CFD Studies Of Educational Closed Loop Water Tank

R. Wulandana[1]

[1]State University of New York, New Paltz, NY, USA

Abstract

This paper describes the utilization of COMSOL Multiphysics® package for the finite element modeling of an educational closed-loop water tank. The custom water tank was built to be part of the fluid mechanics lab belonging to the Mechanical Engineering Program of SUNY New Paltz, NY. The design is constrained by the need to deliver a maximum of about 60 cm per second of flow speed through its 15-by-15 cm square transparent observation channel. The total length of the movable flow tank should be limited to be around seven (7) feet so that it can be fit into the freight elevator in the school. The high cost of such educational water tanks available in the market motivated the project to establish a custom-made device. The Computational Fluid Dynamics (CFD) Module is used in conjunction with the K-epsilon turbulent model to study water behavior in the flow tank, particularly in the observation chamber, using full three-dimensional modeling. As the Reynolds number can reach 90,000, the laminar model is not suitable to be used in the current study despite its easiness for convergence. A convergence study was performed for various mesh refinement and method to ensure that the outcomes are verified. The utilization of partial mesh refinement, focusing on the observation chamber and inlet pipe, shows significant reduction in the computation time as well as increase in solution convergence compared to the global mesh refinement. It was found that that near-fully developed flow profile can be established in the observation chamber. This condition is required for various experiments that will be conducted such as the drag and lift force measurement of objects exposed to fluid flow. During the modeling, an inclusion of internal T-pipe model installed in the collection chamber was found to be essential for the establishment of the fully developed flow in the observation chamber. The CFD model shows twin circle flow pattern at the exit of the observation chamber. This pattern can be observed in the back chamber of the established water tank. The solutions from three-dimensional models are expected to provide estimation for boundary conditions needed for two-dimensional modeling of problems involving objects that are exposed to flow between walls.
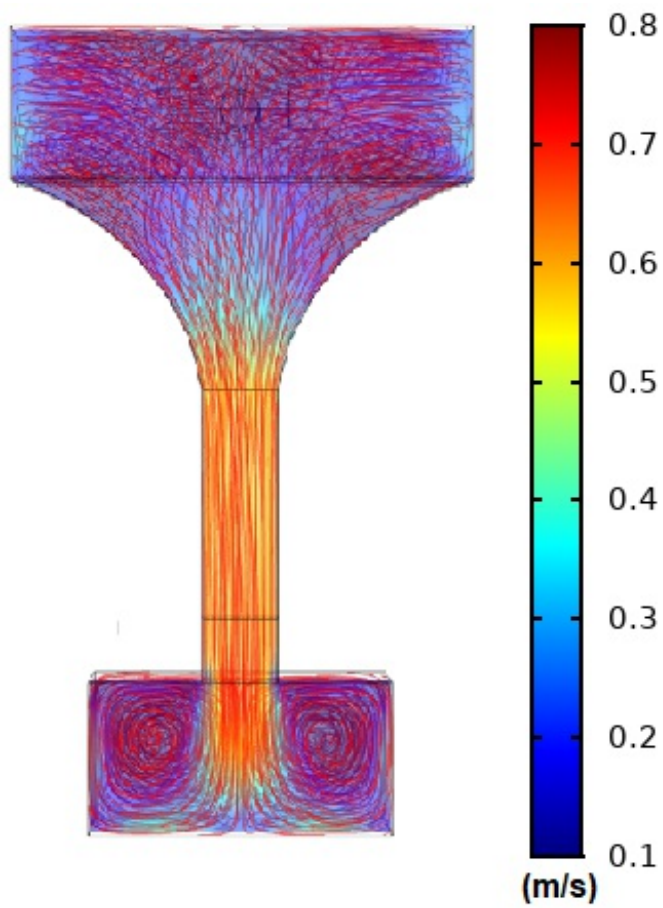
Figures used in the abstract
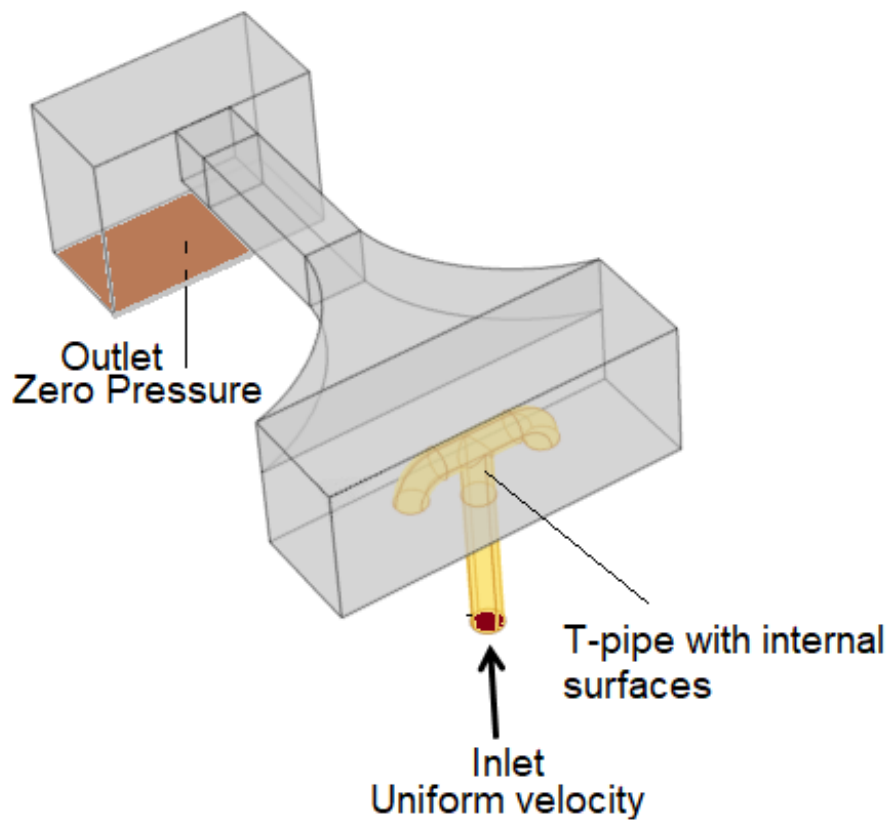
Figure 1 : The 3D model shows the inlet and outlet boundary conditions as well as the internal T-pipe model.
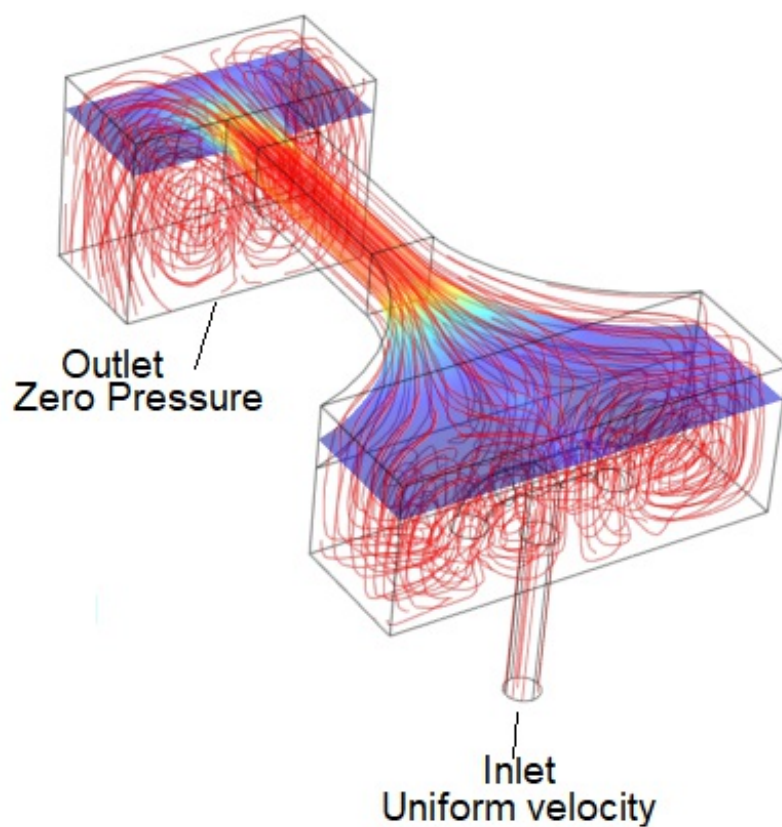
Figure 2 : The 3D streamlines in the flow tank shows the circular flow pattern exiting the observation chamber.

Figure 3 : Streamlines and velocity map shown from top view for the flow around 65 centimeter per second of average speed in the observation chamber.