11th. World Congress on Computational Mechanics (WCCM XI)
5th. European Conference on Computational Mechanics (ECCM V)
6th. European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

Simulation of interfacial flows using a Cartesian explicit finite volume solver

A. Bardin[a,b], G. Oger [a], and D. Le Touzé [a]

[a] LHEEA Lab. (UMR CNRS)
LUNAM Université, Ecole Centrale Nantes, Nantes, France
[b] IRT Jules Verne
Chemin du Chaffault 44340 Bouguenais

Key Words: *Interfacial flows, Level-Set, ACLS, Pseudo-compressible*

An explicit Cartesian finite volume method is currently under development in the LHEEA Lab. [1][2]. These developments are based on a weakly compressible cell-centered scheme where second-order accuracy is provided by using a MUSCL [3] scheme together with various limiters for the hyperbolic part. This paper deals with presenting the method retained for simulating interfacial flows with this model. Among the various Eulerian methods available for the simulation of interfacial flows, we compare the Level Set (LS) method [4] and the Accurate Conservative Level Set (ACLS) [5]. These techniques are both based on the convection of a distance function following the velocity field. The LS method has many advantages in terms of interface description, access to interface properties (normal and curvature). Its main drawback relies on the possible lack of mass conservation. In order to address this problem, the ACLS method uses a hyperbolic tangent function instead of the distance function, ensuring a reduction of errors. In this paper, these two methods are compared, through five criteria: diffusion of the interface, mass conservation, accuracy and CPU performances. Two numerical test cases are used to compare these methods: the Zalesak's disk and the 2-D convection-stretching of a single vortex. The Cartesian Explicit Finite Volume method based on a compressible (hyperbolic) solver is then described. This cell-centered finite volume method is totally Eulerian since the meshes are strictly fixed and a sharp interface should evolve through the grid. A description of the method dedicated to couple this solver with the LS/ACLS method is then provided. Finally, the results obtained on a dam breaking simulation are presented and discussed.

REFERENCES

[1] C.Leroy, P.Bigay, G.Oger, D. Le Touzé , B.Alessandrini, "*Preliminary developments of two-fluid Cartesian-grid Finite Volume Characteristic flux model for marine applications*", WCCM Proceedings, 2012.

[2] P.Bigay, C.Leroy, G.Oger, P.-M. Guilcher, D.Le Touzé, "*Development of a new cartesian explicit solver for hydrodynamics flows*", OMAE Proceedings, 2013.

[3] B. Van Leer, "*Towards the Ultimate Conservative DifferenceScheme. V. A Second-order Sequel to Godunov's Method*", Journal of Computational Physics, vol. 32, pp. 101-136, 1979.

[4] S. Osher, J.A. Sethian, "*Fronts propagating with curvature dependent speed : algorithms based on Hamilton-Jacobi Formulations*", Journal of Computational Physics, vol. 79, pp. 12-49, 1988.

[5] E. Olsonn, G. Kreiss, "*A conservative level set method for two phase flow*", Journal of Computational Physics, vol. 210, pp. 210-225, 2005.