Investigation of Fluid Vortex Development in a Mixing Tank using Computational Fluid Dynamics

Devin O'Donnell
Research mentor: Dr. Javad Zeinali
UCD College of Engineering & Architecture
University College Dublin

Introduction

Computational Fluid Dynamics (CFD) uses computers to predict and simulate fluid motion based on fluid mechanic principles.

CFD utilizes numerical solving methods to solve differential equations relating to flow.

It is used in many industries, such as Aerospace and Defense, Healthcare, Biopharmaceutical, automotive and agriculture.

Within these industries, CFD is used for optimization, efficiency increases and technology advancements. It is used to increase productivity while minimizing overhead cost of production.

CFD is performed by using a simulation software (such as OpenFOAM) and a visualization software (Paraview). Here the simulation is optimized by the researcher

Research Goals

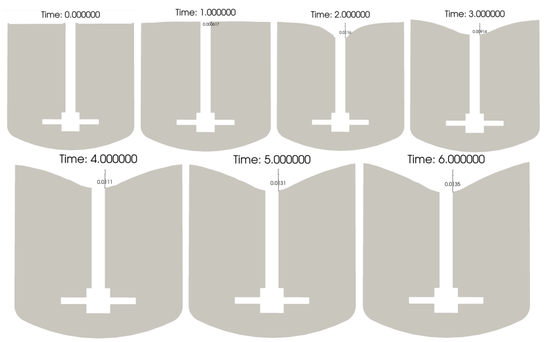
- Use CFD to run a fluid mixing simulation on a 1000 mL tank
- Use the data computed by the simulation to investigate the vortex development and flow fields within the tank
- Develop an understanding of CFD principles to be used in future works

Methodology

- Develop the simulation code to be run by the OpenFOAM software for the tank and impeller
- Run the simulation in OpenFOAM to collect data
- Use data collected by the simulation to run Paraview and visualize/analyze the software

Results

Figure 3: Vortex Depth (in Meters) with respect to Time

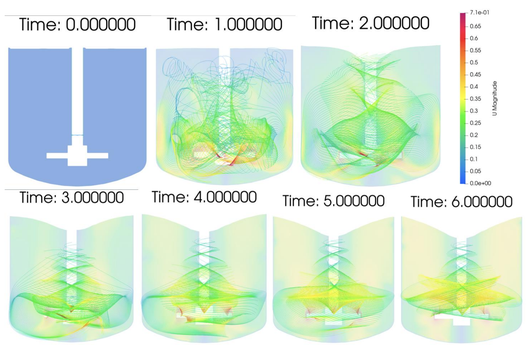


Here, the fluid vortex depth with respect to time can be seen. It demonstrates how the vortex develops in time, and how it tends towards a depth at which it will reach equilibrium.

The vortex quickly develops from time one to time two, as it can be seen with the drastic depth increase

This shows the development of the vortex using stream traces from the side of the tank
It shows the velocity in the tank changing with time using the velocity profile.

Figure 2: Side View of Developing Vortex with Respect to Time, at 1 Second Increments from Time One to Time Six using Stream Traces, Overlaying a Velocity Profile Map



These velocity profiles follow a trend of increasing magnitude, with the lowest average velocity at time one, and the highest average velocity at time six.

Discussion

- Vortex quickly developed from times 1-2, then equilibrated for the remainder of simulation
- As the vortex develops, rotational velocity increases due to the spinning impeller.
- The simulation was consistent with the idea that the vortex would form with time
- Impeller was successful at increasing fluid velocity to cause vortex formation
- Vortex depth is a function of fluid rotational speed
- Future experiments are needed to better understand equilibria of vortex depth

Conclusion

- Based on these results, it can be concluded that vortex formation is crucial to mixing, fluid rotational speed is important to vortex formation

- Future works should include higher detailed mesh, and longer run-time for accurate data

References

1. Ansys. *What Is Computational Fluid Dynamics (CFD)? | Ansys*, Ansys Engineering Simulation Software, www.ansys.com/simulation-topics/what-is-computational-fluid-dynamics. Accessed 19 Nov. 2024.
2. "CFD for Mixing Tanks." *Resolved Analytics*, Resolved Analytics , www.resolvedanalytics.com/cfd-applications/cfd-for-mixing-tanks#:~:text=In%20the%20chemical%20industry%2C%20CFD%2Denhanced%20mixing%20tanks%20are%20used,increased%20production%20and%20reduced%20costs. Accessed 19 Nov. 2024.
3. Ganguli, Arijit. *Computational Fluid Dynamics (CFD): The Role in Scale-up of Process Equipments*, Ahmedabad University - Leading Liberal Education and Research University, 26 Oct. 2021, ahduni.edu.in/blog/computational-fluid-dynamics-cfd-the-role-in-scale-up-of-process-equipments/.
4. Giraldo, Guillermo. "CFD Modeling, Analysis & CFD Simulation for Beginners." *SimScale*, 14 Mar. 2024, www.simscale.com/blog/cfd-analysis-for-beginners/.
5. Goswami, Aanak. "Vortex Mixer: Stirring up Success in the Lab." *Accumax*, 4 June 2024, www.accumaxlab.com/mixing-with-vortex-mixer/#:~:text=At%20the%20heart%20of%20a,rotational%20movement%20throughout%20the%20sample.
6. Lea, Jimmy. "How Accurate Are CFD Simulation Results?" *Cleanroom Technology*, Cleanroom Technology, 20 Apr. 2023, cleanroomtechnology.com/how-accurate-are-cfd-simulation-results--208207.
7. Lomax, H., Pulliam, T., Zingg, D., and Kowalewski, T. (July 30, 2002). "*Fundamentals of Computational Fluid Dynamics*." ASME. *Appl. Mech. Rev*. July 2002; 55(4): B61. https://doi.org/10.1115/1.1483340
8. "Methods for Visualizing CFD Data Sets." *Resolved Analytics*, Resolved Analytics , www.resolvedanalytics.com/cfd-in-practice/how-to-visualize-cfd-simulation-result#:~:text=By%20showcasing%20complex%20CFD%20results,understanding%20of%20the%20analyzed%20phenomena. Accessed 20 Nov. 2024.
9. Tryggvason, Gretar. (2016). Computational Fluid Dynamics. 10.1016/B978-0-12-405935 1.00006-X.