SD Numerical Simulation Technique for Hydrodynamic Flow Gas-Solids Mixing

Mantilla Núñez, Irla*[1] S. De Vicente C.[2]
[1] National University of Engineering, Lima, Perú, [2] Polytechnic University of Madrid.
*Lima 25, irlamn@uni.edu.pe.

Abstract: We formulate a new mathematical model of Gas-Solids Mixing hydrodynamic flow [1] in a combustion chamber with a fluid bed system used in the combustion of mineral coal waste. This model in study is called Model Gas-Solids Mixing and it is constructed by averaging the conservation equations (mass and momentum) for a two-phase flow, which takes into account the existence of a small parameter rho in the order of 10^{-4} . This parameter is related to the ratio of the mass densities of both phases and is a free boundary problem making an asymptotic adjustment in the model. This model is important in the production of thermal energy-based solid waste. The simulation is implemented in COMSOL Multiphysics.

Keywords: Non linear element finite, Laws conservation, Stream lines Diffusion.

1. Introduction

In this model, a nonlinear term is added to the Navier-Stokes equations to take into account the effect of nonlinearity in the compressible flow. In this problem in particular during the process empty areas of particles called singularities appear, ie. when the volume fraction of the dispersed phase is zero caused for the gas velocity. This is why there are serious theoretical problems upon convergence in the solution. Because of difficulties encountered in the model, it there is not an exact solution. In this paper we have developed a weak solution based on the results found in the literature [2]. This contribution is complemented by the construction of a numerical scheme for the quantitative study of the problem. This includes the formulation of decoupling techniques. The solution of the variational problem in space-time we realized the discreta inestability in time during the process. To overcome this difficulty we have used the Galerkin method with a numerical technique to capture the discontinuities in the Stream Lines Difussion (SD) with finite elements of type P1 + P2. We have built a new numerical model evolutionary for hydrodynamic volume fraction of solids and speed-gas in the simulating of the reactor with COMSOL Multiphysics [3] for the effective simulation of the evolutionary problem.

1.1 Physical Model

Assuming the model of the problem of phase solid-gas two-phase flow is then isothermal from a regularization free boundary problem which arises after a calculation model asymptotic phases, due to the steep gradient of solids concentration representing fluidization process in all its magnitude, this means building a whole mathematical methodology and regularization of singularities during the development of the problem of mixing. From a suitable proposal for the state equations and constitutive properties depend viscous stresses. In this way you can get the convergence of the approximation scheme. This phenomenon of hydrodynamic considered dense phase (solid particles) coexists with a continuous phase (gas), may originate suspensions stationary (stationary cloud) of particles in the gas. The purpose of this section is to model the transport of the solid particles from one zone to another to generate a cloud of particles, which fact means that the gas-solid system behaves like a viscous fluid macroscopically high temperatures and isothermal mode. Studies on gas-solid system in bed chamber, the formation and evolution of the bubble size is very important [2] since it is an indication that there is instability in the fluid bed chamber. If we consider a reference volume D, then from the laws of conservation of total mass and momentum for the solid and gas phases, we obtain a system of equations Navier-Stokes compressible flow Non- linear.

Gas phase:

$$\partial_t n + div(nu) = 0 \qquad (1.1)$$

$$\partial_t(nu) + div(nu \otimes u + p_g I) = div(2\nu_g n D(u)) + ng - qm(uv) \qquad (1.2)$$

Solid Phase:

$\partial_t m+div(mu) = 0$ (1.3)

$\partial_t(mv)+div(mv\otimes v+p_pI)=div(2 v_p mD(v)) +mg+qm(u-v)$ (1.4)

where: $n = \rho_g\phi$: specific mass of the gas phase $m = \rho_p(1-\phi)$: particulate phase specific mass, $\phi\in[\phi^*,1]$: porosity (gas volume fraction), $\phi^*\in(0,1)$, $\rho = 1-\phi$: Particle concentration, i.e volume fraction occupied by particulate matter, u and phase velocities of gas and particles, p_h y p_c: Hydrodynamic pressure for collisional pressure gas phase to particle phase. $q=q(\rho)$:Friction between phases v_g y v_p: kinematic viscosities of gas phase and particle phase, considered constant in this model ρ_g y ρ_p : densities of the gas phase and particle phase Assuming Newtonian behavior of the biphasic mixture (valid for low volumetric concentrations of solid particles), is used the stress operator:

$D(w) = ½[grad(w)+(grad(w))^T]$ (1.5)

Assuming the existence of an indicator that measures the ratio of proportionality between the densities of the two phases, in particular the parameter ε such that $0<<\varepsilon<1$.

$\varepsilon = \rho_g/\rho_p$, $\rho =\rho_p\alpha$; $n = n(\varepsilon)$, $m = m(\varepsilon)$, $u = u(\varepsilon)$, $v = v(\varepsilon)$. *When* $\varepsilon\to 0$, result the following mathematical model which is compressible apparently.

$\partial_t\rho+div(\rho v) = 0$ (1.6)

$\partial_t(\rho v)+div(\rho v\otimes v) + \nabla P = div(\rho D(v))+ \rho g$ (1.7)

$\nabla p_h = -\rho q(\rho)(u-v)$ (1.8)

$div((1-\rho)u+\rho v) = 0$ (1.9)

siendo $P = p_c+p_h$.

1.2 Equation of state

$p_c(\rho)=\rho^{\gamma o}exp[k\rho/(\rho^*-\rho)]$ $\gamma_o\geq 1$, $0 \leq \rho\leq\rho^*<1$ (1.10)

Equation for the drag force between phases:

$q(\rho) = C_q/(1-\rho)^s$, $s >0$, $s\in[1.4, 3.6]$ (1.11)

2. Statement of Problem

Let Ω_t an open subset of $[R^3_+ \times [0,\infty>]$,

$\Gamma_0 = \{\underline{x} = (x_1, x_2, x_3)\in R^3/ x_3 = 0\}$, (2.1)

$\Gamma_L = \{\underline{x} = (x_1, x_2, x_3)\in R^3/ x_3 = L, 0<L<\infty\}$, (2.2)

The problem is to find the volume fraction of particles $\rho\in C^1(\Omega_t)\cap C^0(\overline{\Omega}_t)$, velocity of the solid particles $v\in C^{2,1}(\Omega_t)\cap[C^{1,0}(\overline{\Omega}_t)]^d$, gas velocity represented by $u\in[C^{1,0}(\Omega_t)]^d\cap[C^{1,0}(\overline{\Omega}_t)]^d$, to solve the problem is considered hydrodynamic pressure $p_h\in C^{1,0}(\Omega_t)\cap C^0(\overline{\Omega}_t)$, from the state equation (1.6)- (1.9), assuming that d =1, 2 is the dimension of the space of the dependent variables are vector functions that vary in space and time, which satisfy the system of equations (1.6)-(2.4).

2.1 Boundary conditions

$[(1-\rho)u+\rho v)].\bar{n} = M\in C^0(\Gamma_0 \times [0,\infty >)$ (2.3)

$[\rho v].\ \bar{n} = m_0\in C^0(\Gamma_0 \times [0,\infty >)$ (2.4)

$[\rho v\otimes v + PI - \rho D(v)].\ \bar{n} = 0\in C^0(\Gamma_0 \times [0,\infty >)$

2.2 Initial conditions

$\rho(\underline{x},0) = \rho^0(x) \in C^0(R^3_+ \times \{0\})$ (2.5)

$v(\underline{x},0) = v^0(x) \in[C^0(R^3_+ \times \{0\}]^d$ (2.6)

En un espacio unidimensional, introduciendo el vector de estado

$$U(x,t) = \begin{pmatrix} u_1(x,t) \\ u_2(x,t) \end{pmatrix}; \text{ where } u_1 = \rho,\ u_2 = \rho v$$

such that:

P1) $U_t = f(U)_x+G(U)_x = S(U)$ (2.7)

$$f(U) = \begin{pmatrix} u_2 \\ \dfrac{u_2{}^2}{u_1} + p(u_1) \end{pmatrix},\ G(U) = \begin{pmatrix} 0 \\ \mu(\dfrac{u_2}{u_1})_x \end{pmatrix},$$

$$S(U) = \begin{pmatrix} 0 \\ -u_1 g + \dfrac{u_1 q(u_1)}{1-u_1}(M_0(t)-\dfrac{u_2}{u_1}) \end{pmatrix}$$

P2) $u = \dfrac{1}{1-u_1}(M_0(t)-\dfrac{u_2}{u_1})$ (2.8)

Where:

μ: Dynamic viscosity (considered constant)

$M_0(t)$: represents the incoming gas flow (air) in Γ_0. This data is known.
Air is passed through the bottom combustion chamber through a distribution plate with multiple nozzles to be bubbled the bed holding in suspension. The residue is roughly 2 -3% is the total bed weight.
To establish a numerical model, the initial and boundary conditions in particular are given by:

$$u_1(x,0) = \begin{cases} 0.4, & x \le L/4 \\ 0 \;, & x > L/4 \end{cases}$$

$$u_2(x,0) = 0$$

$$M_t(0,t) = 1$$

$$u_2(L,t) = 0$$

4. Numerical aproximation

Consider an weak one dimensional formulation of the problem to obtain solutions with less regularity of the required.

$$\int_0^T \int_0^L \phi_t U + (F(U) + G(U))\phi_x dxdt =$$
$$\int_0^T \int_0^L S(U)\phi\, dxdt - \int_0^L \phi(x,0)U(x,0)dx$$

Taking $\phi \in C_0^1(<0,L> x <0,T>)$, and if choose $x_{i-1/2}$, $x_{i+1/2} \in$ <0,L> and t_j , $t_{j+1} \in$ <0,T>, to $T<\infty$.

$$\phi(x,t) = \begin{cases} 1 & , \; if \quad (x,t) \in \left[x_{i-\frac{1}{2}}, x_{i+\frac{1}{2}} \right] \mathrm{x} \left[t_j, t_{j+1} \right] \\ 0 & , \; another\ case. \end{cases}$$

The discretization of the time variable is done in [5] using the explicit Euler method. Spatial integration is approximated by the midpoint method. Then the discrete problem, P1 is expressed:

$$\frac{1}{\Delta t} \int_{x_{i-\frac{1}{2}}}^{x_{i+\frac{1}{2}}} (U^{m+1} - U^m)dx + (F^m_{i+\frac{1}{2}} - F^m_{i-\frac{1}{2}}) + G^m_i = \int_{x_{i-\frac{1}{2}}}^{x_{i+\frac{1}{2}}} S(U^m)dx$$

$$m \ge 0,\ \Delta t > 0,\ t^m = m\Delta t,\ U^m = U(t^m)\ y\ F^m(t^m)$$
$$\Delta t \le H/max\{|v+c|, |v-c|\}$$

The spatial discretization of the computational domain [0, L] consists of M elements.
Between prototype model and geometric similarity exists when the relationship between all dimensions corresponding to the model and prototype are equal kinematic similarity exists if the path of the moving particles are geometrically similar counterparts and the relationship between the particle velocities are homologous equal, between two kinematically similar systems exist geometric and dynamic similarity if the relations between the forces counterparts in the model and prototype are the same. The forces may be a combination of the viscous forces, due to pressure forces, gravitational forces, forces due to surface tension and inertial forces. The problem in unidimensional space you can see that converges [5] with regularization of Harten Van Leer.

System Diffusive convective flow dimensional reagent [3]. Conservation system compressible dimensional Navier-Stokes can be expressed in its compact form and be almost linearized conservative as seen in the bidimensional case, introducing a vector function of states $\overline{\varphi}$, $\vec{v} = (v_1, v_2)$, thus the preservative system in form convective-diffusive flow-reactive to $\rho_p = 1$ and is expressed as:

$$\overline{\varphi} = \begin{pmatrix} \rho \\ \rho v_1 \\ \rho v_2 \end{pmatrix} = \begin{pmatrix} \varphi_1 \\ \varphi_2 \\ \varphi_3 \end{pmatrix}$$

$$\overline{F}(\overline{\varphi}) = \left(\begin{pmatrix} \rho v_1 \\ p_c + \rho v_1^2 \\ \rho v_1 v_2 \end{pmatrix} \begin{pmatrix} \rho v_2 \\ \rho v_1 v_2 \\ p_c + \rho v_2^2 \end{pmatrix} \right)^T ,$$

$$\overline{G}(\overline{\varphi}) = \frac{2\mu}{3\,\mathrm{Re}} \begin{pmatrix} 0 & 0 \\ 2\frac{\partial v_1}{\partial x} - \frac{\partial v_2}{\partial y} & 2\frac{\partial v_1}{\partial y} - \frac{\partial v_2}{\partial x} \\ 2\frac{\partial v_1}{\partial y} - \frac{\partial v_2}{\partial x} & 2\frac{\partial v_2}{\partial y} - \frac{\partial v_1}{\partial x} \end{pmatrix}$$

$$\mu = \rho v$$

$$\overline{S}(\overline{\varphi}) = \begin{pmatrix} 0 \\ \rho \left(\frac{q(\rho)}{1-\rho} \left(M(t) - \frac{\partial p_h}{\partial x} \right) - g_x \right) \\ \rho \left(\frac{q(\rho)}{1-\rho} \left(M(t) - \frac{\partial p_h}{\partial y} \right) - g_y \right) \end{pmatrix}$$

5. Use of COMSOL Multiphysics

In the two-dimensional case, after a process dimensionless [5] and then being obtained its weak formulation, the domain is located in a rectangular geometry region $\Omega_T = ((0,L)x(0,H))x[0,T)$, has:

$$\begin{aligned} &\int_0^T \int_0^L \int_0^H (\phi_t U + F(U) + G(U))\phi_x dydxdt = \\ &\int_0^T \left(\int_0^L \int_0^H S(U)\phi\, dydx \right) dt - \\ &-\int_0^T \left(\int_0^L \int_0^H \phi(x,0) U(x,0) dydx \right) dt \end{aligned} \qquad (2.9)$$

as shown in Figure 1 with dimensions L as the length of the base and in particular H = 2 is the height of the reactor. Initial conditions are considered in t = 0 the step function and rect. The boundary conditions for input the gas is inhomogeneous Newman type and elsewhere of the reactor homogeneous wall type in the rest of the border. The function for mixing flow, speed and pressure obtained with the sliding-type model Hadamard and the condition is considered homogeneous boundary Dirichlet output. Using COMSOL under study non stationary, i.e., time dependent. The features modeling geometry is building the Bézier Polygon. Adding an new meshing sequence and considering the boundary layers and free triangular in the boundary free problem and refinement. For that, we use the mesh toolbar. For the solution of system PDE with COMSOL Multiphysics Solve and using the weak form using the finite element method $P_1 + P_2$ and stabilization techniques as Streamline Diffusion (SD) ([3]-[4]). Then, the problem time dependent solver with IDA which use variable order variable step zise and rect, known Backward Differentiation Formulas (BDF) method.

Finally, quase linear systems solvers with PARDISO method. The convergence of the numerical scheme parameters depends preconditioned and the rearrangement of the matrix, PARDISO factorization method.

6. Results

The results shown are coal waste assumptions regarding the parameter in the order of 10^{-4}

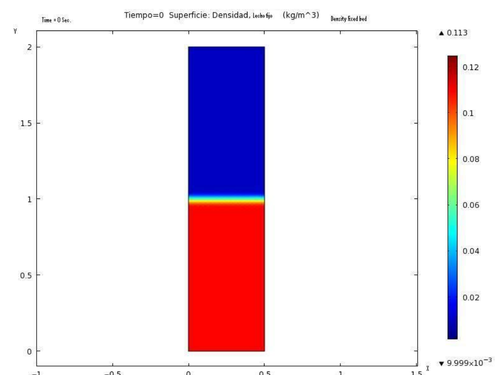

Figure 1. The axial section of the fixed bed reactor is represented in the XY plane.

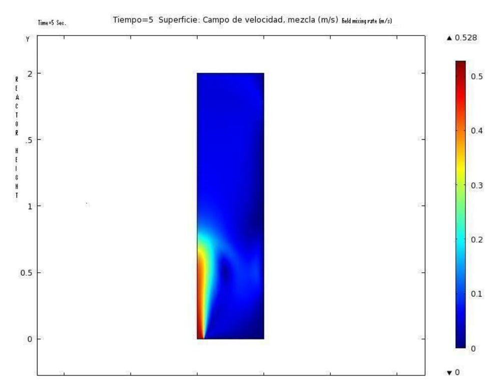

Figure 2. Gas velocity through a nozzle in the bed

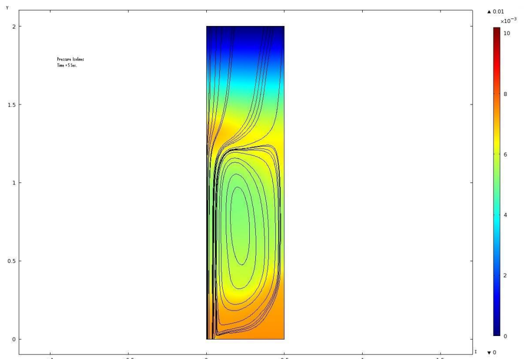

Figure 3. Pressure stream lines in the bed

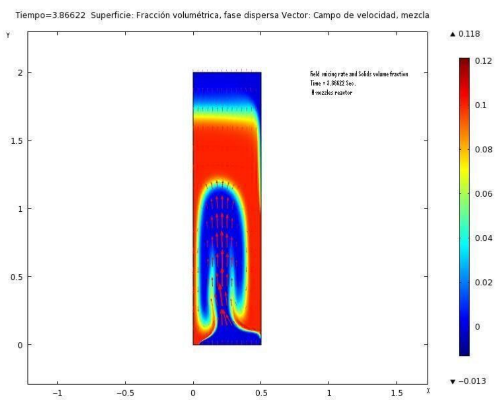

Figure 4. Speed and volume fraction in a reactor of N-nozzles.

7. Conclusions

Treatment was performed for decoupling asymptotic variables system, generating the formulation of a boundary value problem and value initial associated with a system of nonlinear regularized conservation, expressed in terms of a set of partial differential equations of type Navier Stokes compressible viscous flow and appearance for the variables conservative such as speed of flow, bulk density and overall pressure. [5].

This work contributes to the knowledge of numerical techniques that are very important in order to predict the optimal size for fluidizing solid particles in multiphase flow models. The detailed information will be given at the conference [4], [5].

It has built an algorithm of the solution process of a non-stationary mathematical model on a regular domain spaces evolutionary one and two dimensions of the problem of gas phase mixture a boiler solids fluid bed system.

In the two-dimensional case approximates the solution of the problem with the method Galerkin (SD) and a difference scheme (BDF) for the variable explicit Capture and temporal discontinuities of singularities in the streamlines of the convective flow, can be stabilized with a mesh refined time step evolution with close to $10x10^{-3}$ and side length element, maximum and minimum $1.5x10^{-4}$ and with a resolution of 0.25 curvature.

Can be interpreted four characteristics regarding speed, these are: A surface speed which occurs when in the vicinity of the column The section shown in the spectrum of the color palette, where no particles (no red) and only gas flows (presence of blue), a speed minimum fluidization of the results observed with the increase in the flow in bed, manifests a state of suspension caused by the upward flow and gas by multiple nozzles. This flow creates drag force (inertial force) which balances gravity and terminal velocity which is manifested in the rate of free fall of the disperse phase through the fluid and when it stays away column environment. The minimum speed is observed when bubbling the first bubble appears, this is important because it causes the mixing of particles and particle expansion chamber LF.

8. References

1. J.R. Grace, G. Sun, *Influence of particle size distribution on the performance of fluidized bed reactors*, *Journal*, Chem. Eng., **Volume**. 69 (5), pages 1126-1134 (1991).
2. Drew, D.A, *Mathematical modelling of two-phase Flow*. Annual Review *Fluid Mechanical*, **Volume** 15, pages 261-291 (1983).
3. Claes Johnson, *Numerical solution of partial differential equations by the finite element*, pages 182-187, Cambridge University Press, Sweden, (1994).
4. COMSOL MULTIPHYSICS, *User's guide*. Version 4.0, pages 290-298, *Module CFD*, pages 260-308 (2010).
5. Mantilla, Irla, *Mathematical Contribution to Simulate the Numeric Behavior of the Mixture*

*Flow Gas – Solid***,** Doctoral Thesis in National University of Engineering, 2012.

9. Acknowledgements

Acknowledgements to Research Institute of the Faculty of Science of the National University of Engineering, to Department of Applied Mathematics and Computer Science from the Polytechnic University of Madrid, COMSOL and the organizers for letting me participate in the event.