Simulating Impeller Mixing Under Various Conditions

Heather Phillips [1], Jonathan Durbin [2], Kurt Aikens [3]
Department of Physics, Houghton College, 1 Willard Ave., Houghton, NY 14744
[1]heather.phillips18@houghton.edu, [2]jonathan.durbin19@houghton.edu, [3]kurt.aikens@houghton.edu

Abstract

Impeller mixing, owing to its widespread industrial use, can cause significant financial losses if not performed efficiently. Computational fluid dynamics (CFD) can greatly aid in the design of mixing systems when it is sufficiently accurate. For this work, fluid mixing is performed using an A200 impeller operating in a baffled tank. Simulations were performed of this setup using different approaches and under various operating conditions. Results are compared to available experimental data to guide the development of an overall ANSYS Fluent mixing prediction methodology. Qualitative trends of the experimental data were successfully predicted and discrepancies were generally less than 20%. Overall, the methodology appears promising and will be used in the future to make more detailed predictions of mixing when the flow is transitional – a historically challenging task. Additionally, the approximate solution approach utilized here will be further examined in an attempt to reduce discrepancies between the simulation results and those of the experiments.

Computational Methodology

There are few exact solutions to the governing equations for fluid dynamics. However, approximate solutions can be obtained using computers. To approximate the solution to a fluid dynamics problem, the fluid domain must first be discretized by generating a mesh (see Figure 1 for an example). When meshing, one is dividing the fluid volume into discrete cells of various shapes and sizes. After specifying appropriate boundary conditions for the problem, a computer uses numerical methods to estimate the fluid solution for each cell. ANSYS Fluent 17.1 is used in this work for generating the mesh and solving. Specifically, Fluent's coupled, second-order solver was used with default solution control values (such as Courant number, under-relaxation factors, etc.), unless otherwise noted. The fluid inside the tank is assumed to be incompressible (i.e., constant density) and the realizable $k-\epsilon$ turbulence model with the Menter-Lechner wall treatment is utilized [2]. The supercomputer Bridges, located at the Pittsburgh Supercomputing Center (PSC), was used for the present calculations. On average, 28 cores were used and each simulation lasted roughly 10 hours.

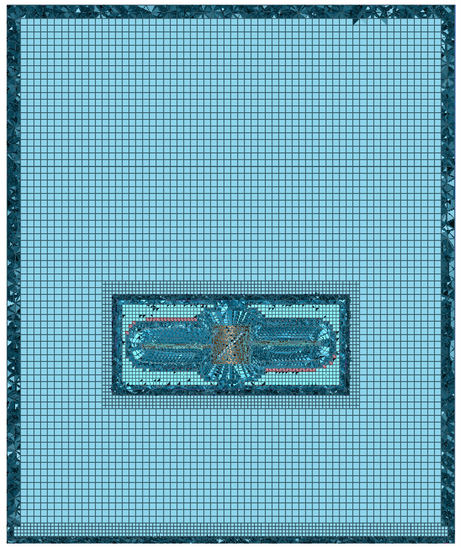

Figure 1: An example of a mesh for CFD. A denser mesh is required in regions where fluid flow is more complex, such as near the impeller and at the boundary layers.

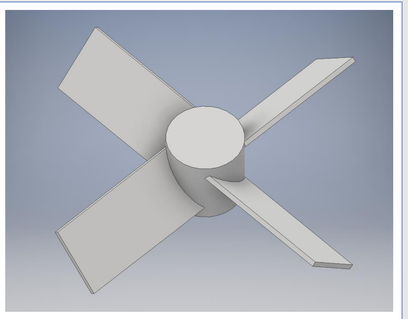

Figure 2: Geometry CAD file of the A200 impeller.

Impeller Simulations

The geometry for an A200 impeller was provided by SPX Flow. Using Autodesk Inventor, the A200 impeller, shown in Figure 2, was placed inside a "baffled tank". Baffles, which aid the mixing process, are vertical walls that extend into the interior of the tank.

A multiple reference frame (MRF) simulation is performed here to capture both the rotation of the impeller and the fixed baffles. Following the MRF approach, the fluid region of the mixing system was split into two zones. The first zone is referred to as the "tank zone". It includes the fixed tank and baffles but has a cylindrical cavity in the middle. The second zone fills the cavity in the tank zone and includes the impeller geometry. This is referred to as the "impeller zone". Figure 3 shows the two zones in the geometry set up. The impeller zone is set to rotate relative to the tank zone, however the geometry of the impeller does not change position relative to the tank. Therefore, a steady-state simulation is performed. This is the MRF model assumption: that the flow near the impeller and the flow near the baffles are de-coupled, so that a fully-rotating impeller simulation is not required.

Once a surface mesh was formed for the given geometry, the volume mesh was calculated using Fluent Meshing's "auto mesh" feature. Within the auto mesh settings, a zone-specific prism layer was created around the impeller. The volume fill was set to use the hexcore method. Additionally, refinement regions were used to increase the number of cells around different features. Baffle and impeller zone refinements are shown in Figure 1 but a refinement region was also added to the bottom of the tank. The mesh for the case described here contained 2.3×10^{6} cells.

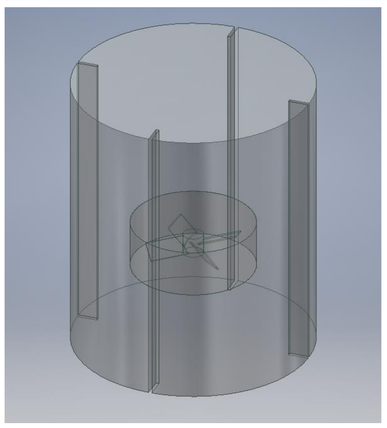

Figure 3: Geometry CAD file of the A200 impeller mixing case. The two "zones" are shown: the cylindrical impeller zone on the inside and the tank zone surrounding it.

Results

When performing fluid dynamics simulations, the power number and Reynolds number are the most important non-dimensional parameters. The Reynolds number is given by $\mathrm{Re} = \rho N D^2/\mu$, where ρ is the fluid density, N is the impeller rotation rate (in revolutions per second), D is the diameter of the impeller, and μ is the fluid viscosity. The Reynolds number fully specifies the state of the system and is an input in our simulations. The power number is a measure of the power the impeller requires for mixing at a given Reynolds number. It is given by $N_P = 2\pi\tau/\rho N^2 D^5$, where τ is the torque required to turn the impeller and the other variables are the same as before. The power number is calculated based on the simulation results.

The simulations for various Reynolds numbers were completed and compared to the experimental power number results from SPX Flow [1] as shown in Figure 4. The simulation data follow a similar trend to that of the experimental data which is encouraging. To get a more qualitative idea of flowfield patterns, Figure 5 shows streamlines for the lowest Reynolds number case, the highest, and one in between. The streamlines are calculated on a vertical plane that is halfway through the fluid domain and cuts through the impeller. While no similar experimental results are available, one expected trend is noticed. Namely, as the Reynolds number increases, the flow becomes more axial through the impeller [3]. At the lowest Reynolds number the flow is decidedly radial exiting the impeller with two separate and counter-rotating regions of fluid. These disappear at higher Reynolds numbers.

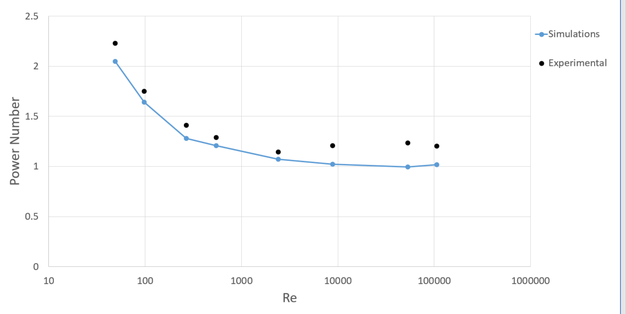

Figure 4: Simulation results compared to SPX Flow experimental data [1].

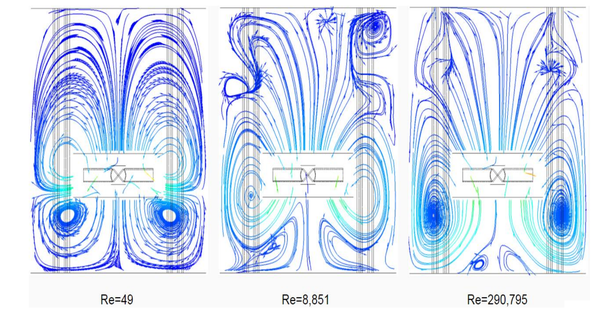

Figure 5: Streamlines constrained to a vertical plane halfway through the fluid domain.

Conclusions

A set of simulations were completed to develop a methodology using ANSYS Fluent to accurately predict impeller mixing flowfields. The simulation results were compared to experimental data collected by SPX Flow [1]. The results suggest that using the realizable k-ϵ turbulence model in Fluent is a reasonable place to start when simulating impeller mixing. In the future, refinements on the present methodology will be sought in an attempt to gain greater congruence with available experimental data. For example, to test the applicability of the MRF approach, simulations are presently being performed with the impeller and baffle geometries oriented differently relative to each other. To more thoroughly test simulation accuracy, additional experimental data for validation would be helpful. With only experimental power number results to compare with, there are only a small number of conclusions that can be drawn. There is a need for different types of experimental data such as velocity at various points in the system and for qualitative comparisons with simulations. In the more distant future, the method should be further extended and tested. For example, the fluids simulated here were Newtonian but many practical mixing applications involve non-Newtonian fluids. Additionally, the methodology will be more extensively used and examined in the transitional mixing regime.

Acknowledgements

The authors would like to thank Dr. David Foster from the University of Rochester for suggesting this research topic, providing feedback and guidance, and connecting us to researchers at SPX Flow. We are also grateful to those at SPX Flow for providing experimental data and the A200 CAD file. The present work would not have been possible without these assets. Similarly, this work was greatly supported by XSEDE which provided us access to PSC Bridges. Lastly, this research would not have been possible without the opportunity given by the Houghton College Summer Research Institute.

References

1. R.O. Kehn, S. Xu, and A. Strand (private communication).
2. S. M. Kresta, V. A. Atiemo-Obeng, and E. L. Paul, *Handbook of Industrial Mixing: Science and Practice*, (John Wiley & Sons, 2017).
3. R. J. Weetman, in *Fluid Mechanics of Mixing: Modelling, Operations and Experimental Techniques*, edited by R. King (Kluwer Academic Publishers, 1992), pp. 19-27.