Commun. Comput. Phys.
doi: 10.4208/cicp.210211.011111a

Vol. **12**, No. 4, pp. 1070-1095
October 2012

The Space-Time CE/SE Method for Solving Reduced Two-Fluid Flow Model

Shamsul Qamar[1,2,*], Munshoor Ahmed[1] and Ishtiaq Ali[1]

[1] *Department of Mathematics, COMSATS Institute of Information Technology, Park Road, Chak Shahzad Islamabad, Pakistan.*
[2] *Max Planck Institute for Dynamics of Complex Technical Systems, Sandtorstr. 1, 39106 Magdeburg, Germany.*

Received 21 February 2011; Accepted (in revised version) 1 November 2011

Communicated by Boo Cheong Khoo

Available online 28 March 2012

Abstract. The space-time conservation element and solution element (CE/SE) method is proposed for solving a conservative interface-capturing reduced model of compressible two-fluid flows. The flow equations are the bulk equations, combined with mass and energy equations for one of the two fluids. The latter equation contains a source term for accounting the energy exchange. The one and two-dimensional flow models are numerically investigated in this manuscript. The CE/SE method is capable to accurately capture the sharp propagating wavefronts of the fluids without excessive numerical diffusion or spurious oscillations. In contrast to the existing upwind finite volume schemes, the Riemann solver and reconstruction procedure are not the building block of the suggested method. The method differs from the previous techniques because of global and local flux conservation in a space-time domain without resorting to interpolation or extrapolation. In order to reveal the efficiency and performance of the approach, several numerical test cases are presented. For validation, the results of the current method are compared with other finite volume schemes.

AMS subject classifications: 76T99, 65Y99, 65M99, 35L45, 35L65, 35L67

Key words: Reduced model, space-time CE/SE method, central schemes, conservation laws, hyperbolic systems, shock solutions.

---

1 Introduction

The physics of single-phase flow is relatively simple compared to two-phase flow due to existing moving and deformable interface and its interactions with the phases. Two-

*Corresponding author. *Email addresses:* `shamsul.qamar@comsats.edu.pk` (S. Qamar), `manshoor@comsats.edu.pk` (M. Ahmed), `ishtiaqali@comsats.edu.pk` (I. Ali)

phase flows are encountered in various scientific and engineering fields related to environmental research, chemical engineering processes, nuclear energy and advanced heat transfer systems. The modeling and simulation of such flows is one of the most challenging problems in computational fluid dynamics. The coupling of interface is challenge in these flows, as the coupling mismatch may generate large errors in the simulation. Significant efforts have been made in recent years to develop accurate general two-phase formulations, mechanistic models for interfacial transfer and interfacial structures and computational methods to solve these models.

This work is concerned with the computation of two fluid flow model of Kreeft and Koren [17]. In one space dimension, the selected model is a new formulation of the original Kapila's five-equation model [15]. The first four equations of the current model are similar to the Kapila's five equation model. These four equations describe the conservative quantities: two for mass, one for bulk momentum and one for bulk energy. The fifth equation corresponds to the energy equation for one of the two fluids and substitutes the topological equation in the Kapila's five equation model. In the fifth equation, the source term on the right hand side is responsible for energy exchange between the two fluids due to mechanical and thermodynamic works. The advantages of current model are the representation of all equations in integral form and as a single system. The current and original models are similar in differential form [17].

Several other models exist in the literature for describing the physics of two phase flows. These models use separate pressures, velocities and densities for each fluid. A convection equation for the interface motion is normally coupled with the conservation laws of flow models. In the literature such models are known as seven-equation models and are considered to be the most complete models. One of such models for solid-gas two-phase flows was initially introduced by Baer and Nunziato [3] and was further investigated by Abgrall and Saurel [1,29–31], among others.

Apart from being complete, the seven-equation model possesses physical and numerical complexities. As the general physics of the model is not needed in several cases, simpler and more compact models were introduced in the literature ranging from three to six equation models [2,4,9,10,31]. The current model is more interesting due to the presence of differential source term in the energy equation for one of the two fluids. The solution of this model needs no explicit algorithm for the interface motion and, hence, can be easily implemented in the existing flow solvers. Other interface-tracking models, such as the volume-of-fluid [13] and level-set [24,32] methods, require explicit equations for the interface motion. In the literature, different numerical methods have been proposed for solving two-phase flow models [1,11,16,17,27,29,33,36].

In this article, a CE/SE method of Chang [5] is implemented for solving the selected reduced model in one and two space dimensions. In the two-dimensional case, a variant CE/SE method of Zhang et al. [35] is applied on rectangular mesh elements. The method is not an incremental improvement of a previously existing computational fluid dynamic (CFD) methods and is substantially different from other well-established methods. This method has many non-traditional features, such as unified treatment of space and time,