Fluid Structure Interaction (FSI) Simulation of Flow through a Squeeze Bottle

V Turaga*, S Bhavi, M Soppin, V Srinivasa
HCL Technologies Ltd., Bangalore, India

ABSTRACT
In the present study, a Fluid Structure Interaction (FSI) simulation has been performed on a squeeze bottle. In a squeeze bottle operation, pressure is exerted on the container with the user's hand, due to which fluid within the container is compressed and thereby expelled through a nozzle. Volume changes due to wall deformation result in pressure rise in the fluid volume influencing the flow characteristics of the liquid being dispensed. The simulation presented here is an attempt made to capture wall deformation vs liquid outflow rate and outlet flow trajectory.

1. INTRODUCTION

1.1. Fluid Structure Interaction Simulation Method

Fluid–Structure Interaction (FSI) is the interaction between a solid structure which is moveable or deformable, with fluid flowing internally or surroundings. Consideration of interaction between fluids and structures is very important in many engineering systems. Examples of such systems include aircrafts, engines, bridges, moving containers, flow of blood through blood vessels and many more. The fluid flow may exert pressure and/or thermal loads on the structure. These loads may cause structural deformation significant enough to change the fluid flow itself. Computational Fluid Dynamics (CFD) and Structural Dynamics methods are used to capture the physics involving flow of fluids and behavior of structures respectively in the engineering systems. FSI simulation is a multi-physics study on how fluids and structures interact. The fluid and structural simulations are set up and solved simultaneously. While being solved, data is automatically transferred between the two solvers to achieve robust and accurate results.

1.2. Squeeze Bottle

A squeeze bottle is a type of container such as a plastic bottle for dispensing a fluid. These types of bottles are typically used for storing liquid detergents, ketchup/sauce, honey dispenser bottles, hand wash, hand sanitizers etc. These containers temporarily deform as a result of pressure exerted on the container walls by means of human hands. Volume changes due to wall deformation result in pressure rise in the fluid volume influencing the flow characteristics of the liquid being dispensed. Figure 1 shows a pictorial representation of how squeeze bottle operates.

*Corresponding Author: vijaykumar.t@hcl.com

Figure 1: Squeeze Bottle operation

1.3. Literature Review

A few research works have been published based on CFD and flow physics in bottles, FEA studies of plastic bottles and FSI techniques as well. Gene et al [1] reviewed numerical methods for solving FSI problems based on conforming and non-conforming meshes. Partition and immersed methods have been discussed for solving FSI problems with confirming and non-confirming methods respectively. Aleksander Karac [2] described FSI simulation between fluid and a flexible structure using a tool such as OPENFOAM and discussed one-system procedure where fluid and structure are modeled together as a single mesh and two-system procedure where fluid and structure are meshed separately. The two-system procedure applied for problems such as lung injury has been cited from references. Friedrich Geiger et al [3] described a CFD based methodology to predict the emptying time of a bottle initially filled with fluid, and various parameters impacting the emptying time. Sumarjumrat et al [4] described a coupled FEA method used to predict drop test of bottle half-filled with water, which was compared with experiment data. Mayer [5] discussed a bottle emptying exercise and the complex flow phenomenon involved in the process. Sandeep Kumar Rathia et al [6] studied the bottle emptying time with respect to different angles of inclinations based on experimental investigations and quantified through high-speed videos and presssure signals. Yuxin Jiang [7] evaluated the factors affecting flow rates of liquids through plastic bottles based on density of liquid and angle of inclination of bottle. Sameul Mer et al [8] presented a comparison between experiments on emptying of a water bottle, and numerical simulations based on a multiscale CFD approach.

Rohilla et al [9] discussed an experimental method used to analyze the fluidics such as bubble dynamics during emptying of a bottle. THONGKAEW et al [10] discussed the role of Finite Element Method (FEM) in coming up with optimum design of 6-liter PET (Polyethylene terephthalate) bottle. Huang et al [11] constructed a three-dimensional elastic-plastic dynamic finite element model to study the effect of the bottle thickness on critical load and stress distribution and validated by comparing with numerical results. Hu et al [12] implemented non-linear analysis to calculate buckling load and stress distribution based on the Abaqus/Explicit program. Based on stress contour of PET bottle obtained by Abaqus, plastic distribution of PET bottle was optimized in order to reduce the weight of PET bottle. Cho et al [13] investigated the large deformation characteristics of a PET bottle under a compressive load using experimental and FEA data. Demirel et al [14] performed FEA and optimized the dimensions of the PET bottle with petaloid shaped base against stress cracking. Yuan et al [15] analyzed the main factors of strength of a beer bottle. Van et al [17] discussed multiphase CFD methods.

2. METHODOLOGY

FSI simulation of bottle squeezing is carried out in 2 phases. ANSYS Workbench Mechanical and FLUENT solvers are used for the simulations.

PHASE-1: Standalone structural and CFD simulations: this includes separate simulations for structural and CFD without data transfer between solvers. Suitable assumptions have been made in applying the boundary conditions.

PHASE-2: FSI simulation of squeeze bottle by coupling of structural and CFD simulations: here, a complete 2-way coupling between Structural and Flow solvers has been implemented. A flowchart representing the process followed is shown in Figure 2.

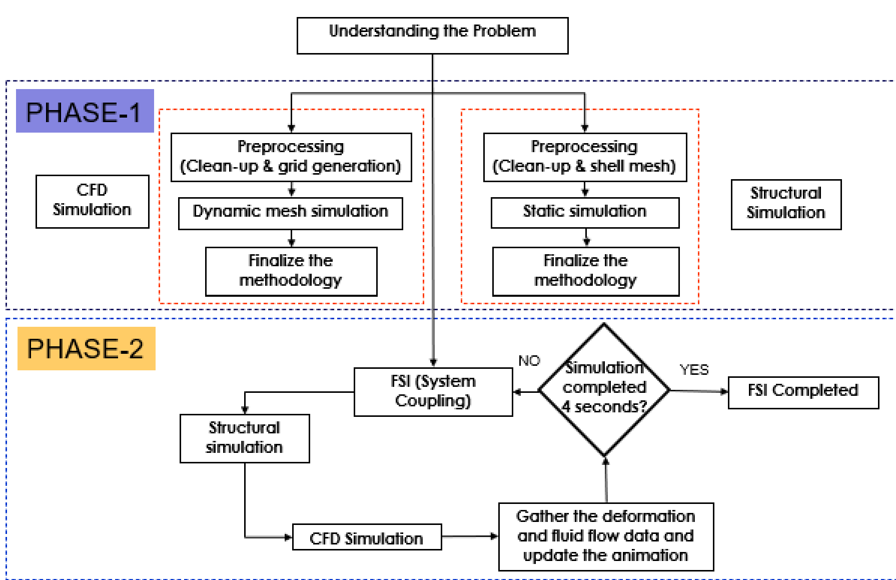


Figure 2: Flowchart showing the simulation procedure

Geometry considered for simulations is shown in Figure 3. Bottle structure is considered, which is assumed to be filled with phenol liquid. Rollers are modeled in structural model to apply pressure on the walls of the bottle. These rollers represent a scenario similar to human fingers pressing the bottle walls.

A cylinder is modeled at the mouth of the bottle in CFD model to represent a container which is filled with fluid exiting from the bottle on applying pressure. This is assumed to be filled with air initially.

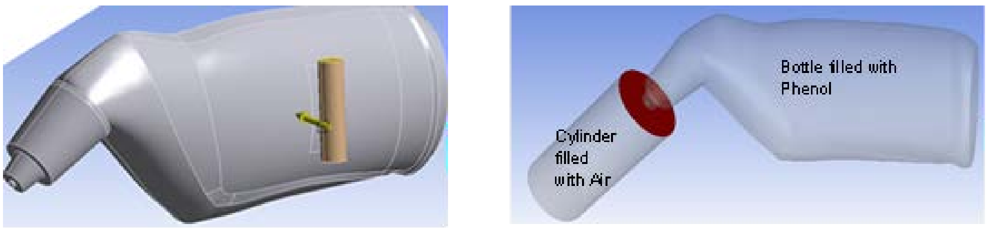


Figure 3: Structural (left) and CFD (right) geometries

Plastic (Polyethylene Terephthalate) is used as material for the bottle. Properties considered are listed in Table 1.

Table 1: Plastic (Polyethylene Terephthalate) properties

Property	**Value**
Density	1200 kg/m^3
Young’s Modulus	2000 MPa
Poisson’s Ratio	0.4
Bulk Modulus	3.33 x 10^9 Pa
Shear Modulus	7.14 x 10^8 Pa

Phenol is used as a primary liquid and air is used as a secondary liquid. Phenol properties are listed in Table 2.

Table 2: Phenol properties

Property	**Value**
Density	1071 kg/m^3
Viscosity	0.008 kg/m-s

"Volume of Fluid” Multiphase model is used. Gravity is activated.

2.1. Phase-1: Standalone Simulations

In this phase, structural and CFD simulations are performed independently. In the structural simulation, cap and bottom surface of bottle are fixed. Transient simulation has been performed for 4 seconds duration. Displacement was applied on rollers as provided in Table 3.

Table 3: Time vs Displacement

	Displacement (mm)
0	0
2	4
4	-4

Deformation of the bottle surfaces has been evaluated. The boundary condition and load condition used for structural simulation is shown in Figure 4.

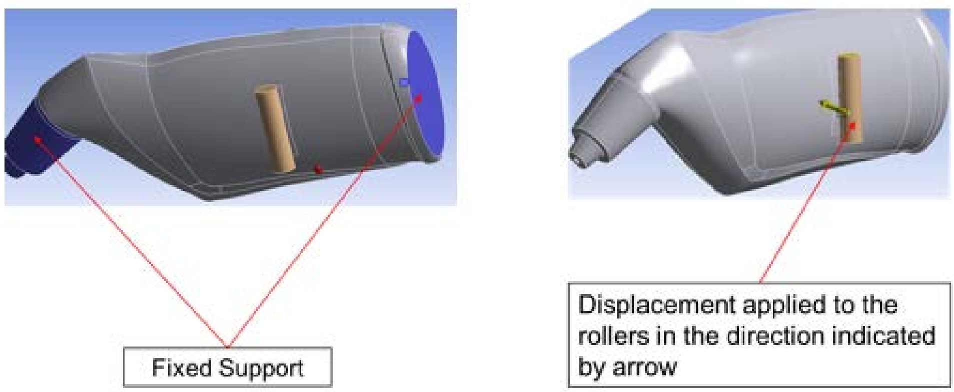
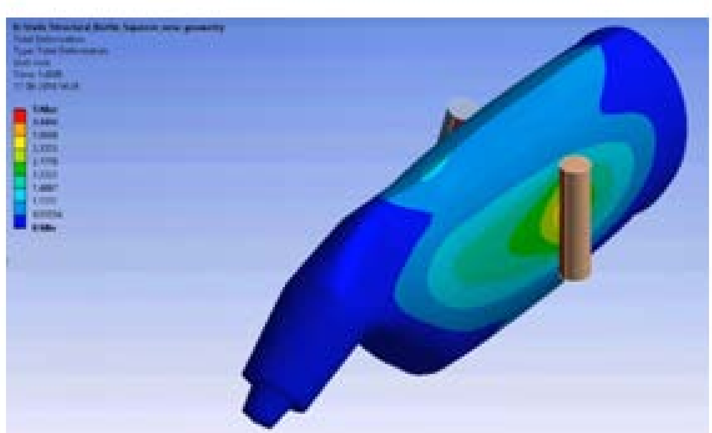
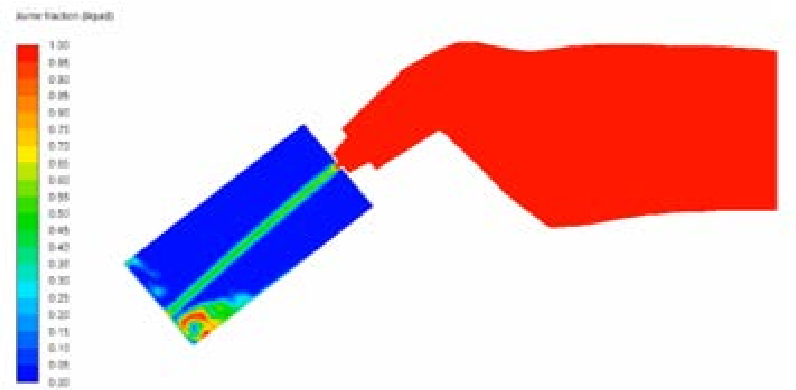


Figure 4: Structural Boundary condition (left) and Load condition (right)

In standalone CFD simulation, fixed pressure is applied on the bottle faces for the duration of simulation. At initial timestep, bottle is filled with phenol and cylinder is filled with air. Dynamic mesh is used to capture deformation of the bottle due to pressure on external walls, which in turn would cause fluid to flow into cylindrical container. Figure 5 shows plot showing the bottle deformation when the rollers move inwards and corresponding fluid flow out of the bottle.



(a)



(b)

Figure 5: Bottle structural deformation (a) and volume fraction of phenol (b)

2.2. Phase-2: Fully Coupled FSI Simulation

In this phase, structural and CFD simulations are coupled together by means of System Coupling component in Ansys Workbench. The displacement boundary condition is applied to rollers in structural simulation and multiphase, dynamic mesh conditions are applied in CFD simulation. These conditions are same as in Phase-1. Once the FSI simulation is started, initially structural simulation was solved for the given time step size and the displacement data obtained from the structural simulation was applied in CFD to solve for the fluid discharge. This process was repeated until the solution reaches the maximum time step of 4 sec. The plot showing the volume fraction of fluid during FSI simulation is shown in Figure 6.

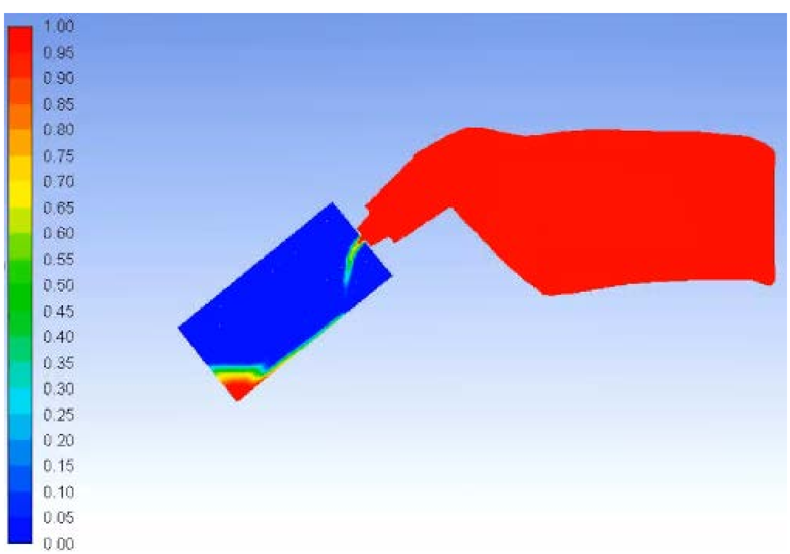


Figure 6: Volume fraction of phenol – FSI Simulation

3. OBSERVATIONS

In Phase-1, standalone simulations were carried out to study the behavior of the bottle in Structural and CFD. It was observed that the methods used for the Structural and CFD simulations were appropriate, and results obtained from the stand-alone simulations are acceptable. However, the results are only representative, since exact deformation of bottle from structural simulation is not provided as input for CFD simulation. This phase has helped in finalizing the models, boundary and loading conditions used for the simulation.

In Phase-2 simulation, 2-way coupling helped in exact representation of the deformation and its impact on the fluid flow. It was observed that rollers modelled in the structural simulation took initial second of time to touch the side surface of the bottle. Because of this, initially no deformation was observed and similarly in the 4th second as well. Since the gravity was applied, liquid flow was noticed during 1st and 4th seconds. This is in line with the practical application.

4. CONCLUSION

The simulation has been carried out in 2 phases - phase-1 with standalone structural and CFD simulations and phase-2 with fully coupled FSI simulation. While standalone simulations helped in freezing the meshing and solution methods of respective simulations, fully coupled FSI simulation has helped in capturing the real time physics involving deformation of bottle and resulting ejection of fluid. This simulation has helped in developing a methodology to capture the operation of squeeze bottle using Fluid-Structure Interaction method. The results of the simulation can help the designers to identify the areas of improvement and improve the flow trajectory.

REFERENCES

[1] Hou, Gene, Jin Wang, and Anita Layton, "Numerical methods for fluid-structure interaction - a review", Communications in Computational Physics 12.2 (2012): 337-377.

[2] Karac, Aleksandar, "From Plastic Pipes and Bottles to Bioengineering Applications: Fluid-Structure Interaction Procedures for Flexible Systems".

[3] Friedrich Geiger, Kai Velten, Frank-Jürgen Methner", 3D CFD simulation of bottle emptying processes", Journal of Food Engineering, Volume 109, Issue 3, 2012.

[4] Suvanjumrat, Chakrit, Tumrong Puttapitukporn, and Satjarthip Thusneyapan. "Analysis of fluid-structure interaction effects of liquid-filled container under drop testing", Agriculture and Natural Resources 42.1 (2008): 165-176.

[5] Mayer, Hans C. "Bottle Emptying: A Fluid Mechanics and Measurements Exercise for Engineering Undergraduate Students" Fluids 4.4 (2019): 183.

[6] Rathia, Sandeep Kumar, Lokesh Rohilla, and Arup Kumar Das. "Effects of Slugging and Flooding on Bottle Emptying Time for Different Inclinations".

[7] Jiang, Yuxin. "Experimental and Theoretical Analysis of the Factors Influencing the Flow-out Rate from a Bottle", Journal of Physics: Conference Series. Vol. 1865. No. 3. IOP Publishing, 2021.

[8] Mer, Samuel, et al. "Simulating the emptying of a water bottle with a multi-scale two-fluid approach", Fluids Engineering Division Summer Meeting. Vol. 51579. American Society of Mechanical Engineers, 2018.

[9] Rohilla, Lokesh, and Arup Kumar Das. "Fluidics in an emptying bottle during breaking and making of interacting interfaces", Physics of Fluids 32.4 (2020): 042102.
[10] THONGKAEW, Kunlapat, and Thanwit NAEMSAI. "Mechanical Properties and Cost-Minimized Design of 6-liter PET Bottle Using Finite Element Method" Walailak Journal of Science and Technology (WJST) 17.6 (2020): 579-587.
[11] Huang, Hsing-Hui, et al. "Design and simulation analysis of lightweight HDPE milk bottle" Polymers and Polymer Composites 26.1 (2018): 91-98.
[12] Hu, Qing Chun, et al. "Structural optimization and lightweight design of PET bottle based on ABAQUS", Advanced Materials Research. Vol. 346. Trans Tech Publications Ltd, 2012.
[13] Cho, S. H., et al. "Numerical analysis and Experiment to Determine Deformation Characteristics of PET Bottle under Compressive Load", Journal of The Korean Society of Manufacturing Technology Engineers 23.1 (2014): 83-86.
[14] Demirel, B., F. Daver, and E. Kosior. "Optimum design of pet bottle bases against stress cracking", SPE-ANTEC Proceedings, Cincinnati (2007): 2146.
[15] Yuan, Wei, et al. "Research on PET Beer Bottle Structural Parameters and it’s Strength", Advanced Materials Research. Vol. 641. Trans Tech Publications Ltd, 2013.
[16] Bakker, André, "Lecture 1-Introduction to CFD Applied Computational Fluid Dynamics." Fluent Inc.: Queens, NY, USA (2002).
[17] Van Wachem, B. G. M., and Alf-Erik Almstedt. "Methods for multiphase computational fluid dynamics", Chemical Engineering Journal 96.1-3 (2003): 81-98.
[18] Rao, Achuth. "Fluid—solid interaction analysis using ANSYS/multiphysics", Computational Fluid and Solid Mechanics 2003. Elsevier Science Ltd, 2003. 1492-1496.
[19] Menter, Florian, et al. "Overview of Fluid-structure coupling in ANSYS-CFX", International Conference on Offshore Mechanics and Arctic Engineering. Vol. 47497. 2006.
[20] Kurowski, Pawel. Finite element analysis for design engineers. SAE, 2017.
[21] Ansys 19.0 Workbench Help.