Dynamic semi-structured meshes for fast numerical simulation of Multi-Phase Modelling in Oil and Gas

EPSRC CASE studentship - for UK nationals only. Sponsored by industrial partner BP PLC

Professor C. C. Pain, Dr P. Salinas, Dr A. I. Obeysekara and Dr A. Nicolle
Applied Modelling and Computational Group, Department of Earth Science and Engineering

This project will develop and implement a semi-structured mesh approach within the Multiphase-Fluidity project to significantly improve its speed. Multiphase-Fluidity incorporates state-of-the-art technology for multiphase Navier-Stokes flow simulation, including unstructured dynamic mesh optimisation, high order element methods. Applications of Multiphase-Fluidity are numerous and include simulation of jets, droplets, boiling reactors, flow in pipes, mixing tanks, etc.

Currently, Multiphase-Fluidity uses fully unstructured meshes. Flexible meshing technology is required in order to honour the complex geometries that naturally occur in engineering applications (e.g. Fig. 1). However, numerical simulations using unstructured meshes are inherently slower than those using structured meshes due to the indirect addressing required by the former. This is partly compensated in Multiphase-Fluidity by using dynamic adaptive mesh optimisation which reduces the number of elements whilst maintaining required resolution to correctly model the physics.

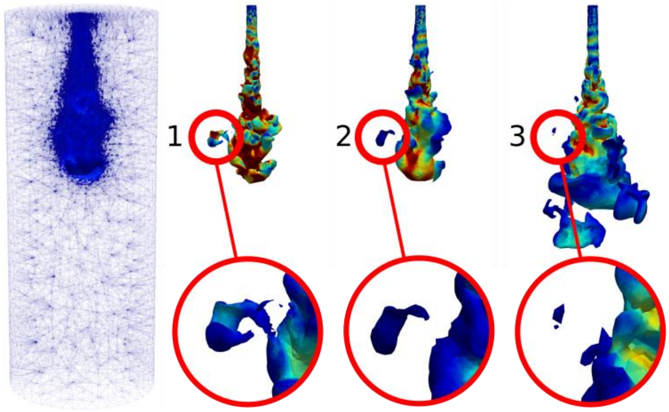

Fig 1. *Droplet formation and dispersion during modelling of multiphase turbulent jets using dynamic mesh optimisation*

In this project, a semi-structured mesh approach within the Multiphase-Fluidity software will be developed, this will require the development as well of a custom linear solver based on the PETSc framework. The semi-structured mesh approach combines the best aspects of structured meshes and adaptive methods: the initial, coarse unstructured mesh adapts to the domain and the nested structured mesh provides the accuracy required. The approach is similar to that used in spectral elements by some of the fastest turbulent flow solvers Combined with dynamic mesh optimisation, this method will provide faster simulation results than an unstructured mesh providing the same quality of the solution.

The performance of the developed method will be assessed studying jet flows. For jet flow simulations high-dynamic precision is almost mandatory to ensure that all the length-scales can be properly captured, this includes for example the generation of droplets (Fig. 1). The use of semi-structured grids combined with the present dynamic mesh optimisation already present within Multiphase-Fluidity will enable to run simulations much faster and with a much higher degree of precision than without this technology.

Applicants should have a strong mathematical background, a good degree in an appropriate subject (e.g. mathematics, physics, computer science or engineering) and a strong interest in computational modelling and code development. The project is hosted by the large and highly successful Applied Modelling and Simulation Group (AMCG) and will involve extensive interactions with other groups within the department and industry. This project is sponsored by BP and is part of its initiative to collaborate with academia to advance numerical techniques in its use of Multiphase-Fluidity. The candidate will get an opportunity to do a three-month placement at BP's UK office as part of their PhD studies. The modelling techniques developed in the project will have a broad impact on multiple industries helping to improve efficiency and safety. Skills developed during this project will include multiphase flows, high performance computing, parallelisation of numerical codes, numerical discretisation techniques, linear and non-linear solvers, dynamic mesh optimisation techniques, and structured and unstructured meshing technologies. The candidate will develop their career and profile by presenting at conferences and publishing in high impact journals. For more information please contact Prof. Christopher Pain (c.pain@imperial.ac.uk) or Dr Pablo Salinas (pablo.salinas@imperial.ac.uk).