULOUSE , 135 Avenue de Rangueil , Toulouse, France
#ONERA/DMPE, Université de Toulouse, F-31055 Toulouse (France)

Key words: Multiphase flows, Gas-particle mixtures, Shock Disturbance.

Abstract. This work deals with the interaction of a mist of solid particles and a stationary detached shock wave. Experiments over a 3 inches sphere were done in the 70's at the Boeing Hypersonic Wind Tunnel[1]. A very strong modification of the detached shock was observed. The aim of the present work was to check the capability of numerical simulations based on an Euler-Euler approach to reproduce this behavior. Comparisons with Hulin and Znaty[2] simulations (performed with an Eulerian-Lagrangian approach) have been carried out and the results are discussed.

1 INTRODUCTION

Studies of gas-particle two-phase flows (both subsonic and supersonic) are fairly recent and complex since they involve a lot of additional physical phenomena compared to single-phase flows. The work presented here deals with the interaction of a stationary detached shock wave with a mist of spherical particles (that are solid particles in the scope of the present study). Such interactions are encountered in a large range of scientific and engineering applications, from planetary explorations to motors design.

During last decades, several studies have been devoted to this problem. Analytical works were done for example by Carrier[3] or Marble[4]. Experimental investigations were performed, for example in Boeing laboratories[1,5] and were completed by numerical simulations (Papadopoulos[6], Palmer[7]). These numerical studies were all based on the so-called Eulerian-Lagrangian method which consists in an Eulerian treatment of the gas phase and a Lagrangian treatment of the particulate phase. This approach is very popular since it can be quickly implemented and allows to easily take into account complex physical phenomena like particle breakup, complex wall-particle interactions, etc. However, when one is interested in the rate of particle impact on a surface or the density of particles in the vicinity of an obstacle, this approach requires the use of a very large number of numerical particles to get accurate results. In addition, due to load balancing issues, this approach is very difficult to implement efficiently on a massively parallel computer.

To overcome this limitation, in the present work, the Eulerian-Eulerian approach has been preferred even if it is more complex both from a numerical and a physical point of view. A finite volume method has been used to solve the equations of both phases. The particulate phase being very diluted in all targeted applications, its volume fraction (but not its mass density) is assumed to be negligible and its influence on the gas flow is taken into account via source terms in the right-hand side of the gas equations.

The paper is divided into four parts. First, the model is presented. We then detail the numerical method used as well as some results of validation tests carried out to ensure that the method has been correctly implemented and gives the expected results in simple but representative cases of application. The last part is devoted to attempts to replicate the Boeing experiment of the 70's concerning the interaction of a shock wave with a mist of particles.

2 PROBLEM FORMULATION

2.1 Eulerian model for the gas

The gas phase is modeled by the Navier-Stokes equations adapted for reacting gases (including vibrational energy effects, chemical reactions of dissociation, etc.). More details can be found in [8]. These equations can be schematically written as:

$$\partial_t \mathbf{U_g} + \partial_x \mathbf{G}(\mathbf{U_g}, \nabla \mathbf{U_g}) = \mathbf{S_{ext}} + \mathbf{S_{ch}} + \mathbf{S_p} \qquad (1)$$

where $\mathbf{U_g}$ denotes the vector of the gas phase conservative variables, G corresponds to the flux terms, $\mathbf{S_{ch}}$ corresponds to the chemical reaction source terms, $\mathbf{S_p}$ denotes the source terms that takes into account the influence of the particles on the gas flow, and $\mathbf{S_{ext}}$ is a source term that takes into account other effects like external forces, etc.

2.2 Eulerian model for the particles

In this paper, we limit ourselves to the case where the dispersed phase is composed of solid spherical particles whose only interactions with the gas phase are the exchange of momentum (via the drag force) and heat (by forced convection). However, the model presented below as well as its numerical treatment can be adapted to deal with more general cases (in particular the case of liquid particles), for which other phenomena must be taken into account like particle evaporation, sublimation, break-up, etc.[18]

There are several variants of the Eulerian approach for the treatment of the particulate phase (see for example [6, 7]). Here, we have chosen the sampling method [9] which has the advantage of being the simplest to implement. It consists of dividing the particulate phase into *N* classes, each characterized by its number density field $n_i(t, \boldsymbol{x})$, its mass density field $m_i(t, \boldsymbol{x})$, its velocity field $\mathbf{v}_i(t,\boldsymbol{x})$ and its temperature field $T_i(t,\boldsymbol{x})$. Assuming that all the particles belonging to the same class and located at the same point have the same velocity, same diameter and same temperature, the following set of conservation equation can be easily derived:

$$\begin{cases} \partial_t n_i + \nabla_x (n_i \, \mathbf{v_i}) = 0 \\ \partial_t m_i + \nabla_x (m_i \, \mathbf{v_i}) = 0 \\ \partial_t m_i \, \mathbf{v_i} + \nabla_x (m_i \, \mathbf{v_i} \otimes \mathbf{v_i}) = n_i \mathbf{F_i} \\ \partial_t m_i \, h_i + \nabla_x (m_i \, \mathbf{v_i} \, h_i) = n_i \, \mathrm{H_i} \end{cases} \qquad (2)$$

where $h_i = h_p(T_i)$ denotes the specific enthalpy of the particles of class i (supposed to be very close to their specific internal energy $e_p(T_i)$), **Fi** denotes the drag force acting on the particles of class i and Hi correspond the convective heat flux for the particles of class i. The drag force is given by:

$$\mathbf{F_i} = -\frac{\pi}{8}\, d_i^2\, \rho_f\, C_D\, \|\mathbf{v_i} - \mathbf{u_g}\|\, (\mathbf{v_i} - \mathbf{u_g}) \qquad (3)$$

where C_D is the drag coefficient. The particle mass density m_i, the particle number density n_i, the particle diameter d_i and the particle temperature T_i are linked by the following relationship:

$$m_i = \frac{\pi}{6} n_i \rho_p(T_i) d_i^3 \qquad (4)$$

where ρ_p denotes the particulate phase bulk density (supposed to depend only on the temperature). The drag coefficient C_D depends on the particle Reynolds number Re_i and particle Mach number M_i. It can be calculated by combining Swain [10] with Clift and Gauvin [11] empirical formula:

$$\begin{cases} C_D^1(M_i, Re_i) & \text{for} \quad M_i < 0.6 \\ C_D^2(M_i, Re_i) & \text{for} \quad M_i > 1.3 \\ C_D^1(0.6, Re_i) + \dfrac{M_i - 0.6}{0.7}\,\left(C_D^2(1.3, Re_\infty) - C_D^1(0.6, Re_i)\right) & \text{elsewhere} \end{cases}$$

with: (5)

$$\begin{aligned} C_D^1(M_i, Re_i) &= \frac{24}{Re_i}\,\left(1 + 0.15\, Re_i^{0.687}\right) + \frac{0.42}{1 + 42500\, Re_i^{-1.16}} \\ C_D^2(M_i, Re_i) &= 1 + 4.66\, Re_i^{-0.5} \end{aligned} \qquad (6)$$

Introducing the Nusselt number, Nu_i, the heat flux $\mathrm{H_i}$ can be defined as:

$$\mathrm{H_i} = \lambda_g\, \pi\, Nu_i\, d_i\, \left(T_g - T_i\right) \qquad (7)$$

with the Nusselt number being calculated thanks to Fox formula [12] which is an extension of classical subsonic fits [13] to supersonic flows:

$$Nu_i = \frac{2\, \exp(-M_i)}{1 + 17\, M_i/Re_i} + 0.459\, Pr^{0.33}\, Re_i^{0.55}\, \frac{1 + 0.5\, \exp\left(-17\, M_i/Re_i\right)}{1.5} \qquad (8)$$

The particulate source term in the r.h.s. on the gas phase balance equations can be easily deduced from the conservation of momentum and energy. It writes:

$$\mathbf{S_p} = -\begin{pmatrix} \mathbf{0} \\ \sum_{i=1}^{N} n_i \mathbf{F_i} \\ \sum_{i=1}^{N} n_i \mathbf{F_i} \cdot \mathbf{v_i} + n_i \, \mathrm{H_i} \end{pmatrix} \qquad (9)$$

where $n_i \, \mathbf{F}_i \cdot \mathbf{v_i}$ corresponds to the power of the drag force. This term does not appear in the particles' equations (2) because the last equation involves the particle specific enthalpy (or G. internal energy) instead of the particle total energy. It is worth mentioning that this formulation is adapted even in the case of supersonic flows because the fields associated with the particulate phase is not discontinuous through shocks (due to the finite relaxation time scales of the particles). If the gas equations are coupled with a turbulence model, it is also necessary, at least from a theoretical point of view and for consistency reasons, to take into account the influence of the particulate phase on turbulence production and dissipation by adding source terms in the turbulence model equations. We will come back to this question in section 5.3.

In most applications, particle interactions with solid walls play a fundamental role. In the Eulerian approach, since all the particles of a given class and located at the same point are supposed to have the same velocity (no pressure term in the momentum equation contrary to the gas phase), it is mandatory to introduce additional classes to deal with the secondary particles created during the impact of particles onto a wall (due to rebound, fragmentation and erosion). The number of additional classes will depend on the complexity of the impact model. In the present work, for the sake of simplicity, we have only considered the simplest model which consists of allocating only one secondary class to each primary class.

Let's denote by i_r the index of the class of secondary particles corresponding to the class i_p of primary particles. At the beginning of the calculation, there is no particle in class i_r (which means that $m_{ir} = 0$ in all mesh cells). When particles of class i_p impinge the wall, new particles are created in the corresponding class i_r. In the present work, the following model was used to compute the mass flux of the reemitted particles, their velocity, their diameter and their temperature:

$$\begin{cases} \mathbf{v_{i_r}} \cdot \mathbf{n} = \epsilon_n \, \mathbf{v_{i_p}} \cdot \mathbf{n} \\ \mathbf{v_{i_r}} \cdot \mathbf{t} = \epsilon_t \, \mathbf{v_{i_p}} \cdot \mathbf{n} \\ \epsilon_n m_{i_r} = G \, m_{i_p} \\ \epsilon_n n_{i_r} = F \, n_{i_p} \\ T_{i_r} = H T_{i_p} \end{cases} \qquad (10)$$

G is a mass gain parameter which allows to take into account the creation of new particles by erosion effects ($G = 1$ corresponds to mass conservation during impact). F is a parameter which accounts for the combined effect of erosion and fragmentation (which both lead to the creation of new particles and thus to $F > 1$). The parameter H allows to take into account the energy transfer during the particle impact. ε_n and ε_t correspond to the normal and tangential restitution coefficients. Naturally a model has to be prescribed for G, F, H, ε_n and ε_t.

3 NUMERICAL TREATMENT OF THE PARTICULE PHASE EQUATIONS

The generalized Navier-Stokes equations for the gas phase are solved using a classical finite volume scheme on structured meshes that will not be described here. Details can be found for example in [14,15,16].

Regarding the spatial discretization of (2), the following finite volume method has been used. For each class, let's first introduce the vector $\mathbf{V}_i$ of specific variables defined as follows:

$$\mathbf{V_i} = (n_i/m_i,\ 1,\ \mathbf{v_i},\ h_i)^t \qquad (11)$$

Using this notation, system (2) can be rewritten under the following generic form:

$$\partial_t\, m_i\, \mathbf{V_i} + \nabla_x\, (m_i\, \mathbf{v_i}\, \mathbf{V_i}) = \mathbf{S_i} \qquad (12)$$

which is more suited for the numerical discretization. Introducing $|K|$ the volume (or area) of a given control volume K and $|e|$ the area (or length) of the cell edge e, and integrating (12) on K leads to the semi-discretized equation:

$$|K|\, d_t\, (m_i\, \mathbf{V_i}) + \sum_{e\, \in \partial K} \Phi_{ie,K} |e| = |K|\, \mathbf{S_i} \qquad (13)$$

where $\mathbf{\Phi}_{ie,K}$ denotes the flux through the edge e. Let's define the edge mean particle velocity (for the class of particles i) by:

$$\mathbf{v_{ie}} = \frac{m_{i,K}\, \mathbf{v_{i,K}} + m_{i,Ke}\, \mathbf{v_{i,Ke}}}{m_{i,K} + m_{i,Ke}} \qquad (14)$$

Applying an upwind scheme based on the sign of $\mathbf{v}_{ie}\cdot\mathbf{n}_{e,K}$ the expression of the flux $\mathbf{\Phi}_{ie,K}$ reads:

$$\Phi_{ie,K} = v^+_{ie,K}\, \mathbf{V_{i,K}} + v^-_{ie,K}\, \mathbf{V_{i,Ke}} \qquad (15)$$

where $v^+_{ie,K}$ and $v^-_{ie,K}$ denote the positive (respectively negative) part of $\mathbf{v}_{ie}\cdot\mathbf{n}_{e,K}$. In addition, this space discretization scheme can be combined with the MUSCL approach [15,17] to get a second order accurate scheme.

As far as the time discretization is concerned, a first order semi-implicit scheme has been used which consists of using an explicit scheme for the flux terms and a partially implicit scheme for the treatment of the source terms. An advantage of this method is that, under a CFL-like condition, it can be shown to insure the positivity of the number density and to satisfy a maximum principle on the components of $\mathbf{V}_i$ (which are directly related to the particle velocity, temperature and diameter).

Regarding the treatment of the particle source term in the gas phase equations, $\mathbf{S}_p$, a relaxation method has been implemented to enforce the stability of the coupling in dense zones.

4 BASIC VALIDATION TESTS

Several tests case were carried out in order to check the correct implementation of the above described numerical model and to assess its capability to accurately reproduce basic interactions between a gas flow and a cloud of particles. Here, for the sake of concision, we only present a one-dimensional test case (but performed using a 2D mesh).

The test consists of injecting gas and particles in a tube in non-equilibrium conditions. The gas flow at the inlet being supersonic, the whole gas and particle states are prescribed at the inlet. No boundary condition is imposed at the outlet. The gas is supposed to be perfect and inviscid and heat conduction effects are neglected. The objective of the test case it to assess the capability of the code to calculate the relaxation towards the equilibrium between the two phases along the tube.

In the first variant of the test, we only focus on the solution at the outlet of the tube which is assumed to be long enough for the particles and the gas to be at equilibrium at the output. By using the conservation of particle mass, particle number, gas mass, global momentum and global energy, it is possible to compute analytically the expression of the equilibrium velocity, pressure and temperature at the tube outlet for any inlet conditions. This calculation was done and numerical tests were performed for several inlet conditions. For all cases, the numerical results were in perfect agreement with the theoretical solution, showing that the global conservation properties are correctly ensured by the code. Due to the lack of room, the results of these tests will not be shown here.

The second variant of the test consists in computing the steady state solution according to the abscissa x along the tube. Solving analytically the full non-linear system is not possible. However, it is possible to compute the analytical solution of the linearized system around an equilibrium state. Each quantity q can be written as the sum of its equilibrium value $\bar{q}$ and a small perturbation $\hat{q}$. In the simplest case when it is possible to neglect the heat transfer between the gas and the particles, the solution of the linearized system reads:

$$\begin{cases} \widehat{\rho_g} = -\dfrac{\overline{m_p}}{\overline{u}} \dfrac{M^2}{M^2-1+\epsilon M^2} \left[\widehat{u_p}(0) - \widehat{u_g}(0)\right] \left\{1 - \exp\left[\dfrac{-x}{\tau_{\mathcal{F}}\,\overline{u}} \left(1+\epsilon \dfrac{M^2}{M^2-1}\right)\right]\right\} \\ \widehat{m_p} = \dfrac{\overline{m_p}}{\overline{u}} \dfrac{M^2-1}{M^2-1+\epsilon M^2} \left[\widehat{u_p}(0) - \widehat{u_g}(0)\right] \left\{1 - \exp\left[\dfrac{-x}{\tau_{\mathcal{F}}\,\overline{u}} \left(1+\epsilon \dfrac{M^2}{M^2-1}\right)\right]\right\} \\ \widehat{u_g} = \widehat{u_g}(0) + \dfrac{\epsilon M^2}{M^2-1+\epsilon M^2} \left[\widehat{u_p}(0) - \widehat{u_g}(0)\right] \left\{1 - \exp\left[\dfrac{-x}{\tau_{\mathcal{F}}\,\overline{u}} \left(1+\epsilon \dfrac{M^2}{M^2-1}\right)\right]\right\} \\ \widehat{u_p} = \widehat{u_p}(0) - \dfrac{M^2-1}{M^2-1+\epsilon M^2} \left[\widehat{u_p}(0) - \widehat{u_g}(0)\right] \left\{1 - \exp\left[\dfrac{-x}{\tau_{\mathcal{F}}\,\overline{u}} \left(1+\epsilon \dfrac{M^2}{M^2-1}\right)\right]\right\} \\ \widehat{p_g} = -\dfrac{\gamma\,\overline{p_g}\,\epsilon}{\overline{u}} \dfrac{M^2}{M^2-1+\epsilon M^2} \left\{1 - \exp\left[\dfrac{-x}{\tau_{\mathcal{F}}\,\overline{u}} \left(1+\epsilon \dfrac{M^2}{M^2-1}\right)\right]\right\} \end{cases} \tag{16}$$

where $\overline{m_p}$ is the equilibrium particle mass density, $\overline{p_g}$ is the equilibrium pressure, $\bar{u}$ is the equilibrium velocity of both phases, $M = \dfrac{\bar{u}}{\sqrt{\frac{\gamma \bar{p}_g}{\bar{\rho}_g}}}$ is the corresponding Mach number, γ is the classical heat capacity ratio, $\varepsilon = \frac{\bar{m}_p}{\bar{\rho}_g}$ is the equilibrium mass loading ratio and $\tau_{\mathcal{F}} = \frac{\bar{\rho}_p d^2}{18\mu_g}$ is the dynamic relaxation time of the particles (Stokes regime) and μ_g is the viscosity.

Figures 1 show the density and velocity profiles for the two phases. The dashed line corresponds to the asymptotic solution given by the global conservation equations. It can be seen that the numerical results (full line) agree very well with the two analytical profiles.

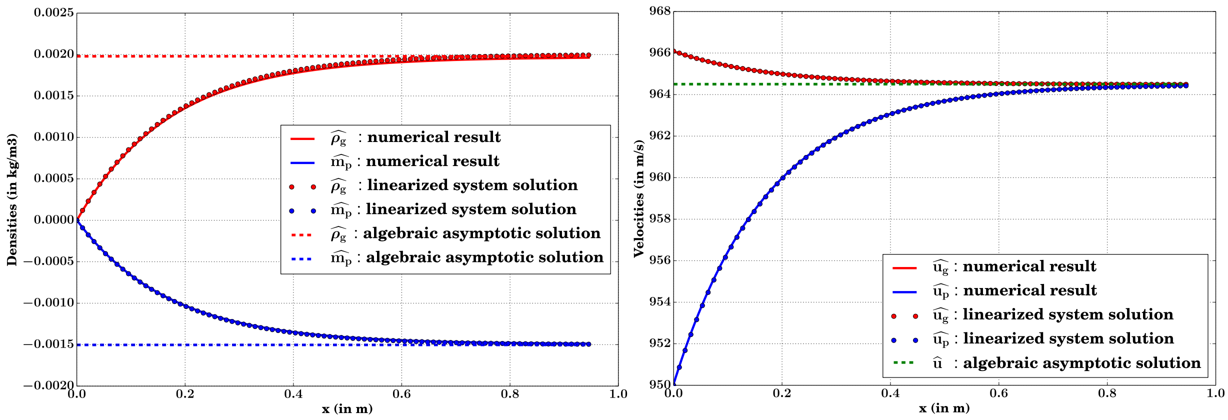

Figure 1: Left: evolution of the gas density and particle mass density along the tube at the steady state. Right: evolution of the gas and particle velocity along the tube at the steady state. Solid lines: numerical results - Dotted lines: theoretical solution - Green dashed line: equilibrium theoretical solution.

5 SIMULATION OF BOEING HYPERSONIC WIND TUNNEL EXPERIMENT

5.1 Experimental conditions

In the 70's experiments were carried out at the Boeing Hypersonic Wind Tunnel (BHWT) [1]. In these experiments, a solid metallic sphere was plunged into a Mach 6.1 free stream flow containing silica particles. In table 1, the flow stagnation conditions (at the input of the tunnel) and the infinity flow conditions (inlet of our computational domain) of the considered test run are summarized:

	Stagnation flow conditions	Infinity flow conditions
Pressure	$44.8\ 10^5$ Pa	2564 Pa
Temperature	633.1 K	75 K
Density	24.6 $kg.m^{-3}$	0.12 $kg.m^{-3}$
Sound speed	504.4 $m.s^{-1}$	173.6 $m.s^{-1}$

Table 1: Stagnations and infinity flow conditions

The silica particles injected in the flow were of 100 µm diameter. In the following, c_∞ will denote the ratio between the particles mass flow rate and the gas mass flow rate. Its value was:

$$c_\infty = \frac{\dot{m}_{part}}{\dot{m}_{gas}} = 7.3\,10^{-4} \qquad (17)$$

A strong shock disturbance was observed due to the presence of the particles, especially in the vicinity of the symmetry axis. The objective of the numerical simulations presented hereafter was to check the ability of model (1)-(2)-(10) to capture this phenomenon and, as far as possible, to use the results to better understand its physical origin.

5.2 Modeling choices of HULIN and ZNATY[2]

Numerical simulations of BHWT experiments have already been successfully performed by Hulin and Znaty [2] but using a Lagrangian approach for the particulate phase. These researchers performed two types of simulation. In the first case, each numerical particle was associated with a real particle so as to account for the discrete nature of the solid phase and the fact that in the experiment the average distance between two neighboring particles is in the order of a few mm which is far from being negligible in the considered application. In the second case, the number of numerical particles that they used was much larger than the number of real particles. Each numerical particle was therefore assigned a weight that could be associated with a probability of presence. This second method, for which the concentration field of the particulate phase is quasi-continuous is therefore very close to an Eulerian approach. As Hulin and Znaty say in their paper that they get the same results in both cases, we expected to obtain comparable results with the Eulerian approach.

Hulin and Znaty have proposed a set of hypotheses to take into account the influence of fragmentation and erosion phenomena due to particle impacts. According to their model, after an impact,

- each incident particle is supposed to fragment into 5 smaller particles ;
- erosion leads to the creation of new particles with the same characteristics as the particles created by fragmentation of the incident particles ; the total mass of the eroded particles is supposed to be four times the mass of the incident particles ;
- the total kinetic energy of the reemitted particles (initial particle fragments + eroded particles) is supposed to represent 30% of the incident kinetic energy ;
- the restitution parameters, ε_n and ε_t are supposed to have the same value.

These assumptions lead to the following values for the impact model parameters (see equation (10)): G = 5, F = 25, $\varepsilon_n = \varepsilon_t = 0.24$. Since no assumption was mentioned in [2] regarding thermal effects, we simply assumed that H=1 for the sake of simplicity, even if it is not realistic from a physical point of view.

5.3 Results

The classical k-ω model[20] was used to account for the effect of turbulence on the gas flow mean properties. We performed two different numerical simulations with the particles. In the first simulation, we did not take into account any influence of the particles on the turbulence production and dissipation. In the second simulation, the turbulence source term model proposed by Hulin and Znaty [2] was used. As far as the turbulent kinetic energy source term is concerned, their model reads:

$$S_p^k = \sum_{i=1}^{N} n_i \, \mathbf{F_i} \cdot (\mathbf{v_g} - \mathbf{v_i}) \qquad (18)$$

It is worth noticing that this model is not correct as it involves the mean slip velocity between the gas and the particles instead of the gas fluctuating velocity as it should be the case from a theoretical point of view. The correct expression should be:

$$S_p^k = \sum_{i=1}^{N} n_i \, \overline{\mathbf{F}_i' \cdot \mathbf{v}_g'} \qquad (19)$$

Since the mean slip velocity is very high behind the shock, model (18) necessarily leads to a very high production rate of turbulent kinetic energy. It is thus not surprising that the presence of particles has a strong influence on the turbulence level, as noticed by Hulin and Znaty.

We point out that the calculations made in this paper do not reach a steady state. Indeed, the particles indefinitely accumulate along the shock. This phenomenon will be discussed later. This accumulation makes difficult the establishment of a steady state. This is why the results presented below are snapshots taken during the calculation.

Figures 2 show a comparison of the obtained numerical results for the gas phase density field without particles (left), with particles but without any turbulence production by the particle (middle), with particles and turbulence production by the particles using model (18) (right). Figures 3 and figures 4 show a similar comparison but for the gas Mach number field and for the turbulent kinetic energy field respectively.

We can see that without any influence of the particles on the turbulence production rate, no shock disturbance is observed whereas a clear modification of the shock appears if model (18) is applied. According to these results, the shock disturbances that have been observed in BHWT experiments seem to result from the strong production of turbulence by the particles in the vicinity of the shock We recover similar conclusions as in [2] even if our numerical results do not exactly coincide with theirs. However, as already mentioned above, model (18) is not correct and is expected to strongly overestimate the turbulence production due to the presence of particles. We can therefore question the truth of this conclusion. To investigate this question, this will need at least to replace model (18) by a more correct one derived from (19) using closure assumptions. This will be the subject of future work.

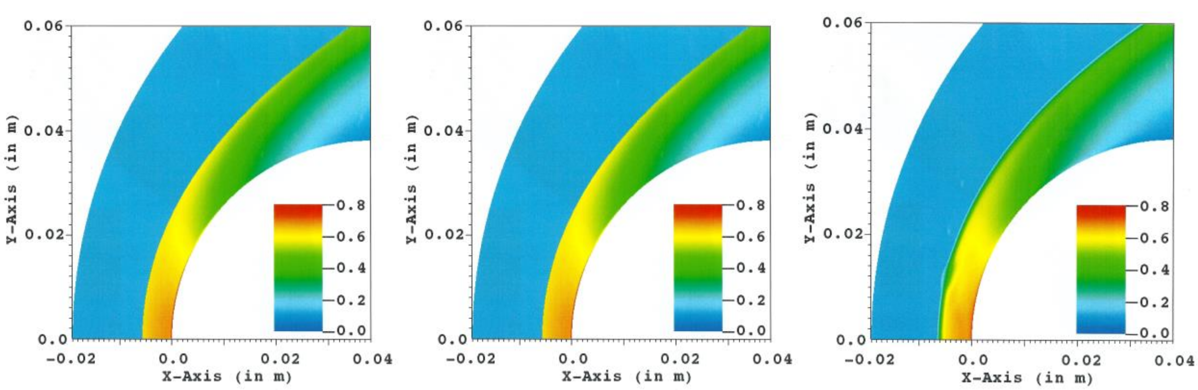

Figure 2: Gas phase density field (in kg.m^{-3}). Without particles (left) – With particles but without turbulence production by the particles (middle) – With particles and model (18) for turbulence production by the particles (right)

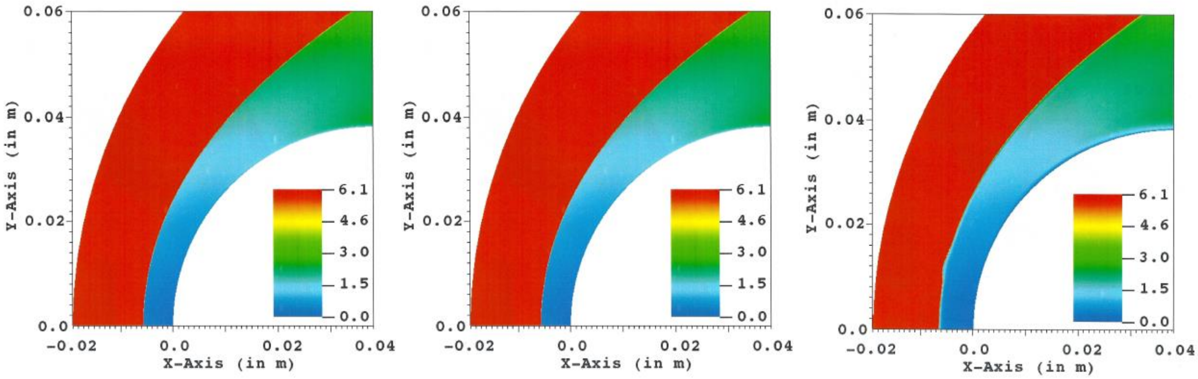

Figure 3: Gas Mach number field. Same legend as for Figure 2.

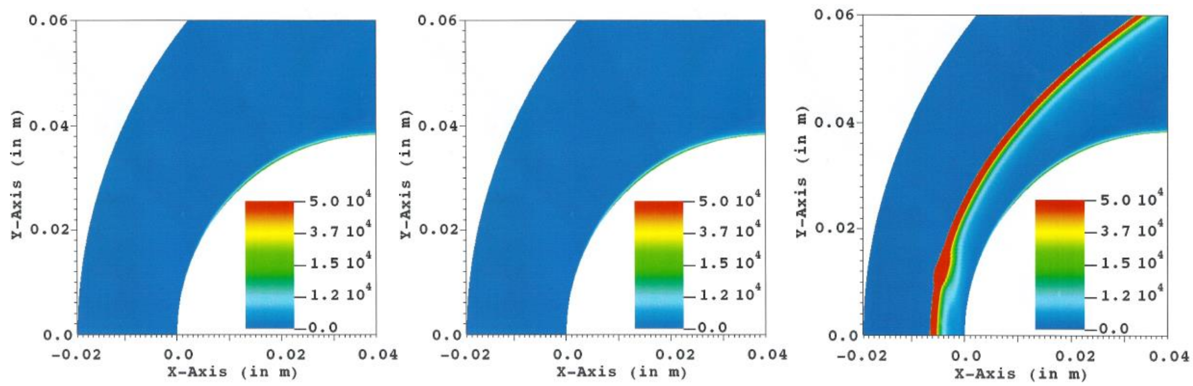

Figure 4: Turbulent kinetic energy field (in $kg.m^2.s^{-2}$). Same legend as for Figure 2.

Another interesting consequence of the interaction of the particles with the detached shock can be observed in Figures 5 on the particle mass density fields. The secondary particles that are reemitted from the wall due to fragmentation and erosion phenomena are strongly decelerated by the gas flow and tend to accumulate in the vicinity of the shock, leading to very high density of particles compared to the far field. Even if this phenomenon is certainly overestimated by the Eulerian treatment of the particle phase (well-known effect of single-velocity Eulerian model [19]), it is physically plausible and could also play a role in the shock disturbance mechanism.

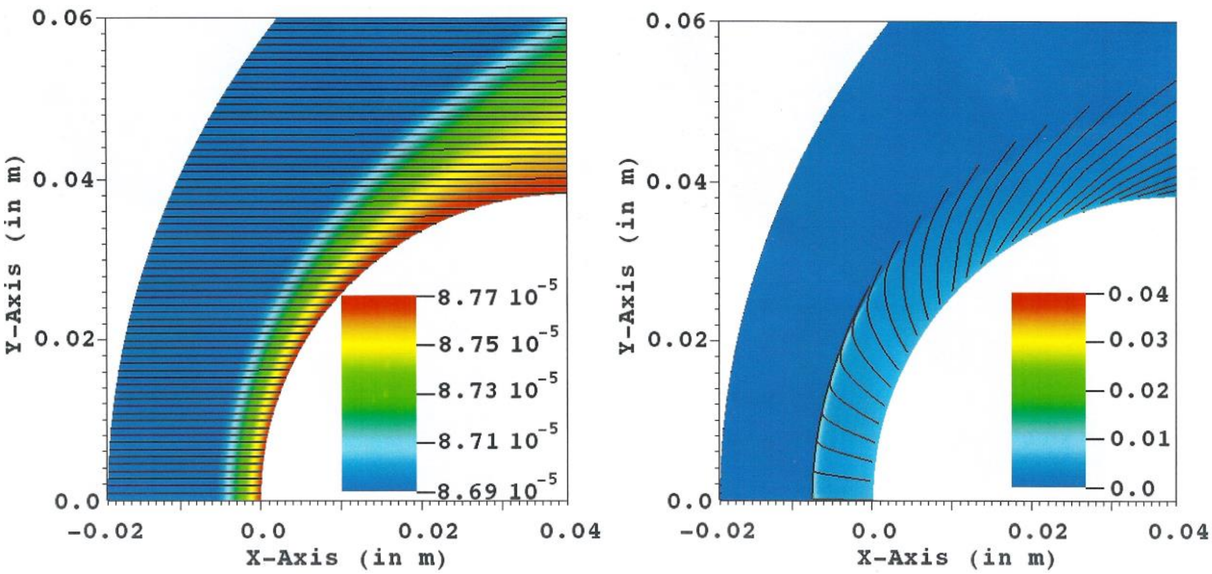

Figure 5: Particle mass density field (in kg.m^{-3}) and particle velocity streamlines. – Left : primary particle class – Right : secondary particle class

CONCLUSION

In this work, an Eulerian-Eulerian approach has been proposed to simulate particle laden hypersonic flows. It has been applied to the simulation of the BHWT experiment with the aim to reproduce the particle induced shock disturbances that were experimentally observed. We have obtained similar results as Hulin and Znaty[2] who already performed the same simulations in the past but using an Euler-Lagrange approach. In the numerical simulations, the influence of the particles on the turbulent kinetic energy production rate in the vicinity of the shock seems to play a determinant role for shock disturbances to appear. However the model used by Hulin and Znaty[2] (and that was used as well in the present study for the sake of comparison) is not correct from a theoretical point of view and tends to strongly overestimate the turbulence generation by the particles. The validity of the simulation results obtained with this model are thus strongly questionable. Further investigations are necessary to better understand the physical phenomena at the origin of the shock displacement and to ensure that Euler-Euler, and as well Euler-Lagrange models, are able to reproduce the experimental results.

REFERENCES

[1] W.FLEENER and R.WATSON. Convective heating in dust-laden hypersonic flows. In 8th Thermophysics Conference, page 761, 1973.
[2]A.HULIN and E.ZNATY. Aerodynamic modelling of hypersonic erosive reentry flows. In Aerothermodynamics for space vehicles, volume 367, page 391, 1995
[3] G.F.CARRIER. Shock waves in a dusty gas. Journal of Fluid Mechanics, 4(4):376–382, 1958.
[4] F.E.MARBLE. Dynamics of dusty gases. Annual Review of Fluid Mechanics, 2(1):397–446, 1970.
[5]L.DUNBAR, J.COURTNEY, and L.MCMILLEN. Heating augmentation in particle erosion environments. In 8th Aerodynamic Testing Conference, page 607, 1974.
[6]P.PAPADOPOULOS, M.E.TAUBER, and I.D.CHANG. Heatshield erosion in a dusty martian atmosphere. Journal of Spacecraft and Rockets, 1993.
[7]G.PALMER, Y.K.CHEN, P.PAPADOPOULOS, and M.TAUBER. Reassessment of effect of dust erosion on heatshield of mars entry vehicle. Journal of Spacecraft and Rockets, 37(6):747–752, 2000.
[8] J.D.ANDERSON Jr. Hypersonic and High Temperature Gas Dynamics. AIAA Publications, AIAA, Reston, VA, 2000.
[9] F.LAURENT and M.MASSOT. Multi-fluid modelling of laminar polydisperse spray flames: origin, assumptions and comparison of sectional and sampling methods, 2001.
[10] C.E.SWAIN. The effect of particle/shock layer interaction on reentry vehicle performance. 1975.
[11] R.CLIFT, J.R.GRACE, and M.WEBER. Bubbles, drops and particles. Academie,New York, 1978.
[12] TW FOX, CW RACKETT, and JA NICHOLLS. Shock wave ignition of magnesium powders. 1978.
[13]R.M.DRAKE, Discussion:"Forced Convection Heat Transfer From an Isothermal Sphere to Water"(Vliet, GC, and Leppert, G., 1961, ASME J. Heat Transfer, 83, pp. 163–170). *Journal of Heat Transfer*, *83*(2), 170-172, 1961
[14]W.K.ANDERSON, J.L.THOMAS, and B.VAN LEER. Comparison of finite volume flux vector splittings for the Euler equations. AIAA journal, 24(9):1453–1460, 1986.
[15] E. F.TORO. Riemann solvers and numerical methods for fluid dynamics: a practical introduction. Springer Science & Business Media, 2013.
[16] B.VAN LEER, J.L.THOMAS, P.L.ROE, and R.W.NEWSOME. A comparison of numerical flux formulas for the Euler and Navier-Stokes equations. 1987.
[17] B.VAN LEER. Towards the ultimate conservative difference scheme V: a second-order sequel to godunov's method. Journal of computational Physics, 32(1):101–136, 1979.
[18]G.MAROIS, PhD Thesis, Toulouse University, in preparation, 2018
[19]O.Desjardin, R.Fox, Ph.Villedieu. A quadrature-based moment method for dilute fluid-particle flows. Journal of Computational Physics, 227(4), 2514-2539, 2008
[20] D.C. WILCOX, Turbulence models for CFD. *DCW Industries, La Cañada, CA, EUA, 1998*