Suppression of the secondary flow in a suction channel of a large centrifugal pump

D Torii\textsuperscript{1}, T Nagahara\textsuperscript{2} and T Okihara\textsuperscript{2}

\textsuperscript{1}Pump and Fan Division, Machinery Systems Division, Infrastructure Systems Company, Hitachi, Ltd.
\textsuperscript{2}Turbomachinery R&D Center, Infrastructure Systems Company, Hitachi, Ltd.
603 Kandatsu-machi, Tsuchiura-shi, Ibaraki 300-0013, Japan

E-mail: daichi.torii.yj@hitachi.com

Abstract. The suction channel configuration of a large centrifugal pump with a 90-degree bend was studied in detail to suppress the secondary flow at the impeller inlet for improving suction performance. Design of experiments (DOE) and computational fluid dynamics (CFD) were used to evaluate the sensitivity of several primary design parameters of the suction channel. A DOE is a powerful tool to clarify the sensitivity of objective functions to design parameters with a minimum of trials. An L\textsuperscript{9} orthogonal array was adopted in this study and nine suction channels were designed, through which the flow was predicted by steady state calculation. The results indicate that a smaller bend radius with a longer straight nozzle, distributed between the bend and the impeller, suppresses the secondary flow at the impeller inlet. An optimum ratio of the cross sectional areas at the bend inlet and outlet was also confirmed in relationship to the contraction rate of the downstream straight nozzle. These findings were obtained by CFD and verified by experiments. The results will aid the design of large centrifugal pumps with better suction performance and higher reliability.

1. Introduction

Large high-pressure centrifugal pumps used for water supply systems must be highly reliable to ensure long-life stable operation. Cavitation at the inlet of the impellers is one of the primary factors that decrease the reliability of those pumps, resulting in erosion on the blades and bearing damage due to large amplitude vibration. Thus, it is desirable to design such pumps to be free from cavitation as much as possible, even when operated at low net positive suction head (NPSH). The direct method to improve pump suction performance is to modify the impeller design. However, optimizing the shape of the suction channel is another effective approach, which is employed and studied in this paper.

Suction channels to be studied in this paper consist of a 90-degree bend and a downstream straight nozzle as shown in figure 1. Fluid flow through a bend generally produces a pair of spirals, whose projection on a cross sectional plane normal to the main flow forms a symmetric pair of vortices. This secondary flow is attributed to two factors. The first is a pressure gradient that forms from the inner to outer portions of the channel, directed along the radius of bend curvature. This gradient forms to balance the centrifugal force acting on the fluid due to the curved motion of the main flow. The second factor is the presence of a boundary layer where the centrifugal force is weaker and fails to balance the pressure gradient. This results in a flow along the wall from the outside to the inside of the channel along the radius of the bend curvature. This secondary flow disturbs the uniformity of the flow at the...
impeller inlet and creates variation in the circumferential component of the flow velocity vector. This variation causes changes in the flow angle relative to the inlet angles of the blades. When the angle of the flow is not identical to the blade inlet angle, the fluid flows with increasing relative velocity. As a result the local pressure drops and cavitation occurs when the suction pressure decreases below a certain level.

Several reports have been published on the influence of the suction channel on flow uniformity at the inlet of an impeller [1-4]. Murakami et al. [1] studied the influence of an adjacent bend on velocity distribution in the channel and the pressure distribution on the channel wall. According to the paper, remarkable asymmetry of the velocity distribution is generated due to a non-uniform flow passing through the upstream elbow, especially when the pump is operated on an over flow rate condition. Studies on suction channels in multistage pumps [2-3], for which the design restrictions on configuration and dimensions are relatively stringent, confirm that there is a relationship between non-uniformity of the flow in the circumferential direction and deterioration of the suction performance, and that the onset of cavitation is governed by the local inlet angle of the flow against the impeller blade. Wakabayashi et al. [4] conducted a systematic study on velocity and total pressure distributions on cross sectional planes at each outlet of various types of suction channels. They quantitatively demonstrated the effects of a contraction bend or contraction cone installed downstream of a bend in regard to pressure loss and velocity distribution.

The present paper describes a detailed study of the influences of configuration parameters on flow uniformity at the impeller inlet. The objective of the study is to improve the pump’s suction performance by optimizing the design of the suction channel; such a design will eliminate cavitation or its uneven distribution through suppression of the secondary flow in the suction channel. Design of experiments (DOE) and computational fluid dynamics (CFD) modeling were applied to evaluate the sensitivity of suction channel performance to several primary design parameters and the optimum combination of these parameters is discussed. Experiments were also carried out to verify some of the results obtained by CFD modeling.

2. Nomenclature

\[
\begin{align*}
g & \text{ Acceleration of gravity (m/s}^2) \\
H & \text{ Total head (m)} \\
NPSH & \text{ Net positive suction head (m)} \\
P & \text{ Pressure (m)} \\
Q & \text{ Flow rate (m}^3/s) \\
u & \text{ Circumferential speed (m/s)} \\
\phi & \text{ Flow coefficient} = Q/(A_d u_2) (-) \\
\sigma & \text{ Cavitation number} = NPSH/(u_2^2/2g) (-) \\
\varphi & \text{ Head coefficient} = H/(u_2^2/2g) (-) \\
\end{align*}
\]

2.1. Subscripts

- 2 Impeller outlet
- d Design point
- i Cavitation inception

3. Design, CFD method, experimental method

3.1. Sensitivity analysis by DOE

A sensitivity analysis of the intensity of the secondary flow at the impeller inlet to changes in the primary design parameters was conducted. The final shape of the suction channel was obtained based on this sensitivity analysis.
The intensity of the secondary flow was evaluated using the quantity \( U_{\text{dev}} \). A cross-sectional area at the impeller inlet was divided into seven annular sections, and for each section the standard deviation of the circumferential velocity distribution was obtained using the averaged velocity for every 45 degrees. The value of \( U_{\text{dev}} \) corresponds to the signal-to-noise ratio of the average of the standard deviations of the seven sections. Smaller values of \( U_{\text{dev}} \) are preferable, corresponding to lower dispersion of the circumferential velocity at the impeller inlet. The minimum pressure on the wall of the suction channel at the outlet of the bend was also evaluated using the quantity \( P_{\text{min}} \). The value of \( P_{\text{min}} \) corresponds to the signal-to-noise ratio of the minimum pressure at the outlet of the bend, which is usually observed on the inside portion of the channel nearest to the center of the bend curvature. \( P_{\text{min}} \) is an indicator of the excessive increase in flow velocity at this point when the bend is a nozzle accompanying a contraction flow. Larger values of \( P_{\text{min}} \) correspond to a larger margin for cavitation to occur, which is desirable.

A DOE approach was employed to conduct the sensitivity analysis. DOE is an efficient method to study the sensitivity of objective functions to a number of control factors with a minimum of experiments, and the combination of the control factor values was determined in accordance with an orthogonal array. Figure 1 shows the control factors of the flow system. Of these, four primary design parameters were examined in this study, and a 4-factor 3-level L9 orthogonal array was applied. In addition to the parameters of the bend radius \( R \) and the ratio \( A_{\text{out}}/A_{\text{in}} \), two nozzle parameters \( L \) and \( A_{\text{min}}/A_{\text{eye}} \) were chosen to reflect the results of Wakahbayashi et al. [4], who report that a nozzle installed downstream of a bend has a large influence on the outlet flow of the suction channel. The parameter \( A_{\text{out}}/A_{\text{in}} \) defines the ratio of the cross sectional areas of the bend’s outlet/inlet only; the variation of the cross sectional area along the flow path is defined to be continuous and as smooth as possible.

3.2. CFD
The nine shapes of the suction channels designed in accordance with the L9 orthogonal array were analyzed using CFD, yielding flow velocity distributions at the impeller inlet and the pressure distribution in the channel bend. Figure 2 shows an example of the analysis models. The computational domain to be analyzed consists of a suction channel and an identical impeller for all experiments. To reduce the computational load, impeller blades were not modeled and the only rotating members were the hub and shroud walls. The number of nodes of the suction channel was kept to around 1.4 million and steady state analyses were carried out. The turbulence model RNG k-\( \varepsilon \) was employed in the calculations, and total pressure and mass flow rates were prescribed at inlet and outlet boundaries respectively.
3.3. Experiments
To verify the results of the sensitivity analysis obtained by CFD, actual experiments were conducted. The suction channel in the experimental apparatus had a transparent observation window. Using a stroboscope synchronized with the rotational speed of the pump shaft, the cavitation at the leading edges of the impeller blades was observed. The experiments were carried out for several flow rates from $\phi / \phi_0 = 0.8$ through 1.2. For each flow rate, the $NPSH$ was varied while the flow rate was held constant, and $NPSH_i$ was measured. The test apparatus was a closed loop, and $NPSH$ was controlled by varying the internal pressure of the loop.

4. Results and discussion

4.1. Results of sensitivity analysis
The modeled response of the fluid flow to the parameters varied in the DOE is shown in figure 3. A double circle in each diagram denotes that the marked level is preferable to the other two levels.

Regarding the radius of the bend $R$, smaller radii resulted in smaller values of $U_{dev}$ even though the direction of the main flow is forced to change 90 degrees relatively rapidly in the bend. This is because of the shorter flow path in the bend, in which the secondary flow is generated and develops. For the parameter $L$, which is the distance from the impeller’s reference point $Z_0$ to the cross sectional plane of the channel where the sectional diameter is the smallest, larger values of $L$ correspond to smaller $U_{dev}$, which means that it is preferable to squeeze the suction channel earlier after the flow has passed through the bend and to have a looser diffuser toward the impeller inlet. On the other hand, preferable levels of $P_{min}$ were produced by higher values of $R$ and lower values of $L$, which was the inverse of the $U_{dev}$ behaviour (so called ‘trade-off’ relationship). The smallest value of $R$ and largest value of $L$ were adopted for subsequent tests by placing priority on the suppression of the secondary flow.

Both $U_{dev}$ and $P_{min}$ showed similar effects of changes in the ratio $A_{min}/A_{eye}$, i.e. the two largest values of the ratio marked by single circles produced relatively constant values of $U_{dev}$ and $P_{min}$. However, the largest value of $A_{min}/A_{eye}$ is 1.0, which means that the minimum diameter of the suction channel is identical to the impeller eye, and that the channel downstream of $Z_{min}$ is a straight pipe. Since this channel shape is the simplest and cheapest to manufacture, the value of $A_{min}/A_{eye}$ was fixed at 1.0. For these three parameters $R, L, and A_{min}/A_{eye}$, the selected levels were indicated by rectangular boxes in figure 3.

4.2. Experimental verification
The ratio of the outlet area of the bend to its inlet, $A_{out}/A_{in}$, governs the balance of the contraction rates of the bend and of the downstream straight nozzle. Among the three levels specified in the DOE, the smallest $A_{out}/A_{in}$ ratio showed the best $U_{dev}$, indicating that squeezing the diameter of the channel rather
aggressively at the bend and having a looser nozzle downstream of the bend effectively produce a uniform flow at the impeller inlet. Figures 4 through 6 show the comparison of the secondary flow and the characteristics of the cavitation inception in two suction channels modeled by CFD having different values of $A_{out}/A_{in}$.

Figure 4 shows the contours of in-plane velocity distributions at the inlet of the impeller obtained by CFD, where dark colours denote large velocities. For the suction channel with a small $A_{out}/A_{in}$, the dark coloured area near the wall on the inside (i.e., toward the center of the bend curvature) is small, indicating that the secondary flow has been suppressed. This is attributed to the suppression of boundary layer growth, which is one of the factors that generate the secondary flow, by squeezing the cross sectional area of the bend gradually toward its outlet to increase the flow velocity. Figure 5 is a visualization result of the cavitation regions, those with pressure less than the fluid’s vapor pressure, generated on the suction side of the impeller blades by two-phase CFD analysis. Each picture overlays a second picture whose impeller rotational angle is shifted by half a blade pitch. For the case of small $A_{out}/A_{in}$, the extent of the area with pressure below the vapor pressure is relatively similar around the impeller. For the suction channel with large $A_{out}/A_{in}$, on the other hand, the areas with pressure less than the vapor pressure on the right half of the picture are larger while they are smaller on the left, indicating an uneven cavitation distribution due to the stronger secondary flow.

In figure 6, experimental results of the cavitation number characteristics are plotted against the normalized flow coefficient. The cavitation number was calculated by $NPSH_i$ measured by the model pump. The cavitation number was smaller for the suction channel with smaller $A_{out}/A_{in}$. The result verifies that a smaller $A_{out}/A_{in}$ ratio suppresses the secondary flow in the bend, and that an improvement in flow velocity uniformity at the impeller inlet is obtained as a consequence.
4.3. Further study on $A_{out}/A_{in}$ in detail

Based on the discussion above, one might choose a small value for $A_{out}/A_{in}$ by placing priority on the suppression of the secondary flow in the same manner as for $R/L_0$ and $L$. However, a detailed examination was conducted to seek the optimum value, since the other evaluation object dropped dramatically as $A_{out}/A_{in}$ decreased.

Among the three levels chosen for flow velocity in the present DOE, the range for the following detailed study was limited to near the lower two levels, around which four values were selected and used in a CFD analysis. Parameters other than $A_{out}/A_{in}$ remained the same as the values marked by the rectangular boxes in figure 3.

The evaluation objects were $U_{dev}$ and $P_{min}$, the same as the previous DOE above, and the results are shown in figure 7. The value of $U_{dev}$, denoted by diamonds in the figure, was found to reach a minimum at a critical value, labeled $\beta$ in this work, and to increase for any other $A_{out}/A_{in}$ ratio. $P_{min}$, on the other hand, was a monotonically increasing function of $A_{out}/A_{in}$ as expected. This is because the pressure drops more when the cross sectional area of the bend’s outlet is smaller, accompanying a faster flow. Although the values of $P_{min}$ plotted in figure 7 are sufficiently low relative to the lowest NPSH expected from the pump station where the suction channel of the current study is installed, the degree of the pressure drop is inversely proportional to the square of the cross sectional area of the channel, and $P_{min}$ exhibits a dramatic fall as $A_{out}/A_{in}$ decreases.

Pressure contours on the walls of the suction channels with $A_{out}/A_{in} = \alpha$ and $\beta$ are shown in figure 8. For the case of $A_{out}/A_{in} = \alpha$, there appears an area where the pressure drops rapidly at the bend’s outlet section on the inside of the channel, while the pressure drop at the corresponding place for the case of $A_{out}/A_{in} = \beta$ is much more limited.

Taking all these issues into account, one would conclude that there is an optimum ratio of $A_{out}/A_{in}$ ($= \beta$) to effectively suppress the secondary flow at the impeller inlet, which keeps the pressure drop at the outlet of the bend within the range that ensures the pump’s high reliability.

5. Summary

Configuration of the suction channel of a large centrifugal pump having a 90-degree bend was studied to suppress the secondary flow at the impeller inlet. A DOE was employed for a sensitivity analysis of the selected configuration parameters, and nine types of suction channels were numerically analyzed by CFD. The results revealed that a smaller bend radius (with a shorter flow path as a consequence) effectively suppresses the secondary flow at the impeller inlet. An optimum value for the ratio of the cross sectional areas of the bend’s outlet/inlet was found by considering the balance of the contraction...
rates of the bend and of the straight nozzle downstream of the bend. The results will help in the design of large centrifugal pumps with better suction performance and higher reliability.

References
[1] Murakami M and Heya N 1966 *Bulletin of JSME* 9(34) 344-52
[2] Tanaka S, Mase M, Ueyama T and Sudou S 1988 *Trans. JSME* B 54 1693-7
[3] Tanaka S, Koseki F and Mase M 1988 *Trans. JSME* B 54 1698-702
[4] Wakabayashi F and Kondo M 1967 *J. JSME* 70 416-9