An implicit boundary approach for the simulation of flow through industrial valves using a hybrid remeshed particle hydrodynamics method

Anas Obeidat∗‡, Thomas Andreas,‡, Stephane P. A. Bordas†, and Andreas Zilian∗

∗University of Luxembourg, Department of Computational Engineering Sciences, Faculty of Science, Engineering and Communication, University of Luxembourg, 6 Avenue de la Fonte, 4364 Esch-sur-Alzette, Luxembourg.

†Visiting Professor, Institute of Research and Development, Duy Tan University, K7/25 Quang Trung, Danang, Vietnam.

‡ROTAREX Solutions S.A., 24, rue de Diekirch, L - 7440 Lintgen, Luxembourg.

ABSTRACT

A particularly challenging issues in manufacturing of medical gas valves and regulators, are gas-dynamic pressure surges and adiabatic compression phenomena, along with the design and optimisation of the complex geometries of valves. Depending on the field of application, the pressure surges can lead to considerable malfunctions, and in very critical cases, to total failure of the respective component. The pressure surges in the case of the medical oxygen regulator can cause the oxygen to ignite, setting fire on the valves and control organ. We present a remeshed hybrid particle-mesh method for the simulation of three-dimensional compressible turbulent flow [1]. As a result of the strain of the flow, the mesh free smoothed particle hydrodynamics (SPH) method suffers from particles distortion which induces an inaccurate representation of the diffusion effect along with the pressure gradient (rate-of-change of momentum). To overcome this problem we introduce a grid which is used to remesh (reorganise uniformly) the particles to ensure a regular particle distribution and convergence of the method. To take advantage of both the Lagrangian and the Eulerian schemes, the right hand side of the turbulence model is computed on the mesh, and the change of momentum is later interpolated to the particles where the advection takes place. The method is capable of providing quality simulations while maintaining its robustness and versatility. We couple the hybrid remeshed particle-mesh method with Brinkman penalisation [2] to be able to simulate viscous flow with high Reynolds number where complex geometries are required. A penalty term is added to the Navier-Stokes equations to impose the boundary conditions [3]. The boundary conditions are enforced to a specific precision with no need to change the numerical method or the grid. The remeshed particle-mesh method with Brinkman penalisation provides a good quality simulation, and the results showed inline with the analytical solutions.

REFERENCES

