CFD Studies of Educational Closed Loop Water Tank

R. Wulandana[1]

1. Mechanical Engineering Program, SUNY New Paltz, New Paltz, NY, USA

INTRODUCTION: Presented in this poster, computational fluid dynamic studies of medium-sized open water flumes. Water flume tanks with closed-loop circulation are essential for fluid mechanics experimentation. Its simple maintenance and versatility are attractive for educational setting. Constrained by limited budget, the newly developed Mechanical Engineering Program of SUNY New Paltz decided to construct such device for its fluid mechanics lab. The 3-hp centrifugal pump is employed to provide 60 cm/s water flow through its 15x15x60 cm^3 transparent observation chamber. Such speed corresponds to Reynolds number of ~ 90K. The total material cost and welding gas to build the device was less than \$7000. The finished flume is shown in Fig. 1 below. The total length of the flume is ~ 2 m and its width is ~ 60 cm.

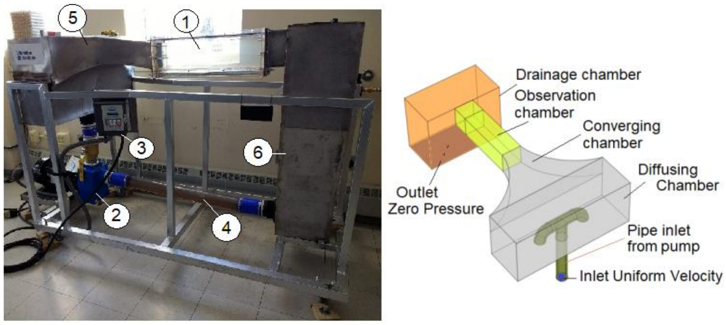

Figure 1. Left - The finished flume is equipped with (1) transparent observation chamber, (2) 3-hp centrifugal pump, (3) Variable Frequency Drive (4) Return pipe to the pump (5) Inlet manifold into the observation chamber and (6) drainage tank. Right – CFD model of the flume with the corresponding sub-domains and the corresponding inlet and exit sections. The observation chamber length is 60 cm.

COMPUTATIONAL METHODS: The Fluid Dynamics module of COMSOL Multiphysics 5.4 is the prime tool in this project. Reynolds-average Navier-Stokes (RANS) model was selected to model the turbulent flow. The default parameters for the K-ε model are selected and the built-in Water model available in COMSOL is taken as the working fluid. All models are studied as steady state. The Navier-Stokes equation and conservation of mass are presented below for clarity, together with the RANS turbulent model.

$$\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \nabla\cdot\left[-p\mathbf{I}+\mathbf{K}\right]+\mathbf{F} \qquad \rho\nabla\cdot(\mathbf{u}) = 0$$

$$\mathbf{K} = (\mu+\mu_T)\left(\nabla\mathbf{u}+(\nabla\mathbf{u})^T\right)$$

$$\rho(\mathbf{u}\cdot\nabla)k = \nabla\cdot\left[\left(\mu+\frac{\mu_T}{\sigma_k}\right)\nabla k\right]+P_k-\rho\epsilon \quad \epsilon = \mathrm{ep}$$

$$\rho(\mathbf{u}\cdot\nabla)\epsilon = \nabla\cdot\left[\left(\mu+\frac{\mu_T}{\sigma_\epsilon}\right)\nabla\epsilon\right]+C_{\epsilon1}\frac{\epsilon}{k}P_k-C_{\epsilon2}\rho\frac{\epsilon^2}{k},$$

BOUNDARY CONDITIONS: At the inlet, which represents the conduit originated from the centrifugal pump, a uniform velocity is assumed. At the exit, zero pressure is assumed at a cut plane of the drainage tank. The inlet pipe is extended into the diffusing chamber and split into two pipes via a T pipe (shown in green) and its surfaces, except its outlets, are conveniently modeled in COMSOL as Interior Wall.

RESULTS: This report highlights effects of mesh generation on the accuracy of the outcome. Focusing on the flow in the observation chamber, the accuracy is measured by comparing its flow rate $Q = \int \boldsymbol{V}dA_{ch}$ (expected parabolic velocity in the chamber) to one at the inlet $Q = V_{in}A_{in}$ (uniform velocity input at the inlet). Two mesh schemes are studied; global refinement and partial refinement. In the global (uniform) refinement, all domains are discretized using the same mesh size. Shown in Figure 2 below, mesh generation of the model with non-uniform refinement, where the sub-domains have been discretized using different mesh size. Shown in Figures 3 and 4, streamlines in the flume. The flume design produces straight streamlines in the observation chamber, suitable for fluid experiments.

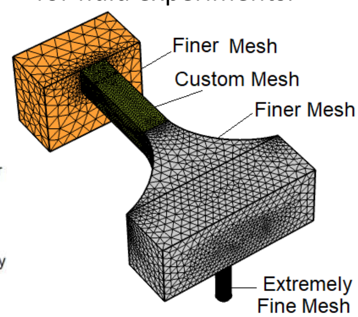

Figure 2. Non-uniform mesh refinement focusing on the observation chamber.

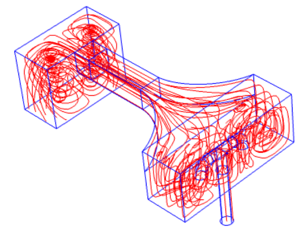

Figure 3. The streamlines show turbulent pattern of flow in the diffusing chamber. The flow shows predominantly straight linear lines in the observation chamber after passing the converging chamber.

RESULTS ON ACCURACY: Shown in Figure 5, comparison of accuracy and computation time by different mesh generation. Circles and squares represent data from global (uniform) and non-uniform mesh generation, respectively. The triangles show data from Laminar modeling (shown for a reference). It can be shown that the high accuracy given by non-uniform mesh despite the low number of elements. This results in low computation time. The uniform mesh requires large mesh volume and computation time in order to achieve high accuracy.

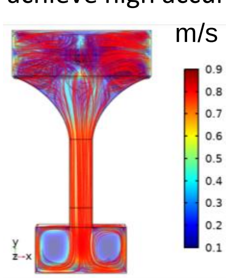

Figure 4. Streamlines and velocity field when input velocity is 3 m/s.

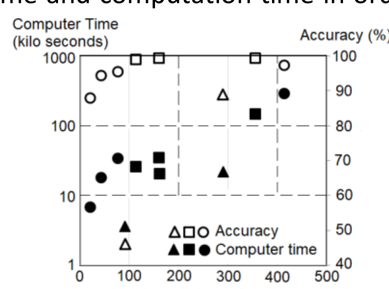

Figure 6. Number of elements versus computation time and accuracy for various mesh generation.

CONCLUSIONS: The CFD module of COMSOL has been used to model fluid flow for design of an open flume. Non-uniform mesh generation and turbulent model have been found the be the most effective discretization method.

REFERENCES:

1. Walter Frei, "Which Turbulence Model Should I Choose for My CFD Application?", COMSOL Blog (web)