The influence of casing treatment on the performance of a centrifugal pump

Yuan Li¹, Hua Chen¹*, Xiangjun Li¹, Minghe Jiang², Guinian Wang²

¹ Dalian Maritime University, 1 Linghai Road, Dalian China 116026
² Danai Pumps Co., Ltd, 86 Liaohe East Road, Dalian China 116630

*Corresponding author, Email: huaichen204887@163.com

Abstract. Centrifugal pumps may run under off-design conditions such as low flows. Under such conditions, back-flow may occur at the impeller inlet and block a part of the flow path and cause energy loss. Casing treatment has been widely used in centrifugal compressors to improve their performance at low flow conditions, but there is little in the open literature on the effect of casing treatment on the performance of centrifugal pumps. In this paper, a SC100-200 centrifugal pump is studied with a casing treatment. First, the CFD method employed was described and verified by the experimental results of the pump. CFD indicates that the starting point of the back-flow is around discharge coefficient of 0.111 of the pump, and the back-flow intensity gradually increases with the decrease of flow rate reaching another critical point at discharge coefficient of 0.061 where the back-flow upstream traveling distance suddenly increases significantly. Then a casing treatment is created in the inlet end-wall and front cover using a U-tube type configuration. The numerical simulation was carried and it is found that a part of the back-flow at the inlet and front-side cavity of the centrifugal pump is sucked to the U-tube of the casing treatment and returns to the pump inlet, the back-flow and entropy production at the impeller inlet are reduced, and the head and efficiency of the pump is little affected. Further research shows that the longer the length of the U-tube is, the stronger the suction effect will be, and the smaller the back-flow of the pump.

Keyword. Centrifugal pump; Casing treatment; U-tube; Back-flow

Nomenclature

| Symbol | Description                                      | Unit     |
|--------|--------------------------------------------------|----------|
| D      | Impeller outlet diameter                         | m        |
| D₁     | Impeller inlet diameter                          | m        |
| D_d    | Pump discharge diameter                          | m        |
| G      | Gravitational acceleration                       | m/s²     |
| H      | Head of the pump                                 | m        |
| L      | Axial distance of the monitoring point from the impeller inlet | m |
| Q      | Volume flow rate                                 | m³/s     |
\[ V_t \] Circumferential velocity, m/s
\[ V_z \] Axial velocity, m/s
\[ \Omega \] Angular speed of pump, radian/s

1. Introduction

For various reasons centrifugal pumps may deviate from their design condition and operate at reduced flows. When a high-speed rotating centrifugal pump operates under small flow condition, a back-flow phenomenon will inevitably occur at its impeller inlet, and as the flow rate decreases, the back-flow region is expanded. The occurrence of back-flow has many adverse effects on the pump, including mainly: (1) energy consumption due to the back-flow, which reduces the efficiency of the pump; (2) instability of the flow and its induced noise and mechanical vibration, which affects the life of the pump and (3) cavitation resulting in violent flow and pressure pulsations. Therefore, it is of great practical significance to study the back-flow and its control when a centrifugal pump needs to operate at reduced flow rates.

Many scholars put forward different solutions to overcome the back-flow problem. Saha et al. [1] cut j-shaped grooves in the inlet pipe of a mixed flow pump to reduce the back-flow velocity and upward propagation distance of the flow. Kochevsky et al. [2] set an adjustable guide vane at the inlet of an axial flow pump to improve the inflow condition by adjusting the incidence angle of incoming flow. Oshima and Toykura [3][4] installed an orifice plate before the inlet of inducer to weaken the back-flow. P Cooper et al. [5] proposed that the back-flow at the suction connection could be solved by installing a reverse current regulator at the inlet of the inducer. Casing treatment has been widely used in centrifugal compressors to improve their performance at low flow conditions. Chen and Lee [6] reviewed the techniques of casing treatment on centrifugal compressors to improve the working state of the compressors at reduced flow rates and compressor efficiency. However, there is no open literature on the effect of casing treatment on the performance of centrifugal pumps.

In this paper, a SC100-200 centrifugal pump from Danai pumps Co., Ltd is taken as the research object subject to a casing treatment. Firstly, the CFD method employed was verified by experimental results to provide a simulation tool. Then, the distribution of the flow field and the morphology of the vortices of back-flow were studied. On this basis, a control scheme was proposed to improve the back-flow characteristics by using casing treatment, and the reasons for the treatment to control the back-flow and reducing entropy production were expounded. Finally, the length of the U-tube in the casing treatment was changed to further expand the control advantage.

2. The pump studied

In this paper, a single-stage, double-volute type SC100-200 centrifugal pump with pear-shaped volute from Danai Pumps is studied. The centrifugal pump has a closed or shrouded impeller. The main design parameters and three-dimensional shape of the pump are shown in Table 1 and Figure 1, respectively.

| Table 1. The main design parameters of SC100-200 pump. |
|---------------------------------|
| H | Q   | \( \omega \)  | Number of blades |
| 52.1m | 0.061m³/s | 308.9radian/s | 6 |
| D | D₁ | D₂ | Working medium |
| 0.209m | 0.125m | 0.100m | Water |
3. **Numerical method and verification**

3.1. *Governing equations and turbulence model*

The interior flow of the centrifugal pump is a turbulent flow of three-dimensional, incompressible viscous fluid. The governing equations used are the Reynolds Averaged Navier-Stokes (RANS) equations. The equations were solved using commercial software CFX which employs a finite-volume method based discretization of the governing equations. The convection term was solved in the high resolution format, and the convergence criterion was set to $10^{-6}$. Since the standard $k-\varepsilon$ model has been widely verified and applied in scientific research and engineering practice, and the numerical results of $k-\varepsilon$ model prediction are stable and reliable, this turbulence model was used in this study.

3.2. *Interface setting, inlet and outlet boundary conditions*

As shown in Figure 2, the impeller and the inlet pipe, the impeller and the volute, the front-side cavity and the inlet pipe, the front-side cavity and the volute, and the back-side cavity and the volute are all connected with a rotor-stator interface and the respective watersheds are connected into one unit. In the steady flow simulation, the Frozen-Rotor interface in the CFX was adopted, and the boundary conditions of the pressure inlet and the mass flow outlet were imposed.

3.3. *Mesh generation*

Since the key components of this study are the impeller and the front-side cavity, the NUMECA software was employed to obtain their structured mesh. The ICEM software was used for other components (inlet pipe, back-side cavity and volute) to obtain highly adaptive unstructured meshes due to their complex structure. The meshes of flow passage components are shown in Figure 3.
Figure 2. Meridian channel and interface of the pump

Figure 3. The mesh of flow passages
3.4. Grid-independence check
In order to avoid the influence of the mesh density of computational grid on solution, a grid-independence check was carried out and the results at discharge coefficient of 0.083 are reported here. By keeping the grid number of other components unchanged, the number of the impeller and volute meshes was independently changed as shown in Figure 4. It can be found that when the impeller grid is more than 3.2 million and the volute grid is more than 2.16 million, the grid has little effect on the head and efficiency. Considering the accuracy and time of computation, 4.1 million impeller grids and 3.15 million volute grids were selected for this study. The total number of grids is 14 million, and the maximum $y^+$ value of the mesh is less than 10, meeting the requirement of meshes quality in the near wall area.

![Figure 4. Grid-independence check](image)

3.5. Numerical method validation
The head coefficient and efficiency curves of the pump were computed, and the numerical results are compared in Figure 5 with those measured by Danai Pumps. The simulated head coefficient and efficiency are in good agreements with the experimental results, indicating that the numerical method employed in this paper is accurate and credible.
4. The back-flow flow field
When the flow of a centrifugal pumps reduces, back-flow may occur in the inlet of the pump, that is, part of the fluid flows back from the impeller to the inlet pipe, producing a velocity in the axial direction opposite to the direction of the main flow. As the impeller is rotating, the returning fluid transfers some of the energy of the impeller rotation to the fluid in the inlet pipe, producing a circumferential velocity and causing the fluid in the inlet pipe to pre-whirl. Therefore, studying the velocity distribution of the return flow field requires analysis of the distribution of axial velocity and circumferential velocity. Numerical method described earlier was used for this purpose.

4.1. Axial velocity distribution of the impeller inlet section
When the flow rate changes, the impeller inlet has different degrees of back-flow, which results in the change of axial velocity distribution at the impeller inlet section. Figure 6 shows pitchwise averaged axial velocity at different radii at the inlet section. As the flow rate decreases, the area and the velocity of the back-flow increase, while the area and the velocity of the main stream decrease.

Figure 6. Pitch averaged axial velocity of the inlet section at various mass flow rates

Figure 7(a) shows the curve of the axial velocity in the inlet pipe at different discharge coefficients (Q/ΩD³). L is the axial distance of the monitoring point from the impeller inlet, and Vz is the axial velocity near the inlet pipe end-wall. The figure shows that axial velocity dips approaching the impeller, and rises again when leading edge of the impeller is reached. When the discharge coefficient is 0.111, the velocity drop is suddenly increased and the velocity becomes negative, implying the occurrence of back-flow, see Figure 6 as well. So, it can be judged that 0.111 or so is a key discharge coefficient at which the back-flow occurs. With the further decrease of
the flow coefficient, the back-flow intensifies, the velocity and the axial dimension of the back-flow area also increase. The dip of the axial velocity is however less dramatic than at 0.111. The back-flow intensity reaches another critical point at discharge coefficient of 0.061 where the back-flow upstream traveling distance suddenly increases significantly.

Figure 7. Near wall velocity distributions in the inlet pipe

4.2. Distribution of circumferential velocity in the inlet pipe
The back-flow transfers the energy of the impeller rotation to the inlet pipe, causing a pre-whirl that produces a circumferential velocity \( V_t \), as shown in Figure 7(b). In the case of discharge coefficient being 0.136, the tangential velocity in the pipe is virtually zero except at the vicinity of impeller leading edge. Here the rotating direction of the impeller is defined as the positive direction of \( V_t \). When the discharge coefficient reduces to 0.111, there is a rapid increase of the tangential velocity near the leading edge, coincided with the occurrence of back-flow shown in Figures 6 and 7(a). As the flow rate further decreases, the tangential velocity and their upstream reach both increase. In the case of discharge coefficient being 0.061, the upstream influence of the back-flow is far reaching and substantial, as can be seen in Figure 7(a) as well. The axial range in which the pre-whirl exists at each flow rate is substantially the same as the range in which negative axial velocity occurs, which indicates that the back-flow is the main cause of the pre-whirl in the inlet pipe.

According to the above analysis, the occurrence of back-flow causes the reverse axial velocity and pre-whirl, which distorts the normal inlet flow field and may jeopardize the operation of the pump, so the back-flow must be controlled.

5. The control principle of casing treatment

5.1. The method of casing treatment
The method of casing treatment is commonly used for centrifugal compressors to control the back-flow near casing end-wall and to extend the operating range of compressors. The concept was applied to the centrifugal pump here, and one implementation is shown in Figure 8. The impeller front or shroud cover and the inlet end-wall are both grooved, defining a U-tube type flow passage. The U-tube connects the impeller with the inlet pipe by passing impeller leading edge. Due to the existence of the wear-ring clearance, the Bleed is divided into two parts, so that the U-tube is connected with the front-side cavity and the inlet pipe in addition to the impeller. The U-tube structure is 360° covering entire circumference, this means that the piece of inlet piping under it will not
hold its position without some kind of support. The support can be provided by a few ribs connect it with the pump casing. Such support was omitted in this study, however, to simplify the work.

![Figure 8. The method of casing treatment of the pump](image)

5.2. General mechanism of casing treatment in the studied pump
Since there is a positive pressure gradient between the Bleed inside the impeller and Injector exposed to the inlet pipe established by impeller blade loading, any back-flow fluids are driven from the Bleed through the Injector to the lower pressure zone of pump inlet, bypassing impeller leading edge to maintain blade loading there. The suction of the end-wall boundary-layer around the Bleed and the keeping of blade leading edge loading ensure a health impeller suction and inflow. Figure 9 sketches the flow around and in the U-tube. The source of fluid entering the U-tube can be divided into two parts, namely the fluid from the impeller and that from the front-side cavity. These fluids are mixed in a U-tube and finally enter the inlet pipe through the injector.

![Figure 9. The flow around and in the U-tube](image)

6. Results and discussion
Numerical simulation was carried out to study the influence of the U-tube casing treatment on the performance of the pump and the detailed mechanisms behind. The back-flow is established and stable under the condition of
discharge coefficient of 0.083, so this discharge coefficient is selected for analysis. Three bridge lengths of the U-tube, namely 34mm, 52mm and 80mm were studied for the effects of the bridge length.

6.1. Velocity distribution of impeller and volute

6.1.1 Velocity distribution of impeller. The flow velocity distribution of the impeller of the original machine is shown in Figure 10(a). It can be seen that near the impeller front cover, the fluid in the impeller recirculates and diffuses into the inlet pipe. Figures 10(b), (c) and (d) show the effect of U-tubes of different bridge lengths on the impeller velocity distribution. As the figures show that the suction of the U-tube reduces the back-flow inside the impeller and further prevents it from spreading into the inlet pipe, the streamlines inside the impeller are more regular with the casing treatment, and the longer the U-tube bridge is, the stronger these effects are. The bridge length effect is due to the injection to the main flow by the injector, Figure 9, which forces a movement of the incoming main flow away from the shroud, thus reduces the forward momentum of the flow toward impeller leading edge near the shroud, and moving the injector away from the leading edge improves the forward momentum near the leading edge.

![Velocity and streamline distribution of impeller with and without casing treatment](image)

**Figure 10.** Velocity and streamline distribution of impeller with and without casing treatment

6.1.2. Velocity distribution of volute. The fluid velocity distribution in the volute is shown in Figure 11. It can be seen that due to the existence of the two volute tongues and the reduced flow rate through the volute,
the flow at the tongues of volute is chaotic in the baseline pump, Figure 11(a), as shown by the two red arrows. Figures 11(b), (c) and (d) show the influence of U-tubes of the three lengths on the flow in the volute respectively. As in the case of centrifugal compressors [5], by connecting impeller with inlet pipe that has a more circumferentially uniform pressure distribution than impeller at reduced flow rates, the U-tube makes pressure more uniform in the impeller and downstream volute of the pump, and reduces the pressure jump across two volute tongues, which then decreases the back-pressure of the impeller subject to when it passing through the tongues, and weakens the back-flow of the impeller. The flow at the volute tongues becomes more uniform and better after the addition of the U-tube, and the longer the length of the U-tube is, the stronger these effects are, which is consistent with the improvement of the U-tube on back-flow shown in Figure 10.

Figure 11. Velocity and streamline distribution of volute with and without casing treatment

6.2. Back-flow vortex morphology
The casing treatment also changes the shape and strength of the back-flow vortices. Figure 12 shows the back-flow vortex morphology and strength at the discharge coefficient of 0.083. It can be seen that original pump has
a strong and large vortex at the front part of its impeller, and a weak one towards impeller exit as indicated by large eddy viscosity values. The enlarged view shows isokinetic streamlines that indicate a large back-flow at the front cover of the original impeller spreading well into the inlet pipe. After the casing treatment, the strength of the both vortices decrease, so is the back-flow intensity, and the longer the length of the U-tube bridge is, the more advantageous the reduction of the strength and size of the vortices will be. In the cases of 52mm and 80mm bridge length, the second vortex is barely visible. These findings are consistent with the results shown in Figure 10.

![Figure 12](image)

**Figure 12.** Vortex morphology of the impeller with and without casing treatment, enlarged view adds isokinetic streamlines.

6.3. **Effect of U-tubes lengths on entropy production**

The casing treatment also causes the change of local entropy production. **Figure 13, Figure 14** and **Figure 15** show the entropy production of different parts of the studied pump when the discharge coefficient is 0.083. In the original pump, Figure 13(a), the stagnation of the flow and the back-flow close to shroud region generate high entropy in the region. When the casing treatment is added, Figures 13(b) to (d), due to the suction action of
the U-tube, the entropy production near the shroud at impeller inlet region is significantly reduced. The production decreases with the lengthening of the U-tube as the suction action is enhanced at the longer lengths.

Figure 14(a) shows the entropy production at the section where the tongue of volute is located, it can be seen that the flow chaos at the volute tongues, Figure 11(a), cause high losses. Figures 14(b) to (d) show that after the addition of the U-tube, the entropy production at the volute tongue regions decreases significantly, and the longer the length of the U-tube, the more reduction of the entropy production is. At the same time, the entropy production area of impeller inducer decreases but some entropy is generated in the mid passages of the impeller. It also can be seen that part of the loss in the volute is carried to the back-side cavity with or without the casing treatment.

Some high entropy is generated at the bottom of back-side cavity as shown in Figure 15. Asymmetries of the entropy production in the circumferential direction can be seen, and this is mainly caused by the downstream volute exerting an asymmetric boundary condition on the impeller and to a less degree by the Frozen-rotor interface used in CFD. In Figure 15(b), the entropy production of the 34mm U-tube is comparable with that of the original machine shown in Figure 15(a). However, with the increase of the U-tube length, the loss in the back-side cavity decreases significantly, Figures 15(c) and (d).

![Figure 13. The entropy production at the impeller inlet](image_url)
Figure 14. The entropy production in impeller, cavities and volute (sectioned through tongues)

Figure 15. The entropy production in the back-side cavity

6.4. Effects on pump performances

Figure 16 compares the performance characteristics of the centrifugal pump with and without casing treatment. It is found that there is a slight increase in the head and efficiency of the pump after the treatment.
7. Conclusion
A SC100-200 centrifugal pump from Danai Pumps Co. Ltd. was studied with a casing treatment. First, the CFD method employed was described and verified by the experimental results of the pump. The CFD indicated that the starting point of the back-flow through the impeller was around discharge coefficient of 0.111 of the pump, and the back-flow intensity gradually increased with the decrease of flow rate reaching another critical point at discharge coefficient of 0.061 where the back-flow upstream traveling distance suddenly enlarged significantly. Then the casing treatment was created in the inlet end-wall and front cover of the impeller using a U-tube type configuration. It was found that a part of the back-flow at the inlet and front-side cavity of the impeller was sucked to the U-tube of the casing treatment and returned to the pump inlet. The back-flow and entropy production at the impeller inlet region were both reduced, and the head coefficient and efficiency of the pump were marginally improved. Further research showed that the longer the length of the U-tube was, the stronger the suction effect would be, and the smaller the back-flow of the pump.

Acknowledgments
The authors would like to thank Danai Pump Co., Ltd. for their support to this project and permission to publish this paper. They also thank Mr. Chao Li at Dalian Maritime University for his valuable helps in this study. The authors disclose the receipt of the following financial support for the research, authorship, and/or publication of this article: This research is supported by the National Natural Science Foundation of China (Grant No. 51906027), the Doctoral Initiation Research Funds of Liaoning province (Grant No. 2019-BS-027) and the Fundamental Research Funds for the Central Universities (Grant No.3132019116).

Reference
[1] Saha S L, Kurokawa J. Suppression of Performance Curve Instability of a Mixed Flow Pump by Use of J-groove. Journal of Fluids Engineering, 2000, 122(3): 592-597.
[2] Kochevsky A N, Kozlov S N, Aye K M, et al. Simulation of Flow Inside an Axial-Flow Pump with Adjustable Guide Vanes. ASME 2005 Fluids Engineering Division Summer Meeting. 2005: 1397-1404.
[3] Oshima M A. Study on Suction Performance of A Centrifugal Pump With An Inducer. B JSME 1967, 10:959-965.
[4] Toyokura T and Kubota A. Studies on back-flow to the suction side of mixed-flow impeller blades. B JSME 1969, 12:215-221.

[5] Cooper P, Sloteman DP and Dussourd JL. Stabilization of the off-design behavior of centrifugal pumps and inducers. Proceedings of the second European congress on fluid machinery for the oil, petrochemical and related industries, March 1984, pp.1984–1982.

[6] Chen H. and Lei V.M. Casing treatment & inlet swirl of centrifugal compressors. Journal of Turbomachinery, ASME, July 2013, Vol. 135/041010-1~8.