Oral presentation | Multi-phase flow

Multi-phase flow-IV

Tue. Jul 16, 2024 10:45 AM - 12:45 PM Room D

[4-D-03] VOF Modeling of Droplet Distortion under Supersonic Flow

*Hong Q Yang[1] （1. CFD Research）

Keywords: VOF, Supersonic Flow, CFD

VOF Modeling of Droplet Distortion under Supersonic Flow

H. Q. Yang
CFD Research Corporation, Huntsville, AL 35806, USA

An Abstract submitted to
ICCFD12
The 12th International Conference on Computational Fluid Dynamics
July 14-19, 2024
Kobe International Conference Center
Kobe, Japan

In the laboratory, shock tubes are most often used to generate large relative velocities between the liquid droplet and surrounding gas flow. Normal shock waves have little effect on the droplet. Instead, they serve as a reliable and repeatable way to create a high-speed flow around the droplet, which is responsible for the droplet's subsequent deformation and disintegration. The validations in the previous section of droplet deformation at different Weber numbers are more qualitative than quantitative due to the fact that experiments are mainly shadow graphs, derived drag, and lateral expansion estimates. Due to the mist of small drops around the original drop, the shadowgraph failed to reveal the detailed quantitative size of drop distortion. The work of Theofanous [1,2], however, utilized an innovative laser fluorescence technique and was able to capture qualitative details of the temporal droplet shape.

In this study, the experimental work of Theofanous utilizing an innovative laser fluorescence technique to capture qualitative details of the temporal droplet shape at several Weber numbers was used to validate our VOF tool. The key dimensional features of the droplet, such as the frontal radius of curvature, the overall depth, and the width of the droplet as a function of time, will be compared to simulation results as done by Moylan *et al. [3]*. The capability of reproducing the observed drop distortion features will be assessed.

Theofanous *et al.* [1] employed a pulse, supersonic wind tunnel, which was capable of generating well-defined, uniform, steady flows of duration up to 100 ms. The dynamic pressure range is up to 10^5 Pa, which (with millimeter-scale drops) yields Weber numbers of up to $3\text{x}10^4$. The facility can operate at pressure levels down to 10 Pa and can deliver a Mach 3 flow as long as the pressure ratio is greater than ~ 40.

3D Results at We=$5.4\text{x}10^3$ with 2M Cells

To investigate the 3D effect, a 3D model is built with 2 million grid cells. The gas-liquid surfaces from the simulation and those from the Theofanous experiment are shown in Figure 1 in the same instances. It is clear that very good agreements are observed all the time. The quantitative comparison is given in Table 1.

With the 3D model, very good agreement with the experiment has been obtained; The average error dropped from 15% ~ 30% in 2D to 5% to 12 %. The average run time is reasonable, around 72 hours/processor. The overnight run can be done with the use of 10-16 processors.

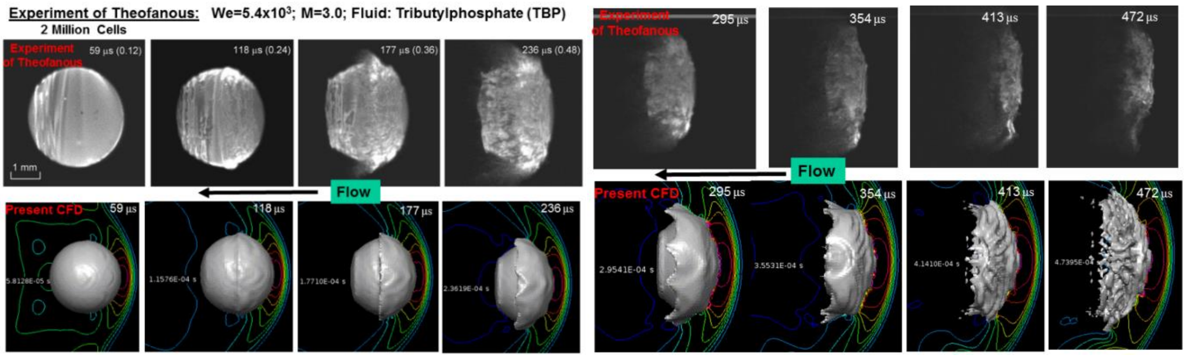
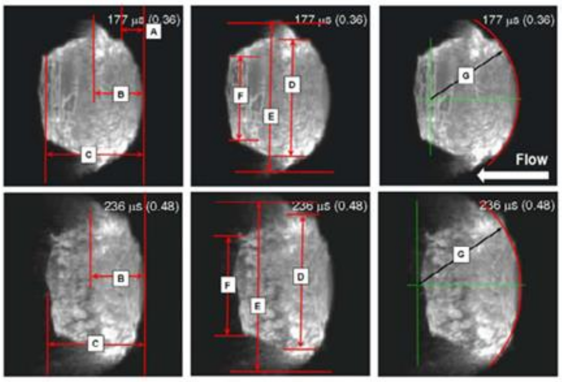


Figure 1. Comparison of Experimental LIF and 3D CFD gas-liquid interface. The flow is from the right to the left. We=5400, M=3. The model has 2 million cells.

Table 1. Comparison of Droplet Features between Experiment and 3D Simulation, 2 Million Cells.



Location	177 µs			236 µs		
	Data	VOF	Diff (%)	Data	VOF	Diff (%)
A	0.63	0.77	22.22	-		
B	1.28	1.51	18.36	1.21	1.27	4.95
C	2.64	3.08	16.59	2.43	2.77	14.19
D	3.14	3.54	12.73	3.00	2.88	4.00
E	3.9	4.04	3.59	4.23	4.22	0.23
F	2.21	2.26	2.26	2.47	2.46	0.40
G	2.29	2.13	6.98	2.72	2.61	3.33
		Avg.	11.82		Avg.	4.52

3D 2 Million Cells

<u>3D Results at We=5.4x10^3 with 10M Cells</u>

Our grid is further refined to 10 Million cells. The comparisons of CFD solutions for this high resolution to the experimental images are given in Figure 2. Now, the detailed, stripped, smaller drops are clearly visible from CFD.

The quantitative comparison is listed in Table 2. Now, one can see that the agreement is excellent. The error is from 4% to 6%, significantly lower than 2D and 3D with 2 million cells. The run time on ten processors took about one week. However, it can be shortened using more processors on High-Performance computers.

Figure 3 left shows the development of the pressure field in and around the drop when the shock wave passes. It is seen that due to the high speed of sound of the liquid, the pressure prorogates faster inside the drop when the shock wave arrives at the drop. Figure 5 right shows the velocity vector around the drop when the shock wave passes. Figure 6 illustrates the development of the R-T wave at the later time instances and the evidence from the experiment.

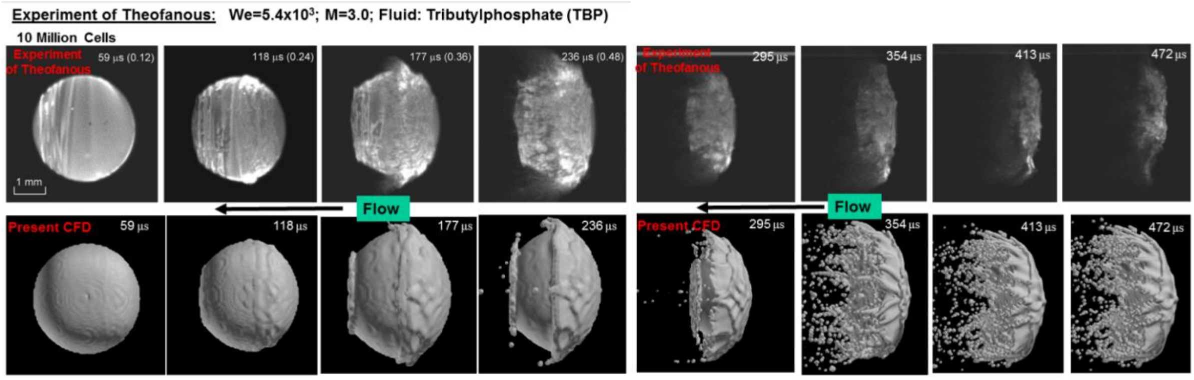
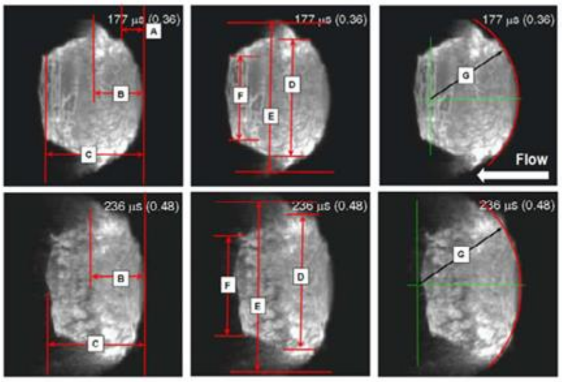
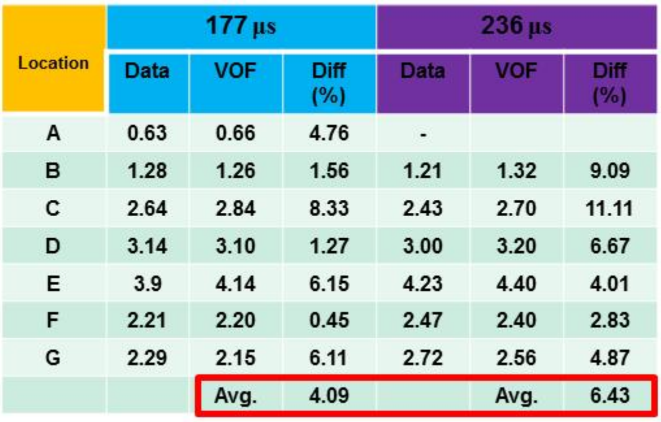


Figure 2. Comparison of Experimental LIF and 3D CFD gas-liquid interface. The flow is from the right to the left. We=5400, M=3. The model has 10 million cells.

Table 3. Comparison of Droplet Features between Experiment and 3D Simulation, 10 Million Cells.





Location	177 µs			236 µs		
	Data	VOF	Diff (%)	Data	VOF	Diff (%)
A	0.63	0.66	4.76	-		
B	1.28	1.26	1.56	1.21	1.32	9.09
C	2.64	2.84	8.33	2.43	2.70	11.11
D	3.14	3.10	1.27	3.00	3.20	6.67
E	3.9	4.14	6.15	4.23	4.40	4.01
F	2.21	2.20	0.45	2.47	2.40	2.83
G	2.29	2.15	6.11	2.72	2.56	4.87
		Avg.	4.09		Avg.	6.43

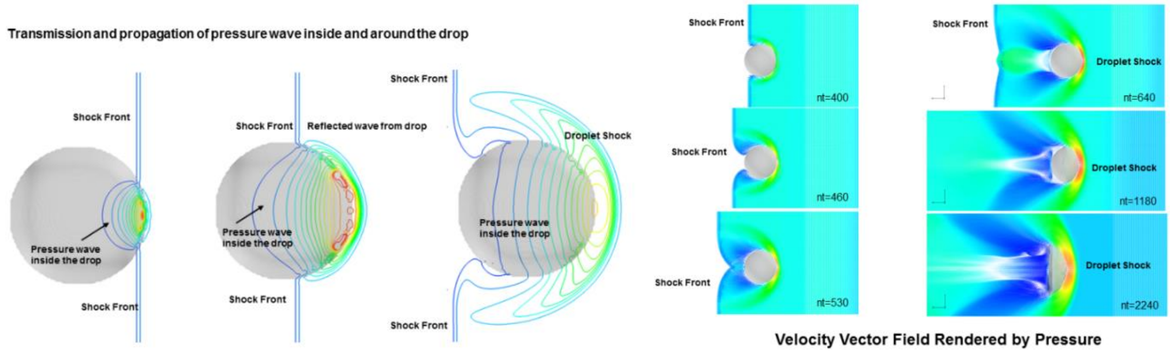


Figure 3. Development of pressure and velocity vector fields in and around the drop when the shock wave passes.

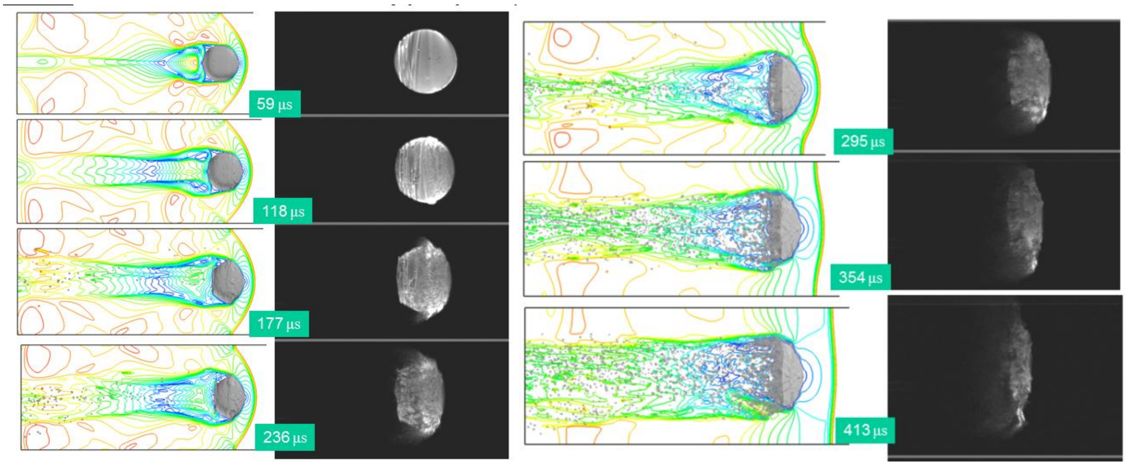


Figure 4. Main and stripped drops from CFD simulation and comparison to experiment.

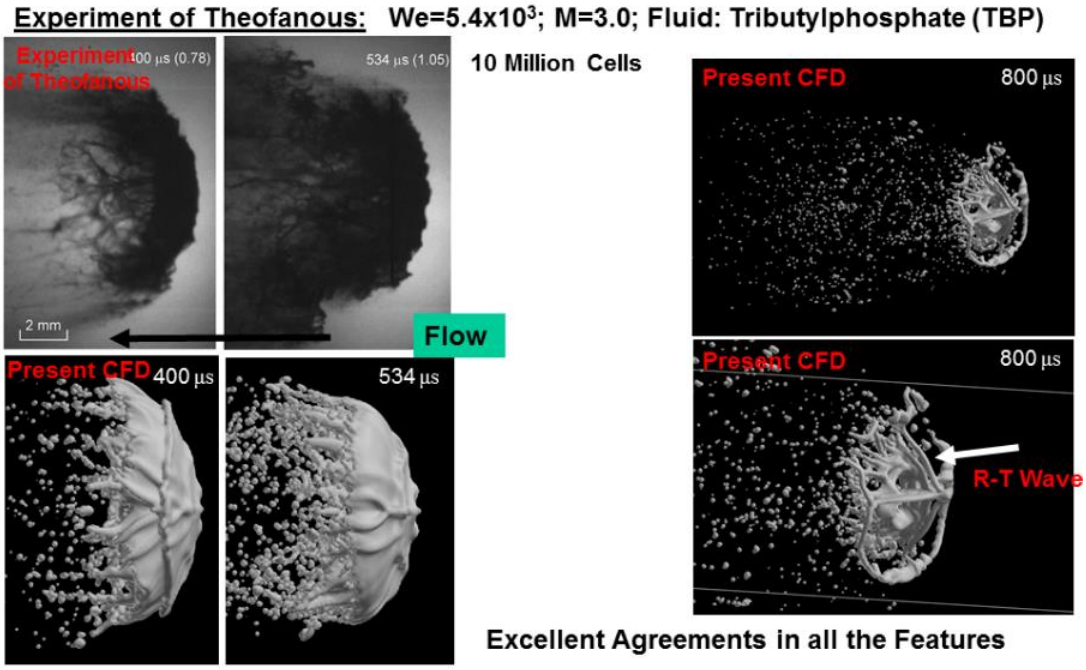


Figure 5. Development of R-T wave at the later time instances and evidence from the experiment.

References

1. Theofanous, T. G. 2011 Aerobreakup of Newtonian and viscoelastic liquids. Annu. Rev. Fluid Mech. 43, 661–690.
2. Theofanous, T. G. & LI, G. J. 2008 "On the Physics of Aerobreakup." Phys. Fluids 20, 052103.
3. Moylan, B., Landrum, B., and Russell, G., 2013: Investigation of physical phenomena associated with rain impacts on supersonic and hypersonic flight vehicles. The 12th Hypervelocity Impact Symposium, Procedia Engineering Volume 58, 2013, Pages 223-23.