Joint Meeting of the JMBC Contactgroups

Multiphase Flow and Computational Fluid Dynamics

Thursday 17th November 2016

Theme: Computational Multiphase Flows

Centrum Wiskunde & Informatica (CWI),
Science Park 123, 1098 XG Amsterdam

Programme

10:00 Coffee

10:30 Welcome by Daan Crommelin on behalf of CWI

10:45 Benjamin Sanderse, CWI and Shell, *Challenges in computational multiphase flow modelling in the oil- and gas industry*

11:15 Guido Oud, Delft University of Technology, *Solving multiphase flow in pipes using cylindrical coordinates*

11:45 Henk Seubers, Groningen University, *Multibody interaction in free-surface flows*

12:15 Barry Koren, Eindhoven University of Technology, *Application of a fully compressible multiphase SPH scheme to hypervelocity impacts*

12:45 Lunch

13:45 Raoul Liew, Frames Group, *Examples and limitations of multiphase CFD in industrial applications*

14:15 Paolo Cifani, Twente University, *High-order interface methods for simulating bubble dynamics in turbulent flow*

14:45 Saurish Das, Eindhoven University of Technology, *A novel multiscale modelling approach for flow through a cylindrical packed-bed reactor filled with porous non-spherical particles*

15:15 Irana Denissen, Twente University, *Front formation in bidispersed shallow granular flows*

15:45 Closure with drinks

Challenges in computational multiphase flow modelling in the oil- and gas industry

Benjamin Sanderse
CWI and Shell
b.sanderse@cwi.nl

In the petroleum industry, multiphase flow occurs when transporting oil and gas through long multiphase pipeline systems. The behaviour of the flow can take many forms, depending on parameters like fluid velocities, pipe properties and fluid properties. An important flow regime is slug flow, in which liquid pockets, separated by gas bubbles, propagate in an alternating fashion with high speed along the pipeline. Such slugs have a large influence on the sizing of receiving facilities such as slug catchers or separators. A promising approach to simulate slugs is using 'slug capturing', which requires accurate numerical solution of the one-dimensional two-fluid model.

In this talk, we look at two interesting properties of the two-fluid model. Firstly, the transition from stratified flow to slug flow, which leads to ill-posedness of the model. The ill-posedness is a result of model averaging in which too much physics is lost. We will look at the effect of ill-posedness on grid convergence and how it is affected by the choice of discretization method. Secondly, we consider the transition of compressible to incompressible flow. In incompressible flow, the presence of hidden constraints requires different time integration methods than in compressible flow. We propose a new approach to the problem using the theory of differential-algebraic equations (DAEs), from which we can derive new integration methods and which places existing solution methods into an overarching framework.

Solving multiphase flow in pipes using cylindrical coordinates

Guido Oud
Delft University of Technology
G.T.Oud@tudelft.nl

In this talk I will present some of the findings of my PhD research on the simulation of immiscible, incompressible and isothermal two-phase flows in straight pipe sections. The incompressible cylindrical Navier-Stokes equations are solved using finite differences, and the flow field evolves in time using an implicit mid-point method. The sharp interface is represented by an iso-contour of a level set function that is supplemented with a Volume of Fluid field to conserve mass globally. This approach is one of several hybrid interface representations commonly known as 'combined level set/VoF' (CLSVOF) methods. The jump conditions at the interface are modeled using the Ghost Fluid method. I will discuss some of the difficulties induced by the use of cylindrical coordinates and the solutions I found. Furthermore, I will show some results based on physical test cases (bubbles, jets and instabilities), and I will conclude with a proposed improvement of the standard Volume of Fluid method along the lines of the state-of-the-art Moment of Fluid method.

Multibody interaction in free-surface flows

Henk Seubers
University of Groningen
h.seubers@rug.nl

What influence do nearby ships have on each other? To simulate such fluid-solid interactions with CFD, the fluid dynamics and rigid-body dynamics must communicate to each other via the boundary conditions at the fluid-solid interfaces. The standard Dirichlet and Neumann conditions however lead to inefficient communication and can even create instabilities in the iterative coupling process. Instead of stabilizing this iterative process afterwards, a mixed-type boundary condition is proposed in order to remove this instability and improve the efficiency of the coupling. The mixed boundary condition is a combination of pressure and displacement prescribed to the fluid that approximates the dynamical behaviour of the ship. Through this modified boundary condition, the flow can anticipate the structural motions. In this presentation, the improved efficiency will be shown for multibody simulations.

Application of a fully compressible multiphase SPH scheme to hypervelocity impacts

Barry Koren
Eindhoven University
b.koren@tue.nl
Iason Zisis (TU/e), Bas van der Linden (LIME BV), Barry Koren (TU/e)

Hypervelocity impacts of space debris onto orbiting spacecraft are typically simulated through the Smoothed Particle Hydrodynamics (SPH) method. Hypervelocity impacts are shock-propagation problems, which in case of inhomogeneous materials require compressible multiphase schemes. From a variational principle, we have derived novel SPH schemes. We validate the accuracy of the schemes against exact solutions of one-dimensional shock-propagation problems, as well as against experimental results for shock-bubble interactions and hypervelocity impacts. The computational results are promising.

Examples and limitations of multiphase CFD in industrial applications

Raoul Liew
Frames Group
r.liew@frames-group.com

Frames Separation Technologies is a Dutch company specialized in oil/gas/water/sand separation solutions for the upstream Oil and Gas industry. Most types of these separators used in this industry rely either on gravitational or centrifugal driven separation. To improve the quality of the separation and minimize the size of gravity driven separators, so-called separator internals (e.g. calming baffles, plate packs, vane packs, wire mesh demisters) are used. Frames' clients more and more often require CFD based validations of separator designs, including their internals.

Multiphase CFD seems to be a powerful tool to verify separator designs and to obtain qualitative information with respect to separator performance. However, it remains restricted and the quality of its results should not be overestimated.

The currently available models to simulate multiphase flow, and which are from a commercial perspective useful, cover only bits of the entire puzzle that needs to be solved. Hence, many assumptions are made in order to simulate the complex processes in 2 or 3 phase separators. Besides the validity of the models, there is only a limited amount of parameters available on which we need to base the simulations.

Besides CFD for client-related projects, we utilize CFD for research purposes. Within the R&D framework of Frames, it is a very useful tool for the development of new types of separators and internals. Together with in-house lab experiments, design models are developed, updated, and/or validated.

In the presentation we will show some typical examples of multiphase flow behaviour in separators, problems typically encountered when using CFD on 2 and 3 phase separators, and how we utilize CFD for R&D.

High-order interface methods for simulating bubble dynamics in turbulent flow

Paolo Cifani
Twente University
p.cifani@utwente.nl

As a preliminary study, a comparative analysis between a compressive scheme and an interface reconstruction technique is carried out for the volume of fluid approach (VOF) for a laminar flow. Positive properties of the compression method are its simple implementation, applicability on unstructured meshes, mass conservation and boundedness. However, this method results into artificial smearing out of sharp interfaces and a loss of accuracy. The interface reconstruction method, on the other hand, gives an accurate representation of the interface which is kept sharp by construction, at the expense of a higher computational time. The platform we work with is OpenFOAM, where a new library was implemented that performs the advection of a colour function by means of a Piecewise Linear Interface Calculation (PLIC) together with a splitting advection algorithm. The two methods are compared for several 2D and 3D advection test cases with prescribed velocity field and then applied to the simulation of a rising bubble, including surface tension and a mass density ratio between the two phases of 1000. The established computational model is then used to simulate a two-phase turbulent bubbly channel flow at $Re_t = 180$. The mean liquid and gas velocities are computed for different gas volume fractions and the values are compared with the corresponding single-phase channel flow.

A novel multiscale modelling approach for flow through a cylindrical packed-bed reactor filled with porous non-spherical particles

Saurish Das
Eindhoven University of Technology
S.Das@tue.nl

Traditionally trickle bed reactors are randomly packed with shaped particles coated with a thin porous layer (washcoat) in which the micro-pores contain the catalyst. The catalyst inside the outer thin porous layer is accessible only via diffusion through those pores and it drives the design towards smaller particles to have a larger surface area. At the same time, the use of very small particles demands a larger pressure drop and one of the solutions is to use comparatively large pellets with open structures (i.e. porous) where washcoat layers are deposited at the outer surface of the solid structures. The different shaped pellets can be used to fill the reactor bed randomly.

Unlike conventional reactors, fluid will flow through and around the porous particles. A multiscale modelling approach has been developed to represent the flow for the two different length scales, i.e. pores inside the porous particles and voidage between the closely packed particles. The flow inside the internal pores of the particles are not fully resolved. Instead, they are modelled by closure terms in the volume averaged Navier-Stokes equations. However, the flow outside the particles is fully resolved in a conventional manner. In this way, simulation results can be obtained with a feasible computational cost and with reasonable accuracy. To generate random packings of non-spherical particles, a glued-sphere discrete element model (DEM) available within LIGGGHTS® has been used. An immersed boundary method (IBM) is incorporated to describe the cylindrical wall of the column in a Cartesian computational grid. The particles of different shapes and size have been considered. The effect of the internal porosity, particle size and shape, and the particle to reactor diameter ratio on the reactor performance is demonstrated.

Front formation in bidispersed shallow granular flow

Irana Denisseni
Twente University
i.f.c.denissen@utwente.nl

Predicting the distance to which hazardous natural granular flows (e.g. snow slab avalanches, debris-flows and pyroclastic flows) might travel is vital for an accurate assessment of the risks posed by such events. This runoff distance is strongly affected by the segregation that occurs in the high solids fraction regions of these flows. Segregation causes the larger particles to rise to the surface, from where they are transported to the front. In many natural flows, these bouldery fronts experience a much greater frictional force, leading to the formation of a bulbous flow front.

Continuum methods are able to simulate the flow and segregation behaviour of such flows, but have to make averaging approximations reducing the degrees of freedom from a huge number of particles to a handful of averaged bulk parameters. We show that a depth-averaged model for the height, velocity and particle size segregation for these flows can be surprisingly accurate. Furthermore, it is shown that these solutions converge to a simple travelling wave solution, which makes the computation of the long-time profile very inexpensive.

This talk will focus on a simple case of size-bidispersed dry granular flows over inclined planes. We will investigate this problem via continuum modelling and particle simulations. Both numerical and analytical solutions will be presented and we conclude by discussing how both can be combined to reveal deeper insight.