

ABSTRACT

With a view to compute three-dimensional multi-component fluid flows in complex geometry encountered in many industrial applications, a novel finite volume method over hybrid unstructured meshes is proposed for the solution of the Navier-Stokes and advection equations. Among the several numerical approaches put forward by various research groups, finite volume method over unstructured meshes offers a promising technique for simulating fluid flow problems in a complex domain. Multi-component fluid flow problems, in particular those involving two fluids distinguished by sharp interface are of practical importance in many engineering applications including mould filling, liquid sloshing, pipeline drainage and fuel injection in an internal combustion engine etc.. Although a considerable amount of work has already been carried out to simulate the multi-component flows, efforts are still going on to handle the complexities associated with different phases. The challenges are mainly due to the jump in fluid properties, the presence of interfacial and large body forces. With the passage of time, a class of algorithms based on force balancing have been evolved. The primary concern of these algorithms is to discretely treat the forcing terms in an identical way. In the recent past, methodologies has been proposed for achieving force balancing over uniform/non-uniform structured meshes, but work on hybrid unstructured meshes is limited.

Considering the aforementioned fact, the present doctoral thesis focuses on the development of a generalized robust computer code for simulating binary fluid flow problems having large property variations across the interface over three-dimensional hybrid unstructured meshes that can handle different types of source term in a simple and efficient way. The code development started from the scratch. Starting from pure diffusion, convection-diffusion, then Navier-Stokes solver is validated against several benchmark problems. The final developed solver for binary fluid flow problems is the outcome after several rigorous testing at different stages during the development of the solver. A hybrid mesh is used with a collocated arrangement of the dependent variables. The diffusion fluxes are computed in a novel and natural way and the pressure-velocity decoupling is avoided by using momentum interpolation. In order to maintain the phase interface as sharp as possible,

separate testing has been performed for volume fraction equation. In the transport equation for the volume fractions, the convective term plays a major role. Knowing the fact that upwind scheme suffers from numerical diffusion and the higher order schemes (central differencing) results in numerical dispersion, in the present solver, high resolution scheme including CUBISTA and CUIBS has been implemented and tested over unstructured meshes. In order to validate the methodology and the computer code, several benchmark problems have been solved. The predicted results are compared with those available in literature. The comparisons are satisfactory.

Working within the algebraic volume of fluid framework, the initial efforts were on the development of a well balanced formulation within a collocated arrangement of variables over hybrid unstructured meshes. Knowing the fact that there is a sharp jump in material properties across the interface, a least squares based gradient calculation technique has been adopted. The benefit of least squares method over the traditional Green Gauss method lies in the fact that the cell centroid gradient is reconstructed from the respective values of gradients at each of its faces. Employing the least squares method and considering all the competing forcing terms in the pressure Poisson equation, actually resulted in a well balanced algorithm, which is then tested and validated against various problems. Apart from these discrete force balancing, we also looked on to the effect of using similar and dissimilar convective schemes among different equations and termed them as consistent and inconsistent treatment, respectively. It has been noticed that, for the problems including the well known static droplet, static tank, filling of a column, convection of an inviscid droplet and the benchmark case of rising of a bubble, the consistent treatment together with the well balanced algorithm gives correct results for almost every cases irrespective of the density ratio chosen.

Subsequently, an alternative approach for binary fluid flows has also been developed, which is based on a variant of the traditional Green Gauss approach. This methodology, which is referred to as modified Green Gauss method is an interpolation free approach and the implementation of this method is relatively easy as compared to the least squares method. The resultant gradient obtained with this approach is as accurate as those of the outcome of least squares based balanced force strategy as described earlier. To demonstrate the ability of the balanced force algorithm described in this particular work, wherein both least squares and modified Green Gauss have been employed, several test cases both in two- and three-dimensional have been carried out over orthogonal and non-orthogonal unstructured meshes. Typically, test cases involving marangoni stresses, surface tension forces, gravity in hydrostatic and piezometric formulation have been solved and results show the proper achievement of force balancing even on unstructured meshes. Later, the solver has also been validated for three dimensional problems whose experimental and numerical data are available in the literature. These include the impingement of a droplet over thin liquid film and dam break over an obstacle. The results show

a reasonably good agreement with the published data.

In the next part of the thesis, the algorithm has been applied to solve two pertinent problems related to two-components fluids.

Firstly, the dynamics of droplet dripping and detachment process from a horizontal solid substrate have been simulated numerically. This problem finds application in the fuel cell technology. The investigations have been carried out to explore the fascinating dynamics of droplets by studying the effect of various parameters including density ratio, viscosity ratio, surface wettability, Weber number (We) and the presence of surrounding droplet. It has been observed that, for the cases having low density ratio together with lower value of We number, force due to gravitational pull is not sufficient enough to overcome the effect of wall adherence, leading to the sticking of droplet on the surface itself. However, later in the study it has been shown that the same droplet which remained stick to the surface, got detached from the wall, when the We number is increased or if the wall wettability is changed or with the merger occurring due to the presence adjacent droplet.

In the second problem, the dynamics of the droplet impingement over thin liquid film with varying thickness has been considered. This problem is a subject of interest to spray coating industries. The thickness of the film has been varied by considering a sinusoidally varying bottom wall with an amplitude of 0.5. It has been observed that, in comparison to the base case (straight horizontal surface), the velocity with which the crown is spreading actually get decreased when the bottom wall is considered to vary sinusoidally. Similar behavior has been observed irrespective of the location of drop fall i.e. either falling over crest or over the trough. Also it has been noted that, in the case when droplet is falling over crest, the thickness of the lower portion of the crown rim also gets increased. Apart from these, the effect of droplet impingement at an angle has also been explored. Two different angles have been considered and the results are then compared with the case when droplet hits the liquid film vertically.

Finally, the thesis concludes with a summary of the main findings and recommendations for future work.