Validation of OpenFOAM's Volume of Fluid Model

Ved Merchant[1], Ravi Duggirala[2], Akshay Kumar[3]

Manipal Institute of Technology, Manipal, KA, India-576104, vedmerchant7@gmail.com
Mercedes-Benz R&D India Pvt. Ltd., Bangalore, KA, India-560066, ravi.duggirala@daimler.com
Mercedes-Benz R&D India Pvt. Ltd., Bangalore, KA, India-560066, akshay.kumar@daimler.com

In fluid dynamics, slosh refers to the movement of liquid inside a closed domain (which is, typically, also undergoing motion). Ideally, the liquid must have a free surface to constitute a slosh dynamics problem, where the volume of the liquid can interact with the container to alter the system dynamics. Fuel sloshing phenomena has been an area of interest for many researchers over the past years as it holds a lot of importance in aircrafts as well as automobile industries. Shift in center of mass or slosh noise are the effects that are normally associated with sloshing phenomena.

In this paper, the OpenFOAM's volume of fluid (VOF) solver (*interFoam*) is validated for two fuel slosh studies from reference [1, 2]. These references were chosen as they include details of the experimental setup and results along with clear specifications of the boundary conditions to be incorporated in the simulations.

Validation of this solver is important so that it can be further used for the fuel slosh simulations with full scale tank geometries. OpenFOAM has been proved to be faster in many other domains compared to the available commercial CFD codes. Validation and implementation of OpenFOAM's solvers in industrial applications can lead to a huge reduction in computational time and expense.

The boundary conditions required for these studies, which include the amplitude and frequency for the acceleration profile and the fill levels are taken from the respective references for each study. The *blockMesh* utility in OpenFOAM is used to generate the geometry and mesh for both the cases.

Fuel Slosh Validation – I

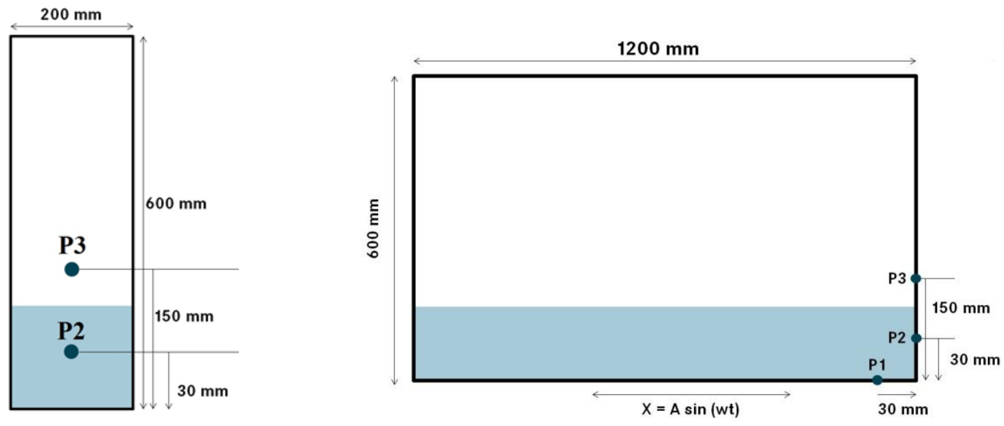


Figure 1: Domain and Pressure probe locations for case one

The mesh is for the above geometry consists of uniform cubes of side 5mm as further refinement of the mesh did not lead to considerable change in the results but doubled the computational time. Acceleration profile is generated using the time period 1.74 seconds and amplitude 60mm as specified in the reference. A pressure plot for probe three which is the most important probe to study slosh as it lies above the water level is shown in Figure 2.

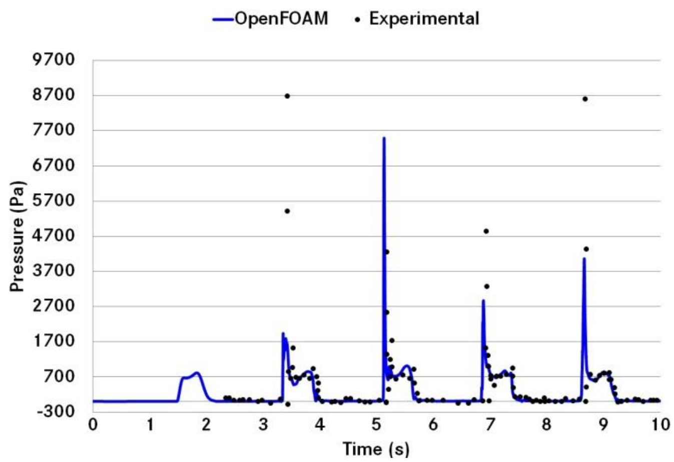


Figure 2: Comparison of OpenFOAM and Experimental results for P3

Courant number plays an important role in these simulations. Hence, a study is carried out to observe the effects of the maximum courant number on the pressure peak values. The figure below shows the results for three different courant number limits.

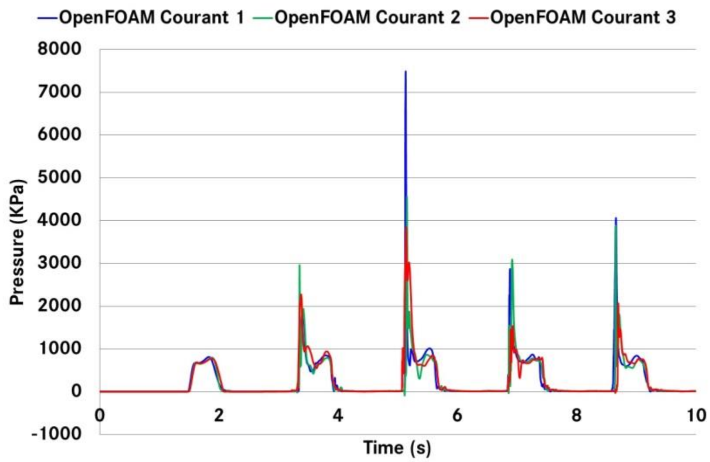
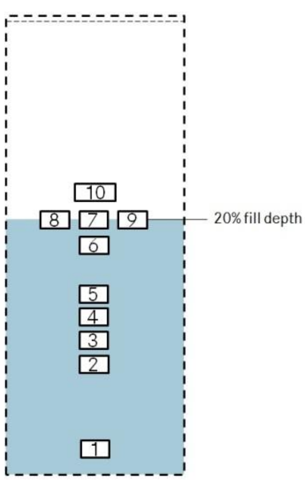


Figure 3: Comparison with different courant number limits

It is observed that increasing the courant number limit leads to a considerable decrease in simulation time due to higher allowable time steps. It also results in lower pressure peak values which depict reduction in accuracy for fuel slosh simulations. Hence for all the further studies, courant number is limited to unity.

Fuel Slosh Validation – II



Number	Position Z (mm)	Position Y (mm)
1	50	15
2	50	90
3	50	105
4	50	120
5	50	135
6	50	165
7	50	180
8	75	180
9	25	180
10	50	195

Figure 4: Pressure probe locations

For this study, a rectangular tank of dimensions 1.3m x 0.9m x 0.1m is taken with acceleration profile corresponding to the frequency of 0.496 Hz and amplitude of 0.2m. Pressure values at all the probes are tracked and the results are compared with the experimental results. The Figure below shows the pressure result for the probe-10.

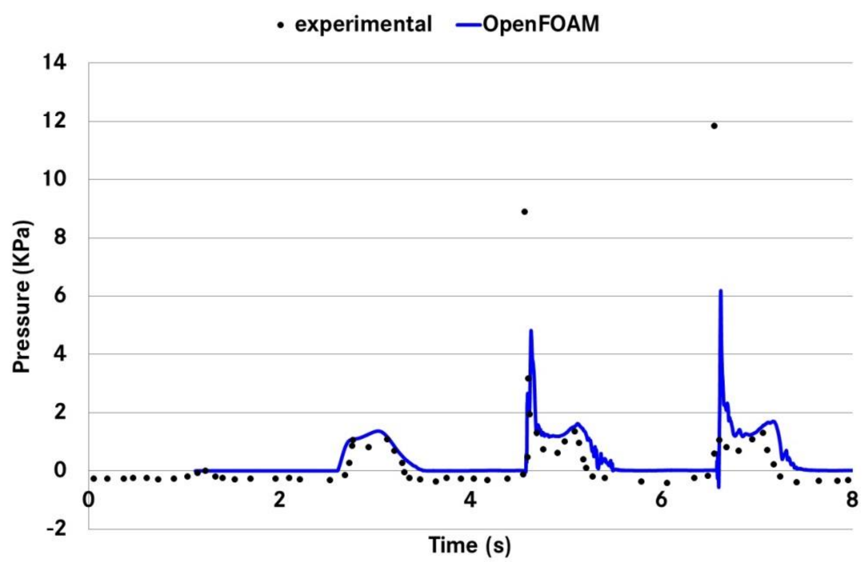


Figure 5: Comparison of OpenFOAM and Experimental results for P10

Conclusion

Results obtained from OpenFOAM's volume of fluid solver (*interFoam*) are found to be agreeing well with the experimental results. The under prediction of the pressure peaks by OpenFOAM may be due to various reasons such as the uncertainties in experimental data or limiting of the courant number to unity which leads to time steps that are higher than that required to capture the pressure peaks. Being able to capture the global features of the flow fairly accurately, it can be used for industrial applications which involve the use of volume of fluid model.

References

[1] R.S. Ganuga, H. Viswanathan, A. Awasthi and S. Sonar "Fluid structure interaction modelling of internal structures subjected to resonance", International Journal of Fluid Mechanics Research, Vol 14, No. 2, 2014.

[2] Ashkan Rafiee, Fabrizio Pistani and Krish Thiagarajan "Study of liquid sloshing: numerical and experimental approach", 8 August 2010, Comput Mech 2010.

[3] OpenFOAM user guide, Version 2.4.0