It can be shown by the principles of dimensional analysis that such flow conditions at an intake are governed by the following dimensionless parameters:

$$\frac{uD}{\Gamma}, \frac{u}{(gD)^{0.5}}, \frac{D}{S}, \frac{uD}{v}, \text{ and } \frac{u^2D}{\left(\frac{\sigma}{\rho}\right)}$$

Where:

u	=	average axial velocity (e.g., at the bell entrance)
Γ	=	circulation of the flow
D	=	diameter (of the bell entrance, or equivalent diameter for noncircular openings, giving the same area as a circular opening)
S	=	submergence (at the bell entrance)
v	=	kinematic viscosity of the liquid
g	=	acceleration due to gravity
σ	=	surface tension of liquid/air interface
ρ	=	liquid density

The influence of viscous effects is defined by the parameter $\frac{uD}{v} = R$, the Reynolds number, and surface tension effects are indicated by $\frac{u^2D}{\left(\frac{\sigma}{\rho}\right)}$, the Weber number. Based on the available literature, the influence of viscous forces and surface tension on vortexing may be negligible if the values of R and W_e in the model fall above 3×10^4 and 120, respectively (Daggett, L., and Keulegan, G.H., 1974; Jain, A.K. et al., 1978).

With negligible viscous and surface tension effects, dynamic similarity is obtained by equating the parameters $\frac{uD}{\Gamma}$, $\frac{u}{(gD)^{0.5}}$, and $\frac{D}{S}$ in the model and prototype. An undistorted geometrically scaled Froude model satisfies this condition, provided the approach flow pattern in the vicinity of the sump, which governs the circulation, Γ, is properly simulated.

Based on the above similitude considerations and including a safety factor of 2 to ensure minimum scale effects, the model geometric scale shall be chosen so that the model bell entrance Reynolds number and Weber number at the pump rated flow are above 6×10^4 and 240, respectively, for the test conditions based on Froude similitude. No specific geometric scale ratio is recommended, but the resulting dimensionless numbers must meet these minimum values. For practicality in observing flow patterns and obtaining accurate measurements, the model scale shall yield a bay width of at least 300 mm (12 in), a minimum liquid depth of at least 150 mm (6 in), and a pump throat or suction diameter of at least 80 mm (3 in) in the model. If auxiliary service water pumps are not the main focus of the study and pump flows are a small fraction of the main pumps to be studied, scaling requirements do not apply to the auxiliary service water pumps. If auxiliary service water pumps are the main focus, minimum scaling criteria shall apply. In instances where service water flows may influence flow patterns, flow withdrawal shall be included.

In a model of geometric scale Lr, with the model operated based on Froude scaling, the velocity, flow, and time scales are, respectively:

$$V_r = \frac{V_m}{V_p} = L_r^{0.5} \qquad \text{(Eq. 9.8.7.3-3)}$$

$$Q_r = \frac{Q_m}{Q_p} = L_r^2 V_r = L_r^{2.5} \qquad \text{(Eq. 9.8.7.3-4)}$$

$$T_r = \frac{T_m}{T_p} = \frac{L_r}{V_r} = L_r^{0.5} \qquad \text{(Eq. 9.8.7.3-5)}$$

Models of closed-conduit piping systems leading to a pump suction are not operated based on Froude similitude, but need to have a sufficiently high pipe Reynolds number, $R = \frac{uD}{v}$, such that flow patterns are correctly scaled.

Based on available data on the variation of loss coefficients and swirl with Reynolds number, a minimum value of 1×10^5 is recommended for the Reynolds number at the pump suction.

9.8.7.4 Physical model study scope

Selection of the model boundary is extremely important for proper simulation of flow patterns at the pump. As the approach flow nonuniformities contribute significantly to the circulation causing pre-swirl and vortices, a sufficient area of the approach geometry or length of piping has to be modeled, including any channel or piping transitions, bends, bottom slope changes, control gates, expansions, and any significant cross-flow past the intake.

All pertinent sump structures or piping features affecting the flow, such as screens and blockage due to their structural features, trash racks, dividing walls, columns, curtain walls, flow distributors, and piping transitions must be modeled. In modeling screens, the screen head loss in the model shall be the prototype screen head loss times the model scale ratio. This may require modifications to the geometric scaling of the screens. The head loss coefficient is a function of the screen Reynolds number, the porosity (percent open area), and the screen (wire) geometry. Scaling of the prototype screen wire diameter and mesh size to the selected model geometric scale may be impractical and improper due to the resulting low model Reynolds number. In some cases, a model could use the same screen as the prototype to allow equal loss coefficients. Scaling of trash racks bars may also be impractical and lead to insufficient model bar Reynolds number. Fewer bars producing the same porosity and the same flow guidance effect (bar width to bar depth aspect ratio) may be more appropriate.

The inside geometry of the bell (and hub, if modeled) up to the bell throat (section of maximum velocity) shall be scaled. Consideration should be given to modeling the hub if the hub occupies 10% or more of the area. In such cases, the hub shall extend downstream beyond the throat to prevent flow separation in the annular velocity measuring plane. Supports for the hub shall be round rods placed so as to not dissipate any pre-swirl generated by the approach flow or influence the velocity data. Additionally, any vanes in the bell shall not be modeled. The bell should be modeled of clear plastic or smooth fiberglass, the former being preferred for flow visualization. The outside shape of the bell may be approximated except in the case of multistage pumps, in which case the external shape may affect flow patterns approaching the inlet bell. The impeller is not included in physical models, as the objective is to evaluate the effect of the intake design on flow patterns approaching the impeller. A straight pipe equal to the throat diameter or pump suction diameter shall extend at least five diameters downstream from the throat or pump suction.

For free surface intakes, the model shall be deep enough to cover the range of scaled submergence.

9.8.7.5 Instrumentation and measuring techniques

Unless agreed upon circumstances indicate otherwise, the following measurements shall be made. The extent of the measurements is summarized in Section 9.8.7.6, Test plan, below.

Flow: The outflow from each simulated pump shall be measured with flowmeters. If an orifice or venturi meter conforming to American Society of Mechanical Engineers (ASME) standards is used, the meter need not be calibrated. The accuracy of the flow measurement shall be within ±2% of the actual flow rate.

Liquid level: Liquid surface elevations shall be measured using any type of liquid level indicator accurate to at least 3 mm (0.01 ft) in the model.

Free surface vortices: To evaluate the strength of vortices at pump intakes systematically, the vortex strength scale varying from a surface swirl or dimple to an air core vortex, shown in Figure 9.8.7.5a, shall be used. Vortex types are identified in the model by visual observations with the help of dye and artificial debris, and identification of a coherent dye core extending to the pump bell or pump suction flange is important. Vortices are usually unsteady in strength and intermittent in occurrence. Hence, an indication of the persistence of varying vortex strengths (types) shall be obtained through observations made at short intervals in the model (e.g., every 15 seconds) for at least 10 minutes, so that a vortex type versus frequency evaluation can be made and accurate average and maximum vortex types may be determined. Such detailed vortex observations are needed only if coherent dye core (or stronger) vortices exist for any test. Photographic or video documentation of vortices is recommended.

Subsurface vortices: Subsurface vortices usually originate at the sump floor and walls, and may be visible only when dye is injected near the vortex core. The classification of subsurface vortices, given in Figure 9.8.7.5a, shall be used. The possible existence of subsurface vortices must be explored by dye injection at all locations on the wall and floor around the suction bell where a vortex may form, and documentation of persistence shall be made, as for free surface vortices.

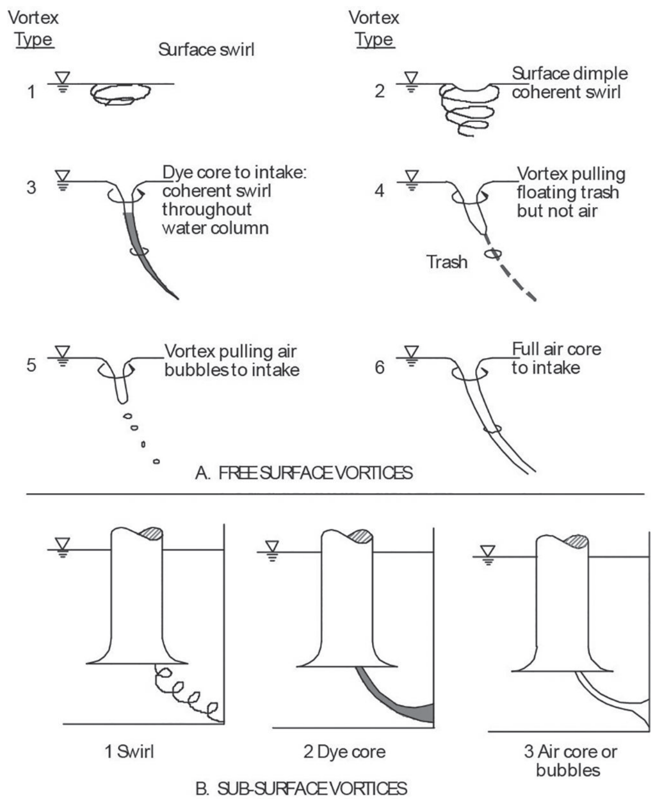

Figure 9.8.7.5a Classification of free surface and subsurface vortices

Swirl in the suction pipe: The intensity of flow rotation shall be measured using a swirl meter, see Figure 9.8.7.5b. The swirl meter shall consist of a straight-vaned propeller with four vanes mounted on a shaft with low-friction bearings. The tip-to-tip vane diameter is 75% of the pipe diameter and the vane length (in the flow direction) is equal to 0.6 pipe diameters. The location of the swirl meter should be about four suction pipe diameters downstream from the bell or pump suction flange to allow for convenient installation of velocity traverse instrumentation. The revolutions per unit time of the swirl meter are used to calculate a swirl angle, θ, which is indicative of the intensity of flow rotation. The swirl meter shall have sufficient support to prevent vibration during testing. The location of the support and area occupied should be such that it does not influence the swirl measurement.

$$\theta = \arctan\left(\frac{\pi d n}{u}\right) \qquad \text{(Eq. 9.8.7.5-1)}$$

Where:

u = average axial velocity at the swirl meter

d = diameter of the pipe at the swirl meter

n = revolutions/second of the swirl meter

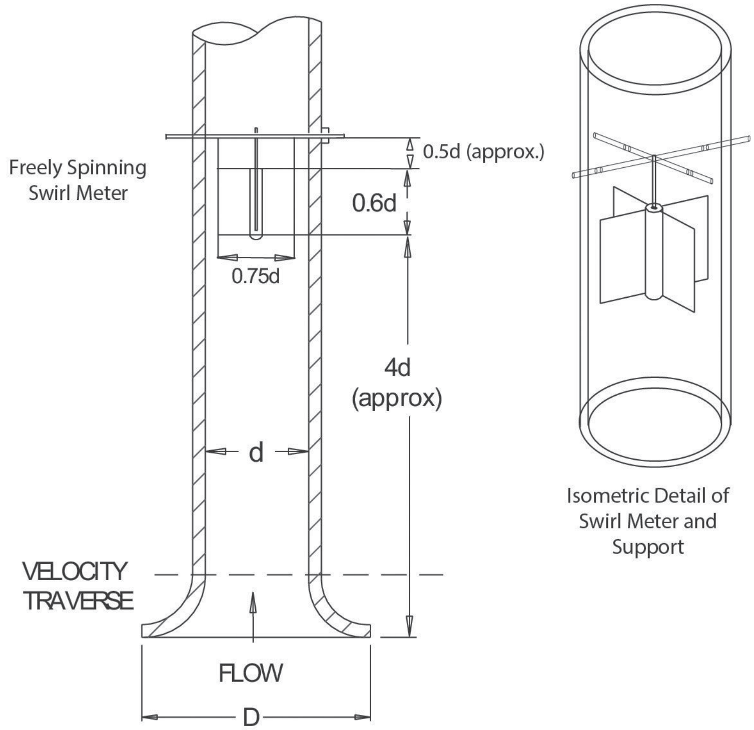

Figure 9.8.7.5b Typical swirl meter

Flow swirl is often unsteady, both in direction of rotation and speed of rotation. Therefore, swirl meter readings shall be obtained continuously; for example, readings during consecutive intervals of 30 seconds, covering a period of at least 10 minutes in the model. Swirl meter rotation direction shall also be noted for each short duration. The maximum short duration swirl angle and an average swirl angle shall be calculated from the swirl meter rotations (see Section 9.8.7.7 Acceptance criteria below). When averaging the swirl meter reading over a timed interval, absolute values should be used, irrespective of rotation direction. Swirl at a dry-pit suction inlet is not of concern if the swirl dissipates before reaching the pump suction flange.

Velocity profiles: Cross-sectional velocity profiles of the approach flow may be obtained using a suitable device at a sufficient number of measuring points to define any practical skewness in the approach flow. The cross-section location shall be selected to be representative of the approaching flow prior to being influenced by the pump, such as at a distance of two intake widths upstream from the pump centerline. Such measurements are in themselves not critical or required, but allow a better understanding of how the approach flow may be contributing to other flow irregularities and what type of remedial devices may be effective.

Velocity measurements to assess the axial distribution and time varying fluctuations shall be obtained at a minimum for the final design at the pump and operating condition indicating the highest swirl and/or vortex activity. Velocity traverses along at least two perpendicular axes at the throat of the model suction bell or the plane of the pump suction in a piping system shall be obtained for the final design. A Pitot-static tube or other suitable instrument capable of determining the axial velocity component with a repeatability of ±2% or better shall be used, with minimal damping. Examples of unnecessary damping effects that may be eliminated are excessive length of instrumentation tubing, extremely soft-sided tubing, or tubing that is too small in diameter. Electronic damping associated with instrumentation shall be minimized.

9.8.7.6 Test plan

Tests should be performed for scenarios that characterize the full range of potential operations, though the potential for model scale effects should be considered if scenarios involving individual pump discharges below the rated capacity are evaluated. Unusual or infrequent operating combinations do not need to be included in the test plan.

Even though vortices are probably most severe at maximum flows and minimum submergence, there are instances where stronger vortices may occur at higher liquid levels and lower flows, perhaps due to less turbulence.

Vortex observations and swirl measurements shall be made for all tests. Axial velocity measurements at the bell throat or suction inlet for each pump in the model are recommended at least for the one test indicating the maximum swirl angle for the final design. Still-photographic documentation of typical tests showing vortexing or other flow problems shall be made.

The initial design shall be tested first to identify any hydraulic problems. If any objectionable flow problems are indicated, modifications to the intake or piping shall be made to obtain satisfactory hydraulic performance. Modifications may be derived using one or two selected test conditions indicating the most objectionable performance.

Practical aspects of installing the modifications should be considered. The performance of the final modified design shall be documented for scenarios that characterize the full range of potential operation. If any of the tests show unfavorable flow conditions, further revisions to the remedial devices shall be made. It is recommended that representative tests of the final design be witnessed by the user, the pump manufacturer, and the station designer.

9.8.7.7 Acceptance criteria

The acceptance criteria for the model test of the final design shall be the following:

- Free surface and subsurface vortices entering the pump must be less severe than vortices with coherent (dye) cores (free surface vortices of Type 3 and subsurface vortices of Type 2 in Figure 9.8.7.5a). Dye core vortices may be acceptable only if they occur for less than 10% of the time or only for infrequent pump operating conditions.

- Swirl angles, both the short-term (30-second model) maximum and the long-term (10-minute model) average indicated by the swirl meter rotation, must be less than 5 degrees. Maximum short-term (30-second model) swirl angles up to 7 degrees may be acceptable, only if they occur no more than 10% of the time or for infrequent pump operating conditions. The swirl meter rotation should be reasonably steady, with no abrupt changes in direction when rotating near the maximum allowable rate (angle).

- Time-averaged velocities at points in the throat of the bell or at the pump suction in a piping system shall be within 10% of the cross-sectional area average velocity. Time-varying fluctuations at a point shall produce a standard deviation of less than 10% of the time averaged signal.

- For the special case of pumps with double suction impellers, the distribution of flow at the pump suction flange shall provide equal flows to each side of the pump within 3% of the total pump flow.

9.8.7.8 Report preparation

The final report of the model study shall include an illustration of the model design, model description, scaling and similitude criteria, instrumentation, test procedure, results (data tabulated and plotted), recommended modifications, and conclusions. The report shall contain photographs of the model showing the initial and final designs, drawings of any recommended modifications, and photographs of relevant flow conditions identified with dye or other tracers. A brief video recording of typical flow problems observed during the tests is recommended.

9.8.8 Use of computational fluid dynamics (CFD)[2]

9.8.8.1 General

Computational Fluid Dynamics (CFD) is an analysis method used in fluid mechanics that deals with numerical solutions of the general flow equations for mass, momentum, and heat transfer. Its origin can be traced to the 1930s, but its rapid development followed the advancements in computing power combined with the needs of the aerospace industry in the 1950s and 1960s. Today, CFD is used within many sciences and industries that involve fluid flow and transport phenomena, including the pump industry.

9.8.8.2 Simulation methods

CFD encompasses a range of methods from very simple meshes and algorithms that can be run on an ordinary personal computer (PC) to the most advanced methods that require high performance computing. Most general-purpose, commercial CFD codes use the Reynolds Averaged Navier–Stokes equations – an approach that offers a reasonable combination of accuracy and computational efforts that is suitable for many industrial applications. For a given code, there are options for establishing boundary conditions, meshing of the flow domain, and turbulence model selection. These factors can influence computational speed and accuracy.

When simulating pump stations (including approach geometry, sump, and pump suction piping) steady-state CFD models are often selected due to their computational efficiency. Steady-state models are time averaged, and thus cannot predict fluctuating phenomena or short-term extreme values such as transient vortex activity and fluctuations in swirl or point velocities. Advanced techniques to simulate time-dependent phenomena are available; however, there are practical limitations in predicting time-dependent and highly curvilinear flow patterns, such as swirls and vortices.

CFD modeling poses many limitations that require skilled and experienced modelers to properly select methods and parameter settings to produce results that correctly represent the behavior of a full-scale prototype, even when using an otherwise well-proven CFD code. Validation of simulations against experimental data is absolutely necessary for a given class of problems. To aid CFD modelers and to eliminate most common pitfalls, some industries have developed best practice guidelines suitable for their specific field.

9.8.8.3 Acceptable uses of CFD modeling in pump suction hydraulics

As of the writing of this standard, there is a lack of generally available correlations of CFD simulations to experimental results for the complex flow patterns near pumps, and there have been no best practice guidelines established for CFD modeling of pump intake and pump suction piping. Until there is satisfactory evidence and appropriate acceptance criteria developed and verified, CFD modeling cannot be used to demonstrate compliance with this standard.

CFD may be useful in determining the general approach flow to a sump and pump suction piping. In particular, the CFD simulation may practically cover a much larger area upstream from the pump than would be possible or practical for a physical model of the scale required to satisfy the minimum Reynolds number given in ANSI/HI 9.8. Therefore, CFD simulations may be used to determine the extent of the physical model and the velocity distribution needed at the physical model boundary. Useful applications of CFD would include determining whether or not physically modeling a single pump bay or single suction pipe would be adequate. CFD simulations may also be used to compare designs, to aid in the initial selection of a design (or design development option) for testing using a physical model, and to better define the range of variables to be tested.

The advances in CFD indicate that further uses of these computational simulations may be possible in the future and the Hydraulic Institute may consider additional applications of CFD in future revisions of this standard.

---

2 The Intake Design and Pump Piping committees met to discuss the application of CFD modeling with regards to the approach flow to a pump. Experts nationally and internationally (academic, HI partners, and nonmembers) were invited to discuss the latest in modeling of intake structures, including current state of the art simulation, postprocessing methods, and case studies. The outcome of the meeting was used by the committee to revise this section (June 27, 2016, Indianapolis, IN).

Appendices

These appendices are not part of this standard, but are presented to help the user in considering factors beyond the standard sump design.

Refer to Section 9.8.2 of the standard which allows for an intake designed to a geometry other than that presented in the standard, such as those contained in these appendices, to be deemed to comply with the standard if the intake is tested by prototype testing or a physical model study performed in accordance with Section 9.8.7, and the test results comply with the acceptance criteria in Section 9.8.7.7.

Requirements for a physical model study are given in Section 9.8.7.

Appendix A

Remedial measures for problem intakes (informative)

Information in this appendix is not part of this standard, but is presented to help the user in considering factors beyond the standard sump design.

Refer to Section 9.8.2 of the standard, which allows for an intake designed to a geometry other than that presented in the standard, such as those contained in these appendices, to be deemed to comply with the standard if the intake is tested by prototype testing or a physical model study performed in accordance with Section 9.8.7, and the test results comply with the acceptance criteria in Section 9.8.7.7.

Requirements for a physical model study are given in Section 9.8.7.

A.1 Introduction

The material presented in Appendix A is provided for the convenience of the intake design engineer in correcting unfavorable hydraulic conditions of existing intakes. None of the remedial measures described herein are part of the standard intake design recommendations provided in Section 9.8. A portion of the material in Appendix A transmits general experience and knowledge gained over many years of improving the hydraulics of intake structures, and such educational material may not include the specific recommendations appropriate for a standard. Corrections described herein have been effective in the past, but may or may not result in a significant improvement in performance characteristics for a given set of site-specific conditions. Other remedial fixes not provided herein may also be effective, and a physical model test is needed to verify whether or not a given remedial design feature results in acceptable flow conditions. This is particularly true because adding a remedial feature to solve one flow problem may have detrimental effects on other flow phenomena of concern.

Appendix A concentrates on rectangular intakes for clear liquids, but the basic principles can be applied to other types of intakes. The material is organized by the general type of hydraulic problem in an upstream to downstream direction, because proper upstream flow conditions minimize downstream remedial changes.

A.2 Approach flow patterns

The characteristics of the flow approaching an intake structure is one of the foremost considerations for the designer. Unfortunately, local ambient flow patterns are often difficult and expensive to characterize. Even if known, conditions are often unique, and frequently complex, so it is difficult to predict the effects of a given set of flow conditions upstream from an intake structure on flow patterns in the immediate vicinity of a pump suction.

When determining direction and distribution of flow at the entrance to a pump intake structure, the following must be considered:

- The orientation of the structure relative to the body of supply liquid
- Whether the structure is recessed from, flush with, or protrudes beyond the boundaries of the body of supply liquid
- Strength of currents in the body of supply liquid perpendicular to the direction of approach to the pumps
- The number of pumps required and their anticipated operating combinations

Velocity profiles entering pump bays can be skewed, regardless of whether or not crosscurrents are present. Several typical approach flow conditions are shown in Figure A.1 for rectangular intake structures withdrawing flow from both moving bodies of liquid and stationary reservoirs. Figure A.2 shows several typical approach flow conditions for different combinations of pumps operating in a single intake structure.

The ideal conditions, and the assumptions on which the geometry and dimensions recommended for rectangular intake structures in this section are based, are that the structure draws flow so that there are negligible ambient currents (cross-flows) in the vicinity of the intake structure that create asymmetrical flow patterns approaching any of the pumps, and the structure is oriented so that the boundary is symmetrical with respect to the centerline of the structure. As a general guide, cross-flow velocities are significant if they exceed 50% of the pump bay entrance velocity. Recommendations (based on a physical model study) for analyzing departures from the ideal condition are given in Section 9.8.7.

A.2.1 Open versus partitioned structures

If multiple pumps are installed in a single intake structure, dividing walls placed between the pumps result in more favorable flow conditions than found in open sumps. Open sumps, with no dividing walls, have been used with varying levels of success, but adverse flow patterns can frequently occur if dividing walls are not used. The trench-type intake structure, described in Sections 9.8.3.4 and 9.8.4.2, is a type of open sump that is an exception. Open sumps are particularly susceptible to cross-currents and nonuniform approach flow patterns. Even if approach flow at the entrance to the structure is uniform, open sumps result in nonuniform flow patterns approaching some of the pumps when operating pumps are arranged asymmetrically with respect to the centerline of the intake structure. This situation can occur when various combinations of pumps are operating or if the intake structure is designed to accommodate additional pumps at some future date. Figure A.3 is an example of flow approaching the pumps in a partitioned structure and an open sump, both operating at partial flow rate.

The example facilities contain four units with two of the four operating. In both structures, flow is withdrawn from a reservoir with no velocity component perpendicular to the longitudinal centerline of the intake structures. In the partitioned structure, flow enters the bay of pump 1 fairly uniformly. It enters the bay containing pump 2 nonuniformly, with a separation area near the right sidewall. However, the length of the bay relative to its width channels the flow and allows it to become more uniform as it approaches the pump. In Figure A.3, example ii, the dashed line at the wing walls shows a rounded entrance configuration that minimizes flow separation near the entrance to the outer pump bays.

In open sumps (Figure A.3, example i), flow may enter the structure uniformly with respect to the centerline of the structure. However, since the location of the two operating pumps is not symmetrical with respect to the centerline of the structure, flow separates from the wall of the structure and approaches pump 2 with a tangential velocity component, greatly increasing the probability of unacceptable levels of pre-swirl.

If all four pumps in the open sump were to operate simultaneously, approach flow would be reasonably uniform, but other adverse phenomena could be present. For example, when two adjacent pumps are operating simultaneously, submerged vortices frequently form, connecting both pumps.

A.3 Controlling cross-flow

If cross-flow is present (i.e., if the pump station is withdrawing flow from the bank of a canal or stream), trash racks with elongated bars can provide some assistance in distribution flow as it enters the pump bay, but if the flow profile is skewed when it enters the trash rack, it will be skewed as it exits. To be effective in guiding flow, trash racks must be placed flush with the upstream edges of the pump bay dividing walls. In this example the trash rack must be vertical or match the incline of the entrance. Both trash racks and dividing walls must be in line with the stream bank contour. Trash racks recessed from the entrance to pump bays, and through-flow traveling screens have a negligible flow-straightening effect (see Figure A.4).

Partially clogged trash racks or screens can create severely skewed flow profiles. If the application is such that screens or trash racks are susceptible to clogging, they must be inspected and cleaned as frequently as necessary to prevent adverse effects on flow patterns.

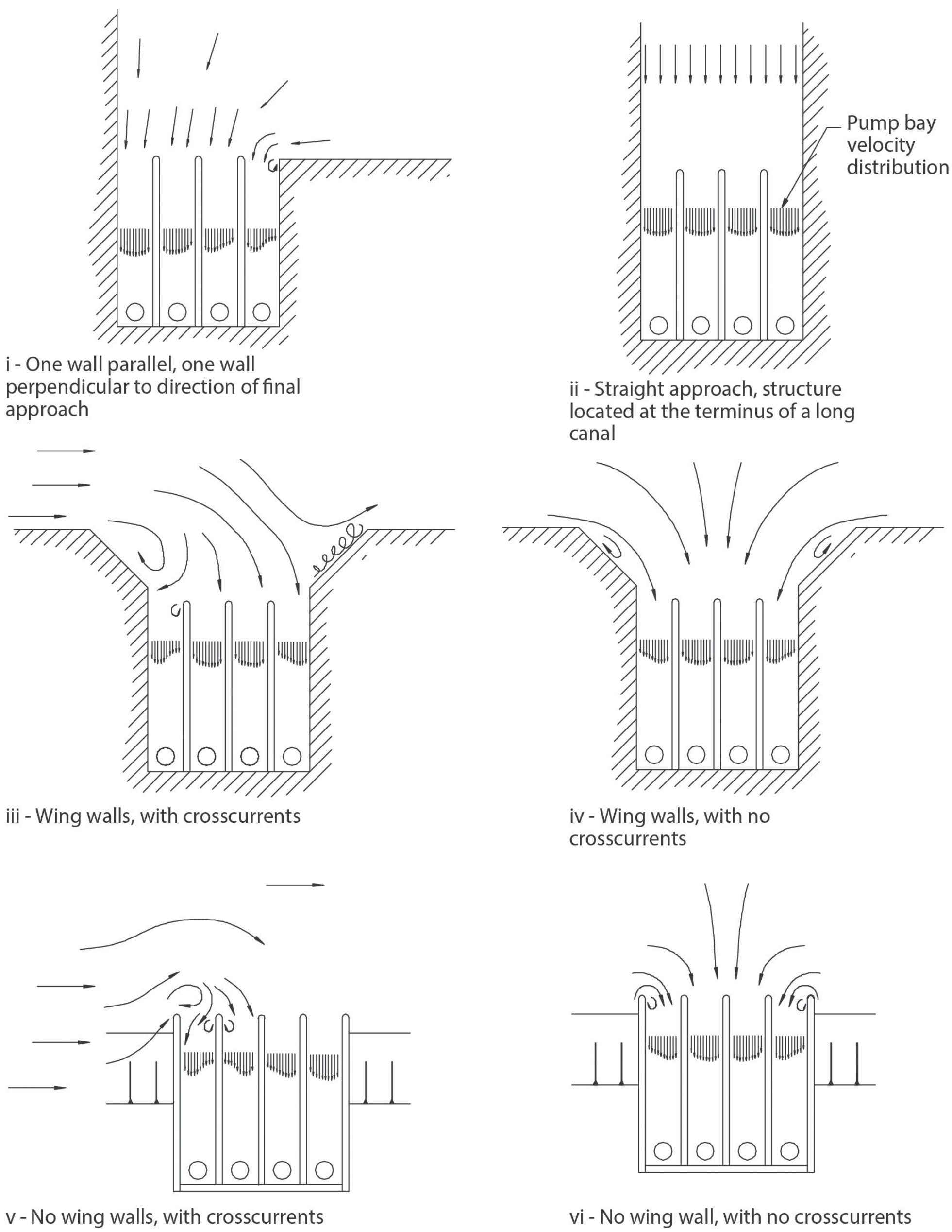

Figure A.1 Examples of approach flow conditions at intake structures and the resulting effect on velocity, all pumps operating

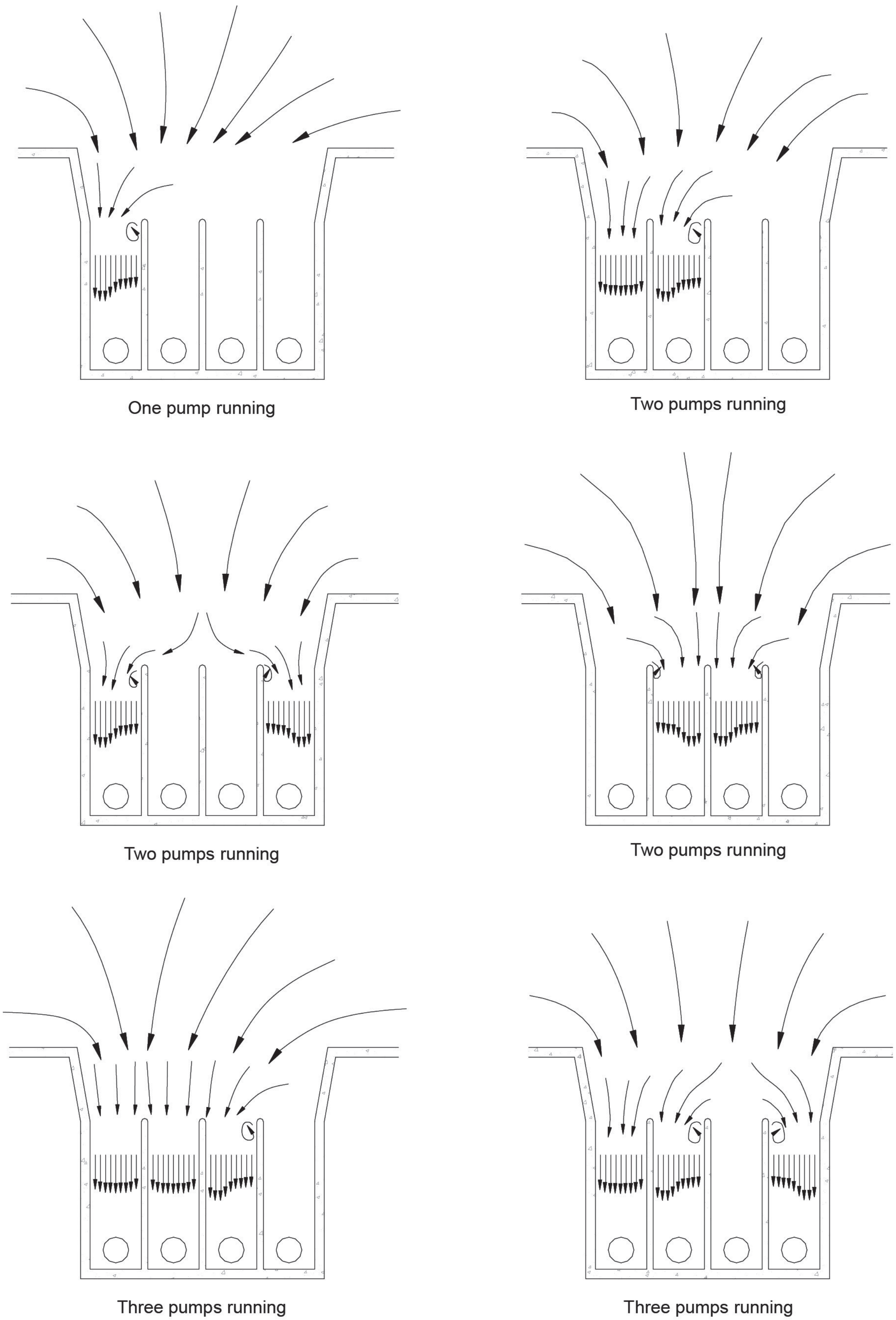

Figure A.2 Examples of pump approach flow patterns for various combinations of operating pumps

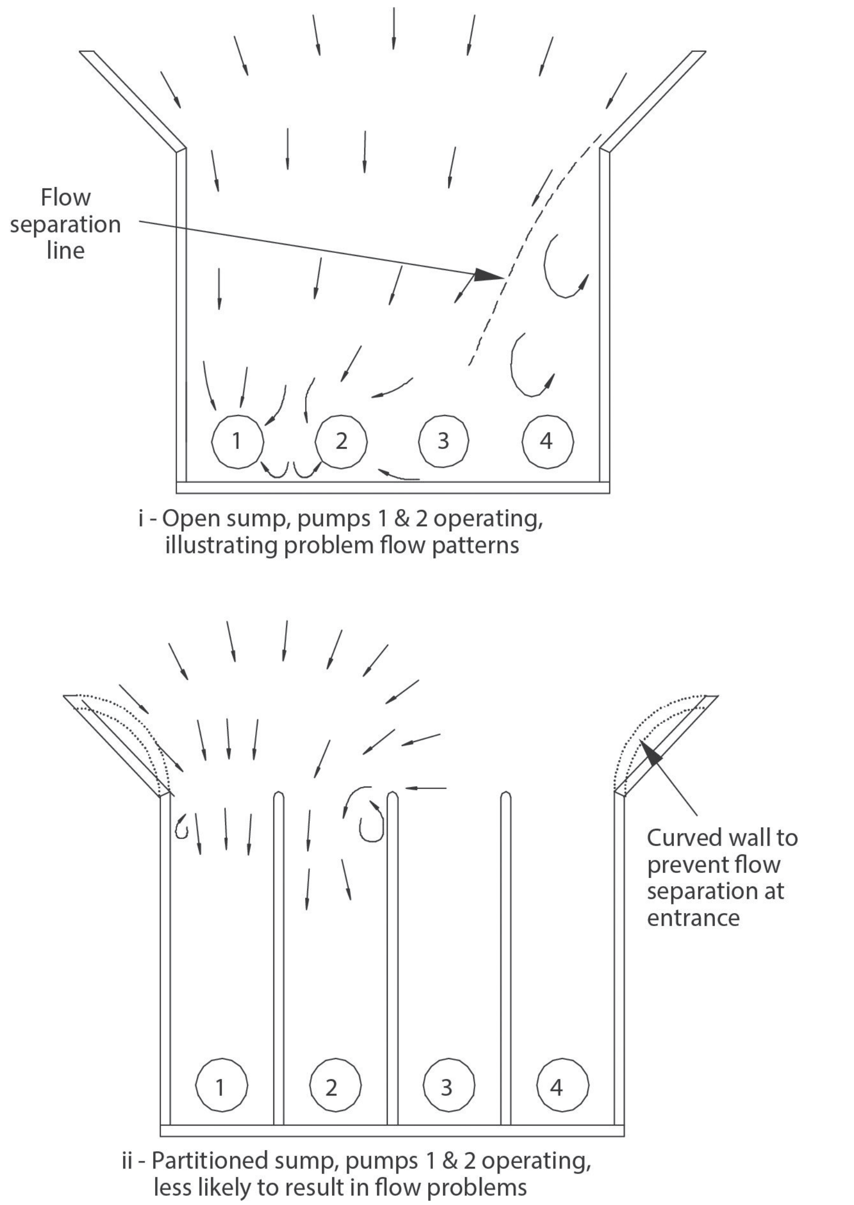

Figure A.3 Comparison of flow patterns in open and partitioned sumps