Massively parallel simulations using OpenFoam – application to air-assisted atomization

A. Dord*‡1 G. Laskowski*‡1

1 GE Global Research, Niskayuna NY 12309 USA

March 12, 2011

InterFoam, multiphase, scaling, performance, air-assisted atomization.

Air assisted atomization can be found in many industrial applications, for example gasification and gas turbines injection systems. While the modeling of primary breakup has received an increasing attention from the industrial and academic communities over the past few years, the prediction of the spray properties remains a challenge, in particular for high-density ratio flows.

Recent studies have looked at the Detailed Numerical Simulation of the breakup of a liquid jet in cross flow, or coaxial liquid jet [1-3]. To model the multiphase flow, Volume of Fluid (VOF) [3], Level set (LS) [1,4] or a combination of both techniques [2] is usually used. As opposed to statistical approaches, the grid requirements for simulating breakup are significant in this case. Hence, liquid breakup simulations have to be run on massively parallel computing platforms, bringing the performance challenge in addition to the modeling challenges.

The objective of the present study is to examine the interFoam flow solver capabilities for the simulation of air-assisted primary breakup. A detailed bottom up validation has been performed in order to look at the different solver components, including interface transport (Figure 1), LES modeling, and boundary conditions.

Figure 1: Deformation of a 3D sphere by a time varying velocity field. Evolution of the volume fraction iso-surface with time.

In a next step, the porting of interFoam (OpenFoam 1.7.0) on to the Jaguar cluster at Oak Ridge National Laboratory has been considered. Performance analysis of selected OpenFoam 1.5 and 1.6 solvers including interFoam on HECToR (Cray XT4) is reported in [6]. The author considers the extended dambreak tutorial on a mesh comprising 6.23 million cells, for strong

‡ Corresponding Author: A. Dord (dord@ge.com)
scaling. In this particular case, the peak performance was obtained for 128 cores (64k cells per core). The objective of the present study was to look at the solver performance and identify bottlenecks and the influence of various parameters in a massively parallel environment (Figure 2). The profiling of the interFoam flow solver was carried out using VampirTrace, for processor counts ranging from 128 to 10368.

After identifying the best practices on the high-performance computing platform on a canonical validation case [5], the simulation of the breakup of a liquid film [7] has been run on a 340 M cell count. The gridding and simulation set-up were performed at ORNL, and all the post-processing was done locally. Results obtained for the liquid film atomization are compared to experimental data. In particular liquid breakup length, structure formation and oscillation frequency will be considered.

Acknowledgments
This research used resources of the National Center for Computational Sciences at Oak Ridge National Laboratory, which is supported by the Office of Science of the Department of Energy under Contract DE-AC05-00OR22725.
The author would like to acknowledge T. Ilsche from ORNL for his contribution to the detailed performance analysis.

REFERENCES


