Development of a consistent and conservative Eulerian-Eulerian algorithm for multiphase flows

Ana Cubero,* Alberto Sánchez, and Norberto Fueyo †

Numerical Fluid Dynamics group (gfn). University of Zaragoza. Spain

March 15, 2011

A new solver for unsteady multiphase flows has been implemented in OpenFOAM: CIPSA Multi-Phase EulerFoam. The algorithm is based on an Eulerian-Eulerian and conservative (IPSA-like) formulation (see, for example, [2]) and incorporates an improved correction for the momentum interpolation approach for calculating face fluxes in a consistent way.

The single-phase Compact Momentum Interpolation (previously developed by the present authors [1]) is extended here to multiphase flows; the procedure takes into account the interphase drag terms appearing in the momentum conservation equations. The resulting formulation leads to consistent solutions in several respects (as it can be found by algebraic analysis): converged and steady solutions are (strictly) independent from transient terms; drag coefficients cancel out when fluxes are summed over each one of the phases; the application of the solver to several identical phases results in the same velocity and pressure fields.

From the momentum conservation equations, the pseudo-velocity $U_\alpha$ of a phase $\alpha$ can be written as:

\[
(A^0_\alpha + A^t_\alpha + A^d_{\alpha\beta})U_\alpha = H^0_\alpha + A^t_\alpha U^{n-1}_\alpha + A^d_{\alpha\beta}U^{k-1}_\beta
\]  

\[
(1 + C^t_\alpha + C^D_{\alpha\beta})U_\alpha = H^0_\alpha/A^0_\alpha + C^t_\alpha\alpha U^{n-1}_\alpha + C^d_{\alpha\beta}U^{k-1}_\beta
\]

where $A^0_\alpha$ is the main diagonal coefficient without the temporal term ($A^t_\alpha = \alpha\rho_\alpha/\Delta t$) and the coefficient for the interphase drag ($A^d_{\alpha\beta} = \alpha\beta K_{\alpha\beta}$; $K_{\alpha\beta}$ is the drag function, proportional to the magnitude of the relative velocity between phases $\alpha$ and $\beta$; $C^t_\alpha = A^t_\alpha/A^0_\alpha$ and $C^d_{\alpha\beta} = A^d_{\alpha\beta}/A^0_\alpha$; $k - 1$ and $t - 1$ represent previous iteration and time step, respectively. This expression is interpolated to the faces and the corresponding face-flux $\phi_\alpha$ is calculated as:

\[
\phi_\alpha = \frac{1}{1 + C^t_\alpha + C^d_{\alpha\beta}} \{U^0_\alpha + C^t_\alpha\alpha \phi^{n-1}_\alpha + C^d_{\alpha\beta}\phi^{k-1}_\beta\}
\]

†Corresponding Author: Norberto Fueyo (norberto.fueyo@unizar.es)
where $\phi_{\alpha_f}^0 = U_{\alpha_f}^0 = (H_{\alpha_f}^0/A_{\alpha_f}^0)$ corresponds to the face-flux calculation in the original Rhie and Chow formulation [4], i.e., for single-phase and steady flows.

Figure 1 shows a comparison of the performance of the extended Compact Momentum Interpolation with the correction proposed by Choi for unsteady flows [3] (which is the one provided by the $ddtPhiCorr$ function in OpenFOAM library). This figure displays the mass flow rate of a dilute granular phase in air ($\alpha_s = 0.001$ and $\rho_s = 24 kg/m^3$) going up a riser and then through an elbow. The present algorithm modification removes some unrealistic discontinuities appearing in the flow just past the elbow, where the computational grid presents an abrupt change in both cell sizes and aspect ratios. Additionally, the $CIPSAMultiPhaseEulerFoam$ solver has been found to be accurate and robust in a variety of cases ranging from verification-like problems (as free fall in dilute still air or laminar phase separation in horizontal channels by gravity) to more realistic flows as the prediction of the typical core-annulus pattern in dense circulating fluidized beds.

![Figure 1: Mass flow rate of the granular phase in a dilute riser (0.305m x 6.505m), with a change in the cell just past the exhaust elbow (center). Predictions of the proposed CMI interpolation (left) and Choi correction (right); detail of the bend area.](image)

Finally, the new features of the developed solver (with respect to the $\alpha$-intensive solver already available in OpenFOAM ($TwoPhaseEulerFoam$ solver)) include, apart from a conservative formulation, a data structure which allows for defining (in a general form) an arbitrary number of phases and an algorithm which accommodates both constant and variable density flows. Furthermore, the implementation of new libraries to model interphase mass and heat transfer, and both homogeneous and heterogeneous combustion is underway. The main goal of the project is to model oxycombustion in circulating fluidized beds.

References