

Advancements in Hydraulic Modeling of Cooling Water Pump Intakes in Power Plants

Andrew Johansson, Philip Stacy, Dean White and M. Padmanabhan,
Alden Research Laboratory

Abstract

Satisfactory hydraulic performance of pump intakes is very important to avoid flow conditions that would adversely affect the performance of cooling water and service water pumps in power plants. Hydraulic performance is best evaluated using physical hydraulic models. Typically, a hydraulic model is used to identify formation of any objectionable free-surface and subsurface vortices, presence of any high swirl or pre-rotation and/or non-uniform axial velocity distribution at the impeller entrance. Typically a model study would evaluate an initial design based on the acceptance criteria and derive remedial modifications as needed to meet the acceptance criteria.

Over the past several years, modeling of pump intakes has evolved in terms of criteria for model scale selection, instrumentation accuracy and computerized data acquisition techniques, the use of Computational Fluid Dynamics (CFD) to evaluate approach flow patterns and the availability of Hydraulic Institute Standards (HIS) [1] for acceptance criteria to evaluate hydraulic performance. Also, hydraulic model studies conducted by various laboratories over many years have contributed to the advancement of the knowledge about vortex formation, swirl and scale effects in models. To the benefit of all, these advancements have resulted in higher confidence in models with more efficient execution of the model studies at minimal costs.

The purpose of this paper is to present the authors' many years experience and involvement with hydraulic models of both circulating water and cooling tower pump intakes, and the advancements that have been made in the field of physical modeling. Examples are provided to show how hydraulic models may be used to first identify unacceptable flow conditions and then to derive modifications to improve the hydraulic performance for both circulating water and cooling water pump intakes.

1.0 Introduction

The hydraulic performance of cooling water pump intakes for power plants is often evaluated using physical hydraulic models. Typically a model study would include i) observation and documentation of flow patterns approaching the pump bays and within the pump bays, ii) observation and documentation of the location, strength and frequency of any free surface and subsurface vortices present, iii) measurement of swirl within the pump bell or in the suction pipe to evaluate any pre-rotation present in the flow approaching the impeller, and iv) measurement of velocity distribution within the pump bell or in the suction pipe to evaluate the axial velocity profile of the flow approaching the impeller. In addition,

the model study may evaluate and derive solutions to other flow related concerns such as headloss associated with the intake structure and silt deposition within the intake.

In the last ten years, significant advancements have resulted from the experience gained from numerous model studies, availability of guidelines and acceptance criteria from the Hydraulic Institute and other publications, rapid developments in the area of Computational Fluid Dynamics (CFD) and availability of more powerful PCs for data acquisition. The purpose of this paper is to address some of these advancements and discuss various design modifications to meet acceptance criteria as derived from hydraulic model studies, all to the benefit of engineers involved in the design, construction and operation of pump intakes.

2.0 Advances in Hydraulic Modeling

2.1 Model Scale Selection

Pump intake models are designed based on Froude similitude, which requires that the Froude number, $V/(gL)^{0.5}$, is the same in the model and prototype (V is a characteristic velocity such as bell entrance velocity, L is a characteristic length such as pump bell entrance diameter and g is the gravitational constant). Froude similitude is needed to simulate free surface flows and assumes gravity is the predominant force. With Froude scaling, the relationship between gravitational and inertial forces within the flow are scaled correctly. Scaling of other forces such as viscous and surface tension forces may not always be possible, resulting in scale effects. Scale effects can result in under prediction of the formation and strength of vortices. Also, if the model is too small, accurate measurements of water depths, swirl and velocities may be difficult. Hence, the selection of the model scale requires considerations of minimizing scale effects while paying attention to practical and cost considerations which favor a smaller size model.

It has been established by several researchers that the scale effects are negligible, if the Reynolds number (Re), indicating viscous effects, and the Weber number (W), indicating surface tension effects, are above certain critical values [2]. However, these critical values differ from publication to publication in addition to some researchers defining Re and W with different characteristic velocity and lengths. The recent revisions in the Hydraulic Institute Standards ANSI/HI 9.8-1998 (HIS) [1] considers the various critical Re and W in the literature, provides a consistent definition of Re and W and prescribes the critical values. Defining $Re=VD/\nu$ and $W=V^2D/(\sigma/\rho)$, where V is the average velocity at the pump bell entrance (or in the suction pipe just upstream of the pump, if there is no bell), D is the pump bell entrance diameter (or suction pipe diameter) and ν , σ and ρ are the kinematic viscosity, surface tension and density of the liquid pumped, it is suggested that the Re and W in the model should be above 6×10^4 and 240 respectively to avoid any significant scale effects. To prevent the model from being too small, which can affect the accuracy of measurements, HIS recommends that the model scale be chosen

to maintain minimum model values of 12" for bay width, 6" for water depth and 3" for the pump throat (or suction pipe diameter).

2.2 Model Boundaries and Internal Details

2.2.1 Inflow

Flow patterns approaching pump bays can significantly influence formation of free surface and subsurface vortices. To simulate the approach flow patterns correctly, it is necessary for the model to include a sufficient length of approach flow channel, including any bends or transitions that could influence flow patterns at the fore-bay of the pump intake. The model should simulate any cross-flow at the channel entrance (i.e. due to river flow past an intake). In cases where the flow to the fore-bay of the intake is by conduits (pipes, tunnels or non circular conduits), it is necessary to simulate sufficient upstream portion of the conduit including any bends, conduit junctions and valves that are close (within ten hydraulic depths upstream) to the entrance to the pump intake fore-bay. Experience and judgment often plays a role in the selection of the model boundary. A preliminary computer study using Computational Fluid Mechanics (CFD), described later in this paper, can also be used to determine how far upstream the model inflow boundary should be in order to simulate complex approach flow conditions.

Figure 1 shows a pump intake model drawing water from a river through an approach channel with a bend and a transition from trapezoidal to rectangular cross section. The model simulated the river cross flow as well as tidal flow reversals. Shown in Figure 2 is a pump intake fed by tunnels drawing water from a lake. As the tunnels were straight, it was not necessary to simulate in the model more than about five diameters upstream. Figure 2 and 3 show models which included energy dissipation and support structures that influenced the approach flow patterns.

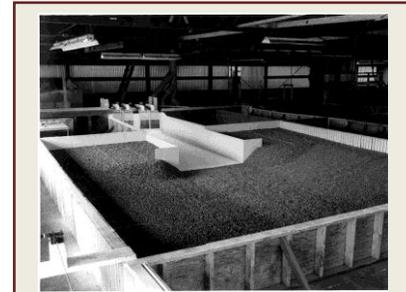


Figure 1



Figure 2



Figure 3



Figure 4



Figure 5



Figure 6

2.2.2 Screens and Racks

Proper simulation of the geometry of screens and racks is very important as these structures can affect the flow patterns approaching the pumps. For example, drum screens and dual flow screens can generate an entirely different flow patterns exiting the screen compared to a through flow traveling screen. The characteristics of the screens and bar racks in the model are designed for an approach velocity based on a selected flow and water depth, usually an average cooling water pump flow and the low water level. Screens are scaled such that the screen head (pressure) loss coefficient for the selected approach velocity is the same in the model and the prototype. Bar racks are scaled to provide the same flow guidance in the model as in the prototype. More details on modeling screens and racks are given below.

2.2.2.1 Traveling Screens

The flow distribution downstream of the screen will be affected by the screen geometry, open area of the screen and the loss coefficient of the screen, K , where K is defined as the head loss across the screen, ΔH , divided by the velocity head of approach flow, $u^2/2g$. In general, K is a function of the screen Reynolds number, Re , for Re less than 500 [3, 4, and 5]. The screen Reynolds number is defined as $ud / [\nu (1-S)]$, where u is the approach flow velocity, d the screen wire diameter, S the solidity ratio (fraction of closed area), and ν the kinematic viscosity of water.

The screen flow paths depend on the type of screen arrangement. Figure 4 shows a dual flow screen arrangement in a pump intake model while Figure 5 shows flow patterns resulting from dual flow screens indicating high velocity jet, flow separations and eddies. Figure 6 shows a drum screen arrangement in a model. As is common, the traveling screens are simulated as fixed (not movable) in the model, as the effect of screen movement on the approach flow patterns to the pump farther downstream would be negligible. The model screen wire size and mesh are selected so that the percent open area of the screen and the screen loss coefficients at the scaled flows in the model are approximately the same as those in the

prototype under corresponding conditions. This often means use of screens in the model with wire sizes as large as the prototype screen.

2.2.2.2 Bar Racks

Model bar racks are usually fabricated to simulate approximately the same percent open area and bar thickness to depth ratio as in the prototype, thus giving the same guidance to the flows. The actual bar thickness need not be scaled geometrically, but bars of suitable thickness can be used to give a bar screen Reynolds number of above 100 in the model so that any scale effects in the model are negligible. For example in one model study, the bar racks with 1/16" (1.58 mm) thick and 1/2" (12.7 mm) deep bars at 11/16" (17.5 mm) center to center spacing were used in the model to represent 3/8" thick and 3.0" deep bars at 4-5/16" center to center spacing in the prototype.

2.2.3 Outflow

The outflow boundary (downstream extent of the model) depends upon the type of pumps and suction pipe arrangement. In the case of vertical/axial flow centrifugal pumps, the pump bells form the outflow boundary and the internal shape of each pump bell is modeled up to the throat of the pump(throat of the first stage bell in case of multiple stages), including any hub located between the bell entrance and throat. The throat diameter is then continued in the model with a straight pipe to accommodate a swirl meter before connecting to the return piping in the model. The outside shape of the bell is also modeled. In the case of multiple stage pumps, the outside shapes of all stages are often modeled in order to simulate the flow path with the obstructed area in the vicinity of the bells, as in can pumps or pumps within vertical risers. For horizontal pumps, the suction pipe leading to the pump entrance, including any bends, expansions, contractions, open valve disc etc, need to be modeled. For Formed Suction Intakes (FSI) the inner shape of the FSI up to the pump impeller is modeled. Neither the impellers nor any pump rotating parts are modeled, being unnecessary for evaluating the intake hydraulics.

2.3 Simulation of Silt

Modeling silt deposition (sedimentation) is dependent on the problem to be studied and is not quantitative as is the scaling of other flow phenomena such as vortex formation. Deposition within pump bays occurs when suspended silt settles in low velocity regions. Movement of deposited silt along the bottom is influenced by approach flow conditions in the fore-bay area and the turbulence generated within the intake bays.

Scaling the incipient motion of material along the bottom is usually based on simulation of bottom shear stress (τ) expressed as a non-dimensional Shield's parameter [6], $F_s = \tau_o / [\gamma (S_g - 1) d]$, where S_g and d are particle specific gravity and size. Shield's diagram correlates Shield parameter (F_s) to the Reynolds number ($R_s = u^* d / \nu$) based on shear velocity $u^* = (\tau_o / \rho)$ and particle diameter d . Note that ρ is the

water (liquid) density. However, most practical applications involving turbulent free surface flows within the pump intakes, the Shield parameter may be assumed independent of R_s . Hence, proper simulation of bed load transport requires that the Shield parameter is about the same in the model as in the prototype. This can be accomplished in a model based on Froude similitude by proper selection of model silt material (density) and sizes. Alternately, knowing the published values of the critical or permissible flow velocity [7], suitable model sediment can be selected to allow the scaling of the permissible velocity to the flow velocity scale ratio. However, values of the permissible maximum velocity vary widely depending on the average particle size and cohesion of the sediment or silt. Characterizing the field sediment is often a difficult problem and, thus, use of published permissible velocities may not be always feasible. Hence, the method based on Shields parameter is more practical. Settling of suspended particles will depend upon the local velocity and turbulence intensity within the intake bay and the settling velocity of the particles. Scaling of the particle settling trajectory is important. For example, the time, t , available for a particle to settle from a given height, H , due to its settling velocity, w , in a distance along the intake, L , while moving at a flow velocity, V , is given by

$$t = \frac{H}{w} = \frac{L}{V} \quad (1)$$

Using only the right hand equality, and the subscript, r , to denote the ratio of model to prototype quantities, we obtain, after cross multiplication

$$\left(\frac{w}{V}\right)_r = \left(\frac{H}{L}\right)_r \quad (2)$$

For an undistorted model, $(H/L)_r = 1$, so

$$w_r = V_r \quad (3)$$

Equation 3 indicates that the material settling velocity should scale as the flow velocity. Alternately, for given prototype and model materials, the ratio of their settling velocities dictates the ratio of the flow velocity to be used in the model, which may differ from the Froude scale velocities used for simulation of free surface phenomena as described in Section 2.1. Such distortion of flow velocity should be minimal as it may result in incorrect simulation of flow patterns.

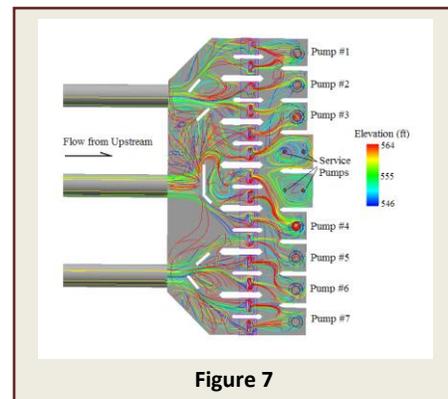
Deposited silt may also be transported along the intake floor. In this case, the material properties and sizes of the model silt selected based on the settling velocity concept as described above, also need to satisfy at least approximately the required equality of Shield's parameter. Due to practical limitations, both experience and engineering judgment are needed to select an appropriate model material to simulate silt deposition and transport.

An appropriate material for model silt can usually be found from a survey of commonly used and commercially available materials (such as plastic, nylon, coal, lime, fine silt, etc.). But, if plastic or similar materials are not commercially available in the required fine particle sizes, it may be necessary to pulverize larger size particles and sieve out the larger particles, to produce the desired fine particles. To determine quantitative results (i.e. rates of deposition or quantities of deposits) from silt tests in a pump intake model, it is necessary to continuously inject silt for long periods while measuring deposition amounts at various locations as a function of time. This effort is often prohibitively expensive; hence, silt tests are often limited to obtaining qualitative data such as locating areas of significant deposits within the intake and forebay, without any efforts to quantify the rate or amount of deposition. The test duration and concentration of the sediment in the model is often determined by trial to fulfill the above test objective.

2.5 Use of Computational Fluid Dynamics Models

Computational Fluid Dynamics (CFD) is an excellent tool for solving flow problems and several CFD codes are commercially available. However, use of CFD for evaluating formation of vortices within a pump bay is not yet acceptable due to difficulties in predicting the strength and persistence of free and sub-surface vortices and resulting swirl or pre-rotation. CFD models can not identify whether a vortex is air entraining nor can it predict the unsteady and unstable nature of vortices which almost always form intermittently and move around within the bay.

In spite of the above limitations, CFD models can indeed complement physical models and can help to reduce the area to be modeled as well as the number of test runs, resulting in cost savings. For example, consider a pump intake approach channel with several bends and control structures upstream in close proximity to the pump intake. A CFD model can be used to analyze flow patterns at the entrance to the fore-bay of the pump intake which can then be reproduced in the physical model by artificial means without modeling the complex geometry of the approach channel and control structures. For pump intakes involving multiple pumps, operating flows, and water levels, a CFD model can be used to identify operating conditions that are likely to result in the worst vortex severities and swirl within the pump bays for testing in the physical model. Figure 7 shows CFD predicted flow patterns for a multiple pump bay pump intake.



Remedial modifications to the pump intake in order to satisfy HIS or other acceptance criteria are commonly derived by testing with the physical model. However, often several combinations of modifications involving splitters, fillets, curtain walls, flow distributors and guide vanes may have to be

evaluated. In this case, it may be advantageous to use a CFD model for an initial evaluation of various possible combinations so that only the ones that are most promising need to be tested with the physical model.

2.6 Advancements in Instrumentation and Data Acquisition

2.6.1 Data Acquisition

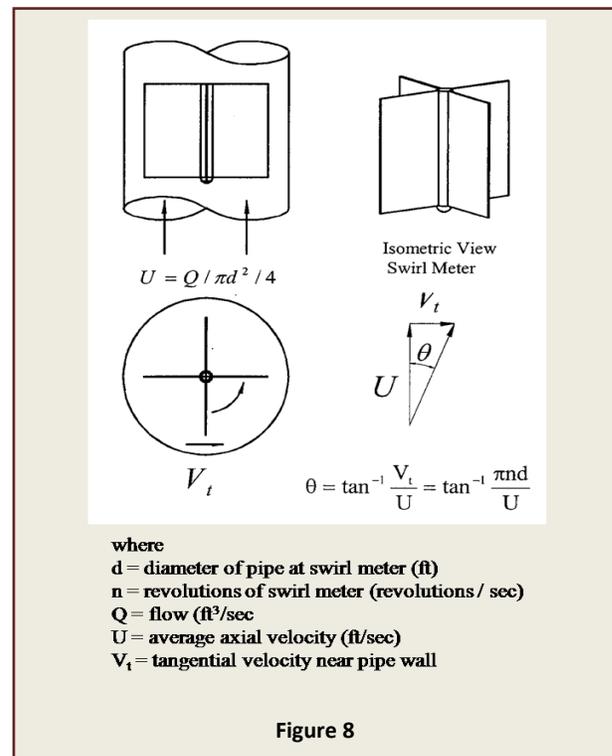
Hydraulic modeling involves measurements of the parameters of water surface elevations, pump flows, flow patterns, swirl angle, velocity profiles, surface and submerged vortexing, sediment behavior, etc. Many of these are quantitative measurements, and some, like sedimentation, can only be qualitatively assessed using a physical model. A practical goal in the development of physical modeling is to reduce the number of qualitative parameters as quantitative measurements lend themselves to more straightforward assessment.

Of particular interest is the use of the personal computer for data acquisition. Virtually any instrument-based recording can be performed using a personal computer including the continuous monitoring and averaging of flows, water levels and head losses. One benefit of computer based acquisition is it allows simultaneous recording of operating parameters, which frees the test engineer to concentrate on recording the visually based assessments such as flow patterns, and free and subsurface vortex activity.

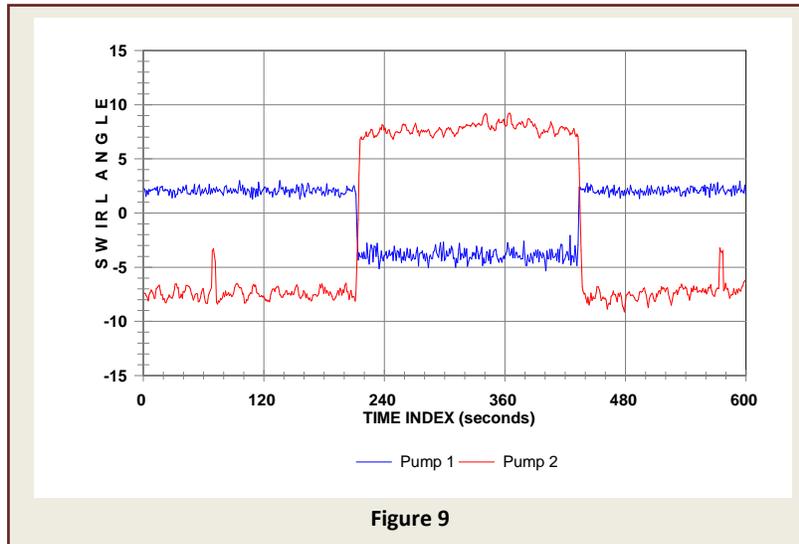
2.6.2 Measurement of Swirl

Swirl meters are used to determine the swirl or pre-rotation at a location in the pump bell or suction pipe just upstream of the pump entrance. As illustrated in Figure 8, a swirl meter consists of four perpendicular vanes attached to a shaft which can freely rotate. The number of rotations of the swirl meter is used to calculate an average swirl angle which is an indication of the swirl intensity. The location of the swirl meter within the simulated pump column is usually four diameters downstream of the bell entrance, as recommended by HIS [1].

With advancements in swirl meter design, reliable sensing devices and computer data acquisition systems, it is possible to continuously track the swirl meter rotations (even fraction of rotations) and the direction of rotation over any desired test duration.



Continuous swirl data allows determination of instantaneous maximum swirl angle as well as average swirl angles over any desired short (few seconds) and/or long (few minutes) duration. An example of swirl angle versus time data collected in a pump intake model study is shown in Figure 9. Such detailed



information is very valuable, as it allows evaluation of the persistence of swirl in addition to the magnitude of swirl (instantaneous, short and long duration averages). In some cases, intermittent and direction changing swirl may be more harmful to the pump performance than a steady one-directional swirl of similar magnitude.

A modern personal computer system equipped with a simple interface is used to record the rotation rate and direction of these swirls meters. In addition, several meters can be recorded simultaneously, which allows the dynamic interaction between adjacent pumps to be investigated, for example, to document cases where swirl intensity, or direction, changes between operating adjacent pumps.

2.6.3 Digital Photography and Video Documentation

As part of most model studies, hydraulic conditions (flow patterns, vortex activity, silt deposition, etc.) within pump intake structures are often documented using still photography and video. With the advent of higher quality and low cost equipment the digital format is now the preferred choice. The digital format has multiple advantages including ease of storage and maintained quality over time. Digital photos and videos save time as they can be transmitted by e-mail to multiple parties involved and the analysis of data can be discussed through teleconferences.

2.7 Hydraulic Institute Standards Acceptance Criteria

Hydraulic Institute Standards, ANSI/HI 9.8-1998 (HIS) [1] provide acceptance criteria for the hydraulic performance of a pump intake in terms of vortex severities, swirl or pre-rotation and the distribution and fluctuations of velocity at the pump impeller entrance. Prior to the availability of HIS acceptance

criteria, there was no single commonly accepted reference defining the acceptance criteria. Some pump manufacturers had their own standards, but there was often considerable variation between manufacturers. As pump manufacturers, hydraulic engineers and modeling experts were involved in the preparation and revisions of HIS, it is advantageous to recognize HIS as the basis for acceptance criteria. However, it may be necessary to use some engineering judgment before suggesting costly remedial modifications, in order to meet HIS or other standard acceptance criteria. For example, in the case of a wastewater pumping station, using engineering judgment and input from the pump manufacturer, one may decide to relax the HIS acceptance criteria to avoid costly modifications even though for one or two less frequent operating conditions the model study showed infrequent and weak coherent core vortices and/or swirl angles beyond the HIS specified limits. In the same situation, it may be prudent to install remedial modifications to strictly conform to HIS standards for a service water pump intake at a nuclear power station.

3.0 Remedial Modifications for Adverse Flow Conditions



Figure 10



Figure 11

3.1 Skewed Approach Flow Patterns

For pump intakes with flow entering the structure through pipes, skewed approach flow patterns in the fore-bay area of the pump intake are usually a result of the concentrated inflow jets. Large eddies with back flow in the forebay can degrade the performance of trash racks and screens. Examples of modifications to improve forebay flow patterns are shown in Figure 10 and 11. Figure 10 shows energy dissipater baffles which were installed to dissipate the kinetic energy from the concentrated inflows. A column type flow distributor upstream of (or at the entrance to) the bays are shown in Figure 11, will provide a more uniform distribution. Flow distributors are head loss devices and are generally only 25 to 40% open to provide sufficient redistribution of the flow. Flow guidance vanes (baffles) are sometimes used, see Figure 2, but may not be effective for all operating conditions, especially when only few pumps operating.

For free-surface inflows such as flow drawn from a river or for cooling tower intakes located on one side of the tower, the flow turning at an angle to enter the pump intake structure could result in significant flow separations and consequent highly skewed flow patterns in the pump intake fore-bay area. For cooling tower intakes with a sloping floor and a widthwise expansion approaching the pump bays, flow separations both along the floor and along the sides can induce significantly non-uniform approach flow to the pump bays resulting in strong free and subsurface

vortices. Typical flow straightening devices include column type flow distributors similar to that used for pipe inflows. Curtain walls at the bay entrances are effective in generating a self-correcting accelerated flow (under flow), which helps isolate the flow within the pump bays from the upstream conditions.

3.2 Objectionable Vortices

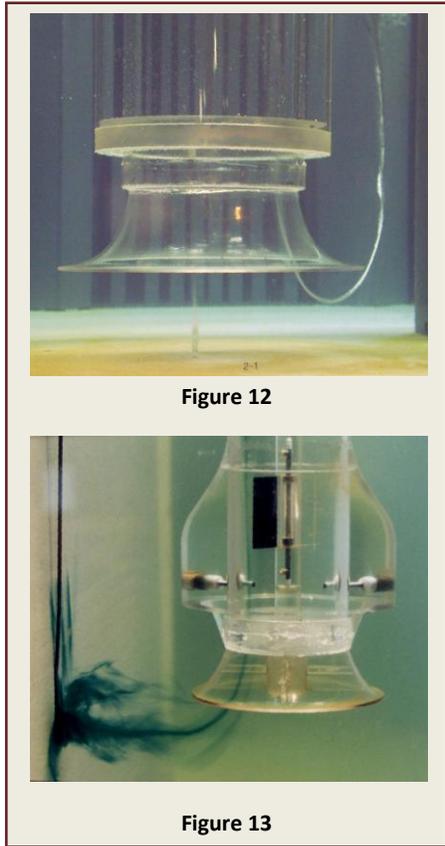


Figure 12

Figure 13

Free surface vortices are induced by insufficient minimum submergence to the bell as well as skewed approach flow patterns within the pump bays. The first step to avoid free surface vortices is to provide a sufficient minimum submergence for vortex-free operation as recommended by the pump manufacturer or using recommended values from HIS [1] or other publications such as IAHR monograph on Swirling Flows [8]. Recommended submergences from the literature assume a reasonably uniform approach flow. When a reasonably uniform approach flow at the pump bay entrances can not be obtained due to practical limitations, strong coherent core air drawing free surface vortices may exist (see Figure 12) even if the submergence requirements are met. A common method of eliminating objectionable free surface vortices is to install a curtain wall (see Figure 14) across the pump bay upstream of the pump, which will force the flow to go under the wall. The curtain wall helps to avoid circulation at or near the water surface and distributes the flow more uniformly under the wall by inducing a back pressure. Horizontal gratings placed a few inches below the water surface are also effective, but may not be practical in wide CW pump bays or if there are debris retention concerns.

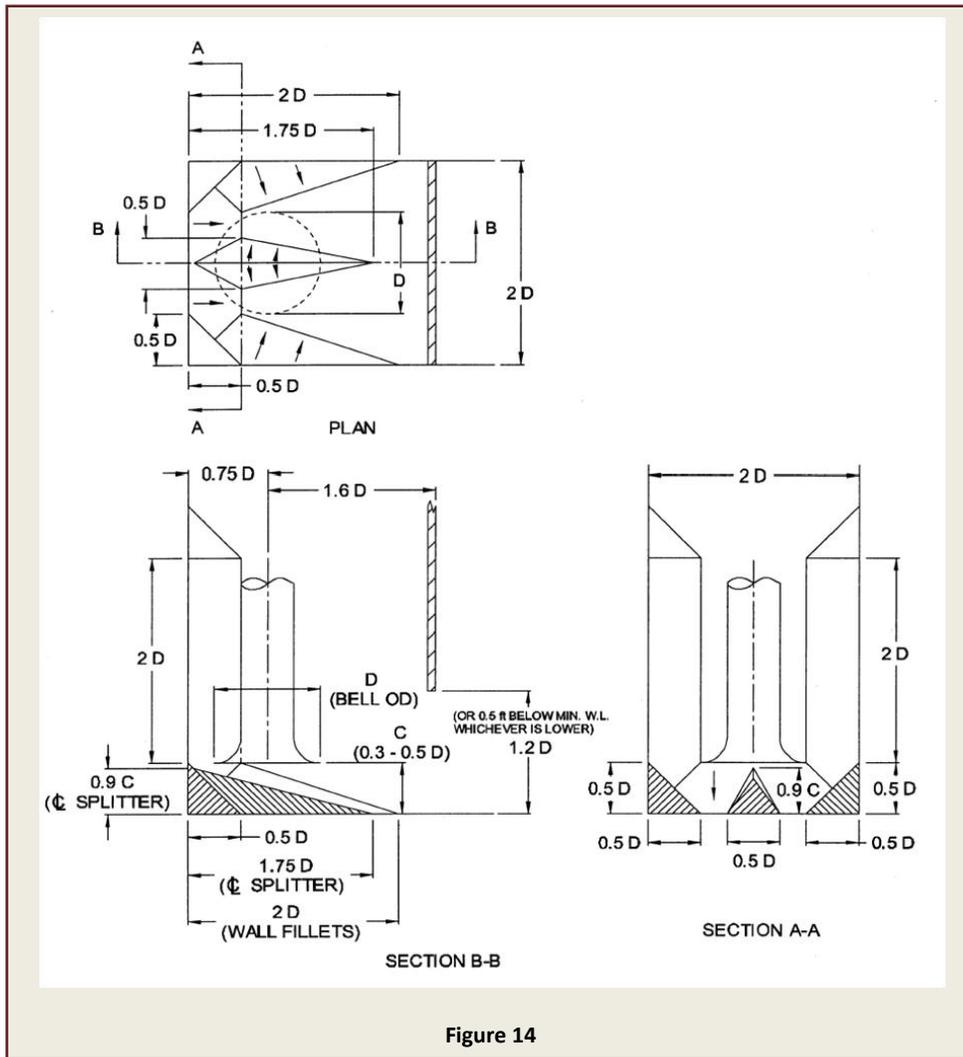
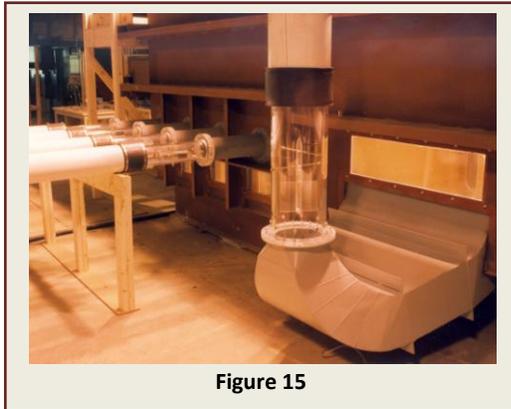


Figure 14

Subsurface vortices are induced by flow separations in the vicinity of the pump bell entrance due to too small or too large floor and wall clearances and/or skewed approach flows within the pump bays. Required wall and floor clearances are recommended by the pump manufacturer and are also available from HIS. However, with appropriate clearances, strong subsurface vortices can still exist due to non-uniform approach flow velocity distribution. Strong subsurface vortices can also form for certain pump operating combinations. Approach flow patterns can be sensitive to operating combination and modifications in the fore-bay area to improve flow patterns may not be sufficient for all cases. Subsurface vortices can also exist between adjacent pumps, if there are no walls separating them. Shown in Figure 13 are examples of subsurface vortices emanating from floor, wall and between adjacent pumps obtained from various power plant pump intake model studies. Floor and wall subsurface vortices can be eliminated by installing floor and wall splitters and fillets which guide the flow into the bell. A typical conceptual design to be further optimized with the hydraulic model study is

shown in Figure 14. Instead of floor splitters, floor cones are sometimes used. Vortices between adjacent pumps in a common bay can be eliminated by grating partitions that dissipate the rotational energy. Unfortunately, standard designs of these devices (such as splitters and fillets, grating partitions etc.) which would provide satisfactory operation for all pump intakes and operating cases do not exist. A most effective design can only be optimized through a physical model study.



Formed Suction Intakes (FSI) are sometimes used in cooling water pump intakes. An FSI has an advantage in that objectionable subsurface vortices are very unlikely due to the accelerating flow within the FSI which induces a more or less uniform flow at the pump entrance. Free surface vortices can be present due to insufficient submergences and/or skewed approach flows at the FSI entrance. Figure 15 shows a FSI studied in a hydraulic model to evaluate free surface vortices.

3.3 Objectionable Pre-rotation or Swirl

Generally, significant pre-rotation or swirl can result from strong subsurface vortices, and to a lesser extent, from free surface vortices. As described earlier, vortex suppression devices to eliminate or reduce vortices will also be effective in reducing or eliminating swirl. In addition, installation of vanes or cruciform's within the bell may further reduce swirl. Diffusers and/or elbows in the suction piping to the pump can augment any swirl present. Guide vanes may be installed within the suction piping to reduce swirl before the flow enters the pump.

3.4 Excessive Silt/debris Deposition

In the fore-bay area of a pump intake, silt generally deposits at locations where the flow separates and a low velocity or dead area exists. For intakes with the inflow through pipes or tunnels, it is possible to use diffusers or flow distributors to spread the concentrated inflow and generate a more uniform flow downstream and towards the pump bays. Guide walls are also often used to redirect the flow from low velocity or stagnant areas. Silt deposition within the pump bays can be prevented by using curtain walls which will generate higher velocities under the wall and sweep away deposited silt. One particular pump intake with significant silt deposits within the pump bays was tested with a hydraulic model using simulated silt. A series of four curtain walls along the bay length (with 3 to 4 ft bottom clearances) was derived using the model. The Curtain walls prevented silt deposition without adversely affecting pump performance.

4.0 Summary

Experience and knowledge gained from model studies over the past several years, availability of guidelines from the Hydraulic Institute and other publications, rapid developments in the area of Computational Fluid Dynamics (CFD) and availability of more powerful PCs for data acquisition, all have contributed to more efficient and reliable hydraulic model studies of cooling water intakes for power plants. Summarized below are some of these advancements described in this paper:

1. Selection of the model scale requires considerations of minimizing scale effects while paying attention to practical and cost considerations. Scale effects will be negligible, if the Reynolds number (Re) indicating viscous effects and the Weber number (W) indicating surface tension effects are above certain critical values. Recent revisions in the Hydraulic Institute Standards, ANSI/HI 9.8-1998 (HIS) consider the various critical Re and W in the literature and suggests that Re and W in the model should be above 6×10^4 and 240 respectively to avoid any significant scale effects. Further, HIS recommends minimum values of 12" for bay width, 6" for water depth and 3" for the pump throat diameter at entrance (or suction pipe diameter) in the model.
2. To simulate the approach flow patterns correctly, the upstream boundary of the model should include a sufficient length of approach flow channel or conduit that feeds the fore-bay region of the pump intake, including any bends, conduit junctions and butterfly valves that are close (e.g., within ten hydraulic depths upstream) to the entrance to the pump intake forebay. Internal structures such as racks, screens and any large supports that could affect the flow patterns should be modeled.
3. Currently, Computational Fluid Dynamics (CFD) is not a reliable means to predict the strength and persistence of free and sub-surface vortices and resulting swirl or pre-rotation. In spite of this limitation, CFD models can indeed complement physical models and help to reduce the modeled area as well as the number of test runs, resulting in saving of time and cost. Also, a CFD model can be used for initial evaluation of flow patterns for several possible combinations of remedial modifications with only the most promising tested in the physical model.
4. Swirl meters are used to determine the swirl or pre-rotation at a location in the pump bell or suction pipe. With the advancements in the swirl meter design, reliable sensing devices and computer data acquisition system, it is possible to continuously record the swirl meter rotations (even fraction of rotations) and the direction of rotation over any desired test duration. Continuous swirl data allows determination of instantaneous maximum swirl angle as well as average swirl angles over any desired duration.
5. Hydraulic Institute Standards, ANSI/HI 9.8-1998 (HIS) provides acceptance criteria for the hydraulic performance of a pump intake in terms of vortex severities, swirl or pre-rotation and the distribution and fluctuations of velocity at the pump impeller entrance. To avoid costly design changes, some engineering judgment is needed to decide whether strict adherence to HIS standards is necessary for infrequent operating cases.

6. Silt deposition and transport can be evaluated qualitatively in a pump intake model. Based on past experience and theoretical considerations, selection of appropriate model silt material is now possible to simulate both settling and bed load transport. Silt deposition can be reduced by relatively simple modifications derived with a hydraulic model study. Examples include installation of curtain walls within the pump bays and deflector vanes in the forebay area.
7. With the availability of digital cameras for video and photography, video clips of flow patterns including formation of vortices can be instantly transmitted via the internet to interested parties to seek their opinion on the need for remedial modifications.

5.0 REFERENCES

1. Hydraulic Institute Standards, American National Standard for Pump Intake Design, ANSI/HI 9.8, 1998.
2. Padmanabhan, M., and Hecker, G.E., "Scale Effects in Pump Sump Models", ASCE Journal of Hydraulic Engineering, Vol. 110, No. 11, November, 1984.
3. Papworth, M., "The Effect of Screens on Flow Characteristics," BHRA Fluid Engineering, Report No. TN1198, November, 1972.
4. Baines, W.D. and Peterson, E.G., "An Investigation of Flow Through Screens," Transactions of the ASME, July, 1951.
5. Padmanaban, M., and Vigander, S., "Pressure Drop due to Flow Through Fine Mesh Screens", ASCE Journal of Hydraulics Division, August 1978.
6. Open Channel Hydraulics, R.H. French, McGraw Hill, 1985
7. Open Channel Hydraulics, V.T.Chow, McGraw Hill, 1959.
8. Swirling Flow Problems at Intakes, Jost Knauss, IAHR Hydraulic Structures Design Manual, A.A.Balkema/Rotterdam, 1987.