

Figure 12-1. Swirl angle measurement location in a physical model.

In a physical model, the swirl angle is measured with a swirl meter, as defined in the HI standards. There are a variety of ways in which this swirl angle can be calculated using CFD. One method to calculate swirl angle from CFD data is by calculating the arctangent of the average tangential velocity divided by the average normal velocity in the pipe cross section, as shown in Equation (12-1):

$$\alpha = \tan^{-1} \left(\frac{\overline{U_{\tan}}}{\overline{U_{norm}}} \right)$$
(12-1)

The acceptable limit for time-averaged swirl angle is 5 degrees in a physical model. Swirl angle from CFD analysis is not recognized by the Hydraulic Institute for a measure in meeting the specific criteria of 5 degrees. Therefore, swirl angle calculations in a CFD analysis should only be used as a general comparison of compliance with HI criteria.

An intake velocity distribution is defined as acceptable if the velocity throughout the cross section at the pump intake is within 10% of the average velocity. Figure 12-2 shows an example of how the velocity distribution is measured in a physical model.

The velocity at the end of the bell in a CFD model can be extracted from each of the finite volume locations to produce a velocity distribution at the pump



Figure 12-2. Physical model velocity measurement probe and measurement location pattern in the pump intake bell.



Figure 12-3. Velocity distribution examples from CFD analysis.

suction. These velocities can be plotted with a velocity legend to illustrate areas that are greater than and less than 10% of the average velocity. For example, in Figure 12-3 the areas that are dark red are greater than 10% higher than the average velocity, and those areas that are dark blue are less than 10% below the average velocity. These areas of dark red and dark blue do not meet the HI criteria. Figure 12-3 shows an example of poor velocity distribution on the left and good velocity distribution on the right.

In general, desirable flow approach characteristics consist of an approach velocity less than 0.5 m/s (1.5 ft/s), adequate depth to reduce the potential of surface vortices, a constant flow acceleration toward the pump, and no regions of flow separation or recirculation. Regions of recirculation can spawn surface and

subsurface vortices, which can be detrimental to pump operation. The HI standards provide a classification for the various types of surface and subsurface vortices that may occur in a physical model. In addition, the HI standards provide criteria for which types of vortices are acceptable and which types are not acceptable. The HI standard vortex types are shown in Figure 12-4. The vortex types within the red boxes do not meet the HI acceptance criteria.

CFD is not able to specifically identify the vortex classification type as defined by HI standards. However, through the use of tools developed by comparison of hydraulic parameters to physical model vortex activity, CFD analysis can be used



Figure 12-4. Classification of free-surface and subsurface vortices, as defined by HI standards.

to identify areas of potential vortex activity. Figures 12-5 and 12-6 show a comparison between vortex activity in a physical model, as visualized using tracer dye, and potential vortex activity in the CFD model, as identified by lambda 2 values and marked with streamlines.



Figure 12-5. (a) Physical model subsurface vortex activity compared to (b) CFD analysis of potential vortex activity.



Figure 12-6. Physical model surface vortex activity compared to CFD analysis of potential vortex activity.

12.3 PHYSICAL MODEL AND CFD COMPARISONS

When using CFD to evaluate approach conditions for pump stations, it is important to understand the advantages and disadvantages of both physical modeling and CFD modeling of pump station approach conditions. The following should be considered when deciding on the modeling approach and the use of CFD:

- Physical modeling is accepted by the Hydraulic Institute, whereas CFD modeling is not.
- CFD modeling is usually an order of magnitude less expensive.
- CFD modeling can be performed with significantly less time.
- CFD modeling allows for efficient evaluation of several alternatives.
- Physical modeling can predict vortex classification, whereas CFD modeling can only identify potential vortex activity.
- CFD modeling can predict swirl angles; however, the reference criteria are based on the physical model testing vane. In addition, vortices can affect swirl angle, which CFD does not capture with average velocity vector calculations.
- CFD provides much more spatial detail about velocity distribution. Timevarying analysis can be performed using transient CFD simulation methods. However, this analysis requires additional computational time and resources to generate accurate results.

Although significant progress has been achieved using CFD in evaluation of pump approach conditions, additional work is still required to develop reliable CFD pump intake evaluation methodologies. A summary of conclusions developed between previous comparisons of CFD to physical model evaluations is listed here:

- Vortex comparison: Vortex algorithms, such as lambda 2 or swirling strength, provide the best correlation to physical model results.
- Swirl comparison: Trends of swirl angle are captured by CFD, but individual swirl measurements are loosely correlated to physical model observations. In addition, CFD results are significantly affected by vortex activity, and they often underpredict swirl angle when there is significant vortex activity.
- Velocity distribution comparison: Velocity comparisons show generally good agreement between CFD and physical modeling, but they are sensitive to where the physical model measurements are taken. In addition, physical models may miss regions of high or low velocity within the intake velocity plane.
- CFD results are extremely sensitive to the volume mesh size, meshing schemes, model run times, turbulence models, and time steps. Generally, sensitivity analyses are recommended to evaluate solution accuracy with respect to mesh and model parameters.

- Adequate mesh size, run times, and time step require an exponential increase in computational power. This requirement makes only simple projects suitable for desktop or workstation analysis. More complex problems require cluster computational server architecture to adequately perform the CFD analysis in a reasonable time.
- CFD is a powerful tool to assist with pump station design, and it can efficiently assist in the evaluation of potential alternatives.
- Physical modeling is currently still required for confirmation of final configurations to meet the HI acceptance criteria.

Where possible, the recommended approach is to use CFD modeling in conjunction with physical modeling to evaluate pump approach conditions. The CFD model can be used to efficiently evaluate layout alternatives and determine potential remedial measures. The physical model can then be used for refinement of alternatives and final documentation of performance. This conjunctive use of CFD and physical modeling can often lead to the most cost-effective way to develop acceptable pump intake designs.

12.4 CFD SIMULATION OF PUMP INTAKE CHAMBER

12.4.1 Background

One important component of a water and wastewater treatment plant is the pump intake system. The main purpose of the intake system is to reliably deliver an adequate quantity of water of the best quality (Baruth 2005). In many water and wastewater treatment plants, the pump intake chamber is a critical location where hydraulic issues (Claxton 1998), such as vortex and air entrainment, are prone to occur and therefore is of particular interest to CFD modelers as well. A simple but typical pump intake system may consist of either a wet well or channel leading to a pump intake chamber.



Figure 12-7. General geometry setup for the CFD demonstration case study.

| Table 12-1. Information | Related to the CFD Simu | ulation | |
|-------------------------|-------------------------|---|---------------------------------|
| Flow conditions | ltems | | Reference |
| | System | Single bay L-shaped channel leading to a pump intake | |
| | Geometry | Figure 12-7 | Demonstration case study |
| | Dimensions | 500 in.×300 in.×205 in. | |
| | | ID = 13.5 in., OD = 14.0 in. | |
| | | or | |
| | | 12.7 m × 7.62 m × 5.207 m | |
| | | ID = 0.343 m, OD = 0.3556 m | |
| | Model flow rate | 10 million gal./day (i.e., 0.44 m^3/s) | |
| | Reynolds number | 8.4×10^4 (at inlet, based on inlet width) | |
| Model information | Simulation | Method or Governing Equation | Reference |
| | Type | RANS finite volume | |
| | Flow | N-S equation | Section "CFD Model"—See |
| | | | Chapter 4 for model description |
| | Turbulent SGS model | Realizable k - ε model | |
| Software used | Type of software | Name of software | Availability |
| | Meshing tool | Trelis | Commercial |
| | Solver | Fluent | Commercial |
| | Postprocessing tool | Tecplot 360 | Commercial |
| Computational | ltems | Remarks | |
| Information | Computational grid | 1,790,000 elements | |
| | Computing device | Mobile workstation, 8-core CPU, | |
| | equivalent | לם KAIVI 04 קט היי | |
| | Computing time | 7 U | |

PUMPING INTAKES

105

The use of CFD to simulate a pump intake chamber have been well documented in literature such as Li et al. (2006), where different aspects of pump intake design, including evaluation of swirl angles within the pump suction, have been demonstrated. The simple case study in the following section is a hypothetical one, with the intention of demonstrating the application potential of CFD in predicting the presence of vortex near a pump intake.

12.4.2 Objectives

1. Identification of vortex location near a single pump intake shown in Figure 12-7.

12.4.3 Results and Discussion

12.4.3.1 Vortex Prediction and Identification

To demonstrate a case study with vortex formation near a pump intake, a hypothetical L-shaped approach channel was intentionally used, shown in Figure 12-7. The reasoning behind using this geometry is to create an uneven velocity distribution to induce the formation of vortex, for demonstration



Figure 12-8. CFD results of uneven velocity distribution in the approach channel at the bend.

purposes. In this simulation, a pump intake was placed at the end of the L-shaped channel and an intake flow rate of 10 million gal./day (i.e. 0.44 m³/s) was used. More details of the simulation are listed in Table 12-1.

Figure 12-8 showed the streamline of approach velocity in the *XY* plane immediately after the bend. The streamlines converge at the intake and enter the intake at different velocities. The magnitude of velocity can be seen in Figure 12-9 to differ spatially in the horizontal plane. Further examination of the velocity magnitude in the vertical plane in Figure 12-9 shows steep velocity gradients, especially at the far end of the channel, an indicator of potential vortex formation near the intake.

In Figure 12-10, streamlines were drawn at locations where potential vortices may form. The streamlines showed swirling motion starting from the surface and extending into the pump intake column. Three surface vortices (S1, S2, and S3 in Figure 12-10) were identified using the CFD model. The largest vortex, S1, formed at the inside of the bend near the intake, and two other vortices, S2 and S3, were identified at the end of the intake bay.



Figure 12-9. Velocity magnitude plot at different cross sections in the horizontal and vertical plane.



Figure 12-10. Streamline of the three vortices S1, S2, and S3 identified near the pump intake.

12.4.3.2 Other Considerations

The simulation showed the capability of CFD as a tool to identify vortices and problems related to the flow dynamics near a pump intake. Often, vortex formation can be mitigated by using antivortex devices. These structures may be such shapes as cones, walls, or fillets, as mentioned in Choi et al. (2010). CFD can be used to study the effectiveness of the vortex suppression devices based on the placement and dimensions of these devices. Another application of CFD in pump intake simulations is in estimating the swirl angles (Li et al. 2006) within the pump intake. Currently, the role of CFD for pump intake study is still in its infancy, and CFD should not be regarded as a direct substitute for a physical model. However, the use of CFD as a prediction tool is demonstrated in this case study and can be seen as advantageous when used in preliminary pump intake studies.

References

- Baruth, E. E. 2005. Intake facilities: Water treatment plant design. 4th ed. New York: McGraw-Hill.
- Choi, J.-W., Y.-D., Choi, C.-G., Kim, and Y.-H., Lee. 2010. "Flow uniformity in a multiintake pump sump model." J. Mech. Sci. Technol. 24 (7): 1389–1400.
- Claxton, J. 1998. American national standard for pump intake design: ANSI/HI 9.8-1998. Parsippany, NJ: Hydraulic Institute.
- Li, S., J. M., Silva, Y., Lai, L. J., Weber, and V. C., Patel. 2006. "Three-dimensional simulation of flows in practical water-pump intakes." *J. Hydroinf.* 8 (2): 111–124.