Unsteady RANS simulation of three-stage centrifugal pumps with different impeller-diffuser gaps

T Takamine¹, S Watanebe² and K Miyagawa³

¹ Graduate School of Engineering, Kyushu University, Fukuoka, Japan
² Department of Mechanical Engineering, Kyushu University, Fukuoka, Japan,
³ Faculty of Sci. & Eng., Waseda University, Tokyo, Japan

fmnabe@mech.kyushu-u.ac.jp

Abstract. In this study, unsteady RANS simulation is attempted for a three-stage centrifugal pump, a target pump for the workshop “Single- & Multi-Stage Pump Flow Prediction” which is being held in 29th IAHR Symposium on Hydraulic Machinery and Systems. A commercial code SCRYU/Tetra developed by Software Cradle Co. Ltd. is adopted for the unsteady RANS simulation. Our primary interest is to understand the effect of the gap between impeller trailing edge and diffuser leading edge on the axial thrust characteristics of the multi-stage centrifugal pumps. To do so, in addition to the simulation for the original pump model, the impeller-diameter cut model is also simulated. The diffuser inlet to the impeller outlet diameter ratio is 1.05 for the cut model against 1.02 for the original model. The good agreement is obtained for the hydraulic performance of the original model at the design flow rate, but only the fair agreement is obtained for the axial thrust force. From the pressure distributions inside the front and back side gap of the impeller, it is found that the discrepancy is due to that of the pressure distribution inside the back side gap of the second (and also perhaps the first) impeller. The effect of impeller-diffuser gap on the hydraulic performance and axial thrust force is predicted to be small at the design flow rate through the present computations. At the low flow rate, the balancing flow rate is significantly over-predicted by the present simulation. The reason for this remains unclear and will be hopefully made clear in our future study.

1. Introduction

In recent years, turbomachines are expected to have higher head and larger capacity with compact structures. Centrifugal pumps are designed with larger number of stages and higher operating speed in order to meet this expectation. Consequently, the significant vibration of shaft system caused by unstable fluid forces acting on rotors could become serious problem considering the huge axial thrust forces acting on the rotor system in high pressure multi-stage centrifugal pumps. Therefore, in order to ensure the reliability of such pumps, it is important to understand the characteristics of fluid forces as well as those of unsteady flow phenomena.

In our previous studies, a series of experiments has been carried out for a three-stage centrifugal pump. Firstly, we investigated the effect of axial rotor offset on fluid forces, and it was found that the axial offset significantly affected the axial thrust force in the low flow rate range [1]. Besides, by cutting the impeller diameter aiming at weakening the rotor-stator interaction, we investigated the effect of impeller-diffuser gap on the fluid forces, and it was found that the effect of axial offset on the axial thrust was reduced with larger gap [2]. At the same time, only in the case with larger impeller-diffuser
gap, the remarkable pressure fluctuation was observed, which seemed to be caused by single-cell rotating stall in the vaned diffusers propagating in 14% of impeller rotational speed [3].

There have been many studies on unsteady flows in impellers and diffusers in single stage turbomachines, some of which focused on the impeller-diffuser interaction. Arndt et al. [4] studied the pressure pulsation due to the interaction between impeller trailing edge and diffuser leading edge in several vaned diffusers. Sano et al. [5] carried out the experimental analysis on the effect of impeller-diffuser gap on the alternate/rotating stall in vaned diffusers. In addition, some numerical studies on the impeller-diffuser flow interactions have been more recently reported [6], [7]. However, there are only a limited number of studies reporting on the flow instabilities in multi-stage centrifugal turbomachines, even though the unsteady flow in such machines might have distinctive characteristics related to large number of stages. Shibata et al. [8] studied the rotating stall in the vaned diffuser in the part stage model of multi-stage centrifugal pump. Gianmario et al. [9] studied the rotating stalls in a multi-stage centrifugal blower with vaneless diffuser. Hiradate et al. [10] studied the rotating stall and surge phenomena in a multistage compressor focusing on the effect of inlet guide vane opening on the onset conditions of those instabilities.

Besides, the change of flow condition at impeller-diffuser gap may also influence the pressure distribution in the impeller side gaps, which is highly related to the axial thrust force. For suitable design of thrust-balancing devices as well as the selection of bearings, the accurate prediction of axial thrust force acting on rotor system is very important. In order to ensure the reliability of multi-stage centrifugal pumps, it is necessary to understand the effects of impeller-diffuser gap on the entire flow field of pumps, including not only unsteady flow phenomena in impellers and diffusers but also the impeller side leakage flow. However, as far as the authors know, there are few studies focusing on the effect of impeller-diffuser gap on the flow conditions in a multi-stage centrifugal pump, especially on the impeller side leakage which is responsible for the axial thrust forces generated by impellers.

In the present study, in order to understand the characteristics of side leakage flows and the axial thrust force acting on shaft system, the unsteady RANS (Reynolds Averaged Navier-Stokes) simulation is carried out. A three-stage centrifugal pump provided by the Workshop “Single- & Multi-Stage Pump Flow Prediction” is used, whose geometry is the same as the test pump of our previous experiments. The computation model include not only the main flow paths but also the leakage flow clearances at annular seals such as liner ring, inter-stage bush and the balance piston. The influences of impeller-diffuser gap are investigated by changing only the impeller vane outer diameter. The calculations are conducted at the design flow rate and a part load condition (15% of design flow rate) for each impeller-diffuser gap case.

2. Analysis model and methods

2.1. Three-stage centrifugal pump

Figure 1 shows the schematic view of a three-stage centrifugal pump analyzed in this study. The pump consists of a suction casing, three identical impellers and diffusers, two identical return channels and a discharge casing. A part of working fluid (water) exiting from the third stage impeller flows into the impeller back clearance and return to the suction casing through the radial clearance at balance drum, while the main flow through the third diffuser directly goes into the discharge casing. The number of vanes of impellers is 7, while those of diffuser and return channels are 10. The gap between the impeller and the diffuser is small \((D_s/D_2 = 1.02)\), where \(D_2 = (318\text{mm})\) and \(D_2\) are the exit diameter of impeller vane and the inlet diameter of diffuser vane respectively. The rotational speed of impeller is set to the design rotational speed, \(N=1,600 \text{ min}^{-1}\). The design flow rate and head are 1.5 \(\text{m}^3/\text{min}\) and 113 \(\text{m}\) respectively. The stage specific speed of the test pump calculated following a textbook by Gulich [11] is \(n_p=35.9\) \([\text{m, m}^3/\text{s, min}^{-1}]\).

There exist some kind of leakage flow in the three-stage pump as below; firstly, the impeller front leakage flow from the impeller exit into the radial clearance at the liner ring through the impeller front side gap, which finally returns to the impeller inlet. Secondly, the impeller back leakage flow which
comes from the next stage inlet through the clearance at inter-stage bush, flowing into the diffuser inlet through the impeller back side gap. Finally, the balancing flow at the balance piston as mentioned above.

![Diagram](image)

**Figure 1.** Three-stage centrifugal pump.

### 2.2. Experimental method

In this study, in order to validate the results of present CFD analyses, our previous measured data are used. The total head rise of the pump is calculated from the difference between outlet and inlet pressure $P_{\text{out}}$ and $P_{\text{in}}$, assuming that inlet and outlet flows have uniform velocity distribution. The shaft torque $T$ is measured by a torque meter installed between the pump shaft and the motor shaft. The pump performance is summarized in non-dimensional forms, the flow rate coefficient $\phi$, the head coefficient $\psi$ and efficiency $\eta$, which are defined as follows;

$$\phi = \frac{Q}{AU}, \quad \psi = \frac{gH/3}{U^2}, \quad \eta = \frac{\rho g Q H}{T \omega} \quad (1)$$

where $A$ is the area of flow passage at the exit of impeller, $Q$ is the flow rate, $\omega$ ($=2\pi N/60$) and $U$ ($=D_2 \omega/2$) are the impeller angular and peripheral speeds, $g$ is the gravitational acceleration and $\rho$ is the density of water.

In order to investigate the mechanism of axial thrust force acting on each rotors, the static pressure at the several locations in the impeller side gaps $P_2$ and $P_3$ are measured in the second and third stages respectively. Because of the difference of pressure levels between each stages, the measured impeller side pressures were unified as the differential pressure from the static pressure $P_{d0}$ and $P_{d1}$ measured at the diffuser outlet of the upstream stages, for example second diffuser outlet pressure $P_{d2}$ is subtracted from the third stage side gap pressure $P_3$. Then, the static pressures measured at the impeller side gaps are summarized in static pressure coefficient $\psi_f$, defined as follows;

$$\psi_{fs} = \frac{P_f}{\rho U^2/2} \quad (2)$$

The axial thrust acting on shaft system is measured by load cells supporting the ball bearing, which is summarized as axial thrust coefficient defined as follows;

$$C_{FA} = \frac{F_A}{\rho AU^2/2} \quad (3)$$

where $F_A$ is the measured axial thrust for the whole rotor.

### 2.3. Numerical method

The unsteady RANS simulation is conducted using the commercial code Software Cradle SCRYU/Tetra V13 [12]. In the numerical model, the entire flow passage of the target three-stage centrifugal pump is modeled including the impeller side leakage paths and the balancing flow channel. The computational
A computational grid for the model is shown in Figure 2. Tetra-prism mesh is used for the suction and discharge volute casing, and hexahedral mesh is used for impellers, diffusers, impeller side clearances and annular seals domains. The number of computational grid is totally about 37 million nodes. The unsteady RANS simulation is carried out using the k-ω SST model for turbulent closure. For the inlet boundary, the fixed mass flow rate condition is applied, and constant static pressure condition is applied for the outlet boundary. ALE (Arbitrary Lagrangian Eulerian) method is applied between the rotating domains and stationary domains for unsteady analysis.

The time step of the unsteady simulation $\Delta t$ is set to be 1/360 of impeller rotational duration, i.e. $\Delta t \approx 1.04 \times 10^{-4}$ s. Simulations are carried out at the design flow rate ($\phi / \phi_d = 1.0$) and 15% of the design flow rate ($\phi / \phi_d = 0.15$) for at least six revolutions of impellers. The results of steady flow simulation with frozen rotor interfaces between rotationary and stationary domains are used as the initial condition. The steady performance such as pump performance and the axial thrust working on shaft system are evaluated by the time averaged flow field in the last two impeller rotation periods.

In the present study, since we are focusing on the effect of impeller-diffuser gap on the leakage flows and the resultant axial thrust force, we conduct the numerical simulation, in addition to the original gap case ($D_3/D_2 = 1.02$), in the case with larger gap than the original one by cutting the impeller vane outer diameter. The diameter ratio is $D_3/D_2 = 1.05$ ($D_3 = 310$ mm). It should be noted that, in our previous experiments, the shroud walls were also cut, therefore the radial clearance between the rotating shrouds and the stationary casing is relatively wide with 5 mm, while the shroud walls are not cut in the present CFD study.

3. Results and discussions

3.1. Pump performance and axial thrust force
The head coefficient and efficiency characteristics are shown in Figure 3. Figure 4 shows the balancing flow rate $q$ plotted against the flow rate normalized by the design flow rate. Open and closed symbols indicate experimental results and computed results respectively, while red and blue symbols indicate the original and the impeller-cut cases respectively. In simulation results, these performance characteristics are evaluated by the same way as in the experiment. The same impeller peripheral speed with the original gap is used for the normalization.
Through the comparison at the design flow condition ($\phi/\phi_d = 1.0$) for the original gap ($D_3/D_2=1.02$), it is found that the head is quantitatively well predicted by the present analysis, although the balancing flow rate is slightly under-predicted. The efficiency is over-predicted by the present simulation due to the neglect of the mechanical loss as well as the increase of volumetric efficiency with the smaller leakage. Looking at the difference of the design performance between two gap cases, the head is about 2.7% smaller for the smaller gap case ($D_3/D_2=1.05$) than the original gap case ($D_3/D_2=1.02$), which is consistent with the reduced Euler head with the impeller diameter cut.

Looking at the performance at the partial flow rate ($\phi/\phi_d = 0.15$), the head coefficient and the efficiency predicted by the present simulation is approximately agree with the experiment. However, the balancing flow rate $q$ is over-predicted in contrast to the case of the design flow rate; about 42% larger than the experiment. The little deviation of predicted head and efficiency from the measured ones at partial flow rate ($\phi/\phi_d = 0.15$) seem to be concerned with the increased working flow rate of impeller due to this large balancing flow rate, resulting in the reduced Euler head and the volumetric efficiency. The head at this partial flow rate is reduced by cutting the impeller diameter; one of the possible reasons is also the reduced Euler head with the impeller cut, and another possible reason is perhaps due to the increased flow unsteadiness due to weakened impeller-diffuse interaction. For quantitative evaluation of unsteadiness, more revolutions of impellers in the simulation seemed to be necessary, which will be reported in our future study.

Figure 3. Pump performance.

Figure 4. Balancing flow rate.

Figure 5. Axial thrust coefficient.

Figure 5 shows the axial thrust coefficient $C_{F_A}$ of the whole rotor plotted against the flow coefficient ratio of $\phi/\phi_d$. If we compare the thrust coefficient at the design flow rate ($\phi/\phi_d = 1.0$), it is unfortunately said that the accuracy of the present prediction is not very good; in the simulation, the value is positive while negative in the experiment. This result will be later investigated by the comparison of flow in the leakage flow gaps. The effect of the impeller cut on the thrust coefficient is
small at the design flow rate as can be seen in the figure 5. At the partial flow rate of $\phi/\phi_d = 0.15$, the axial thrust is more overestimated compared to the rated flow rate. At this flow rate, the effect of the impeller-diffuser gap is more significant than that of the design flow rate; the axial thrust force is seen to be increased by the impeller cut.

3.2. Static pressure and tangential velocity distribution in impeller side gaps
To understand the mechanism of impeller axial thrust, the impeller side pressure and velocity distributions are evaluated at lines 1 and 2 shown in Figure 6. Presented data are circumferentially averaged data of time-averaged value during two revolutions of impellers. Figures 7 and 8 show the static pressure distributions at the mid-section of impeller side gaps for the second and third stages respectively in the case with the original impeller ($D_3/D_2=1.02$) at (a) $\phi/\phi_d = 1.0$ and (b) $\phi/\phi_d = 0.15$. The horizontal axis in those figures indicates the radial position in the side gaps.

![Figure 6](image6.png)

**Figure 6.** Positions at impeller side gaps where pressure and velocity distributions are evaluated.

At $\phi/\phi_d = 1.0$, in the second stage, the CFD results agree well with the experimental ones in the front side gap, but do not quantitatively agree in the back side gap; the pressure difference between the front and back side gaps is over-predicted in CFD, resulting in the overestimation of the axial thrust force produced by the impeller in the second and perhaps the first stage. In the third stage, the difference of static pressure between the front and back gaps is very small, which is also simulated qualitatively well in CFD analysis. Therefore, the over-prediction of impeller axial thrust for the first and second impellers seem to be responsible for the discrepancy of total thrust force previously shown in Figure 5.

At the partial flow rate of $\phi/\phi_d = 0.15$, the CFD results do not agree well with experiments even qualitatively, especially in the second stage. In the third stage, the difference of static pressure between front and back sides is simulated to be as small as the experiment, while that in the second stage is significantly over-predicted. The qualitative characteristic of static pressure distribution, i.e. the slope of static pressure against the radial position is clearly different between CFD and experiment in the second stage. Similar to the result at $\phi/\phi_d = 1.0$, this could lead to the significant overestimation of the axial thrust force acting on impeller in the second and perhaps in the first stage, resulting in the over-predicted axial thrust of whole rotor shown in figure 5.

![Figure 7](image7.png)

**Figure 7.** Impeller side gap static pressure distribution (second stage, original impeller).

![Figure 8](image8.png)

**Figure 8.** Impeller side gap static pressure distribution (third stage, original impeller).
Figures 9 and 10 show the simulated side gap static pressure distributions in the second and third stage impellers respectively with $D_3/D_2=1.02$ and 1.05 for (a) $\phi/\phi_d=1.0$ and (b) 0.15. At the design flow rate ($\phi/\phi_d=1.0$), it can be seen that the impeller side pressure is entirely decreased slightly by reducing the impeller vane diameter because of the reduced head rise of each impeller. The pressure difference between the front and side gap of the impellers are almost unchanged, resulting in the similar total thrust coefficient in the both gap cases at this flow rate as has been seen in Figure 5. At the partial flow rate of $\phi/\phi_d=0.15$, the static pressure distributions are slightly different between the two impeller-diameter cases in the second stage, while almost the same in the third stage; the slope of side gap pressure is little more gradual with larger gap ($D_3/D_2=1.05$). However, this minor change of static pressure distribution could produce no significant increase of axial thrust force. Consequently, the decrease of negative axial thrust force produced by the balance drum due to the reduced balancing flow rate seems to be the main cause of the reduced total axial thrust in the impeller cut case ($D_3/D_2=1.05$).

Figure 9. Effect of blade cut on side gap static pressure distribution (second stage).

Figure 10. Effect of blade cut on side gap static pressure distribution (third stage).

Figures 11 and 12 show the computed tangential velocity distributions in the front and back side gaps of the second and third stage impellers respectively with $D_3/D_2=1.02$ and 1.05 for (a) $\phi/\phi_d=1.0$ and (b) 0.15. At the design flow rate ($\phi/\phi_d=1.0$), the impact of the impeller-diffuser gap is not very significant for the both second and third stage impellers. The velocity in the back side gap of the second impeller is much smaller than the half of local impeller circumferential velocity ($r\omega/2$ with $r$: radius and $\omega$: angular rotational speed), which seems to be the reason for the predicted larger pressure in this gap as has been seen in Figure 7. The leakage flow rate from the third stage inlet to the back side gap of the second stage (perhaps also from the second stage inlet to the first stage back gap) seems to be over-predicted by the present simulation. At the partial flow rate of $\phi/\phi_d=0.15$, with larger gap, the tangential velocity is slightly larger at front side but smaller at backside in the second stage, resulting in the slight change of static pressure distribution as has been shown in figure 9.

Figure 11. Effect of blade cut on side gap tangential velocity distribution (second stage).

Figure 12. Effect of blade cut on side gap tangential velocity distribution (third stage).
3.3. Velocity distribution at impeller outlet

Figures 14, 15 and 16 respectively show the circumferentially-averaged radial, tangential and axial velocity distributions along the line 3 at the second impeller outlet as indicated in Figure 13 for each impeller diameter case at (a) $\phi/\phi_d = 1.0$ and at (b) $\phi/\phi_d = 0.15$. The horizontal axis indicates the axial position along the line 3 from front shroud side. The corresponding illustration of the impeller shroud walls and stator walls are shown in each figure. At the design flow rate of $\phi/\phi_d = 1.0$, there is a little difference of velocity distribution between the cases with the original ($D_3/D_2=1.02$) and the impeller cut ($D_3/D_2=1.05$) cases, except that the magnitude of tangential velocity is smaller in the impeller cut case. On the other hand, at the partial flow rate of $\phi/\phi_d = 0.15$, substantial differences of velocity distribution between each gap cases are observed. With the larger gap ($D_3/D_2=1.05$), the tangential velocity is larger at the front side while smaller at the back side. The larger tangential velocity is brought into the front gap by the leakage flow, probably enhancing the rotation of fluid as shown in figure 11(b), and the smaller tangential velocity at the back side could be the evidence of the weaker fluid rotation in the impeller back gap. Generally, the impeller back leakage flow radially outward weakens the fluid rotation in the back gap [13]. However, in the present study, nevertheless the leakage flow rate at interstage bush is slightly reduced with larger gap ($D_3/D_2=1.05$), the tangential velocity in the back gap is weakened especially at the radially outer region. The reason might be explained as follows. In the back side gap of the impeller of the second stage, the net leakage flow is radially outward. However, due to the strong impeller-diffuser interaction, the some part of the main flow from the impeller enters the back side gap, which brings the angular momentum given by the impeller into the back side gap. By cutting the impeller diameter, the angular momentum of impeller exiting flow is decreased as seen in figure 15 (b), resulting in the reduced rotation of flow near the exit of the back side gap.
4. Conclusion
In this study, unsteady RANS analysis of three-stage centrifugal pump are carried out in order to investigate the effects of impeller-diffuser gap in the thrust forces acting on the rotor system. The primary results of this study can be summarized as follows.

1. The computed head and efficiency roughly agreed with measured one at both of the design and part load (15% of design) flow in the original impeller-diffuser gap case. However, the balancing flow rate was under-predicted for the design flow and overestimated at the part load flow by the present simulation. This perhaps leads to the small deviation of predicted head and efficiency from the measured ones.

2. The computed total axial thrust force was quantitatively over-predicted even at the design flow rate. The reason for this seemed to be the under-prediction of tangential velocity in the back side gaps of the first and second stage impellers, which leads to the over-prediction of the axial forces produced by these impellers.

3. The effect of impeller-diffuser gap on the axial thrust force was observed especially at the partial flow rate. The flow distributions in the front side gap of the second impeller was slightly different between two impeller-diffuser gap cases. However, this small difference seemed to have little effect on the axial thrust produced by the impeller. The reduced anti-axial thrust force produced by the balance drum seemed to be the main reason for the difference of the total axial force by the impeller diameter cut.

Acknowledgement
This study has been carried out under the activity of WG2 in HPC (High Performance Computing) Research Committee of Turbomachinery Society of Japan (TSJ). The authors would like to express sincere gratitude to the members of WG2 for fruitful discussions throughout WG2 meetings. The hydraulic design of the test pump was made by Miyagawa Lab. of Waseda University, Japan, and the staffs/students involved are highly appreciated for their contributions. The computational grid of base model used in the present study was provided by Dr. Hiroyoshi Watanabe of Ebara Corporation. Finally, we would like to thank to TSJ for the organization of the research committee and to DMW Corporation, Ebara Corporation, Hitachi, Ltd. and Shin Nippon Machinery Co. Ltd. for their financial supports.

References
[1] Yamashita T Watanabe S Hara Y Watanabe H and Miyagawa K 2015 Measurements of Axial and Radial Thrust Forces Working on a Three-Stage Centrifugal Pump Rotor. Proc. the ASME-JSME-KSME Joint Fluids Eng. Conf. 2015 AJKFluids2015-33515
[2] Furukawa D Yamashita T Watanabe S Hara Y Watanabe H and Miyagawa K 2015 Axial and Radial Thrust Characteristics of a Three-Stage Centrifugal Pump with Axial Offset of Rotors Proc. 13th Asian Int. Conf. on Fluid Machinery AICFM13-057
[3] Takamine T Furukawa D Watanabe S Watanabe H and Miyagawa K 2018 Experimental Analysis of Diffuser Stall in a Three-Stage Centrifugal Pump International Journal of Fluid Machinery and Systems (IJFMS) Vol.11 No.1 pp.77-84
[4] Arndt N Acosta A J Brennen C E and Caughey T K 1990 Experimental Investigation of Rotor-Stator Interaction in a Centrifugal Pump with Several Vaned Diffusers ASME J. Turbomac. Vol.112 No.1 pp. 98-108
[5] Sano T Nakamura Y Yoshida Y and Tsujimoto Y 2002 Alternate Blade Stall and Rotating Stall in a Vaned Diffuser JSME Int. J. Series B Vol.45 No.4 pp. 810-819
[6] Shi F Tsukamoto H Numerical Study of Pressure Fluctuations Caused by Impeller-Diffuser Interaction in a Diffuser Pump Stage 2001 J. Fluids Eng Vol.123 No.3 pp.466-474
[7] Karanth V K and Sharma Y N CFD Analysis on the Effect of Radial Gap on Impeller-Diffuser Flow Interaction as well as on the Flow Characteristics of a Centrifugal Fan 2009 Int. J. Rotating mac. Vol.2009 293508
[8] Shibata A Hiramatsu H Komami S Miyagawa K Meda M Kamei S Sano T and Iino M 2016 Study
of Flow Instability in Off Design Operation of a Multistage Centrifugal Pump J. Mech. Sci.
and Tech. Vol. 30 No. 2 pp 493-498

[9] Arnulfi G L, Micheli D and Pinamonti P 1996 Experimental Investigation on Rotating Stall in a
Centrifugal Blower with Two and Four Stages and Vaneless Diffusers ASME 1996 Int. Gas
Turbine and Aeroengine Congress and Exhibition Paper No.96-GT-171

[10] Hiradate K, Sakamoto K, Shinkawa Y, Joukou S and Uchiyama T 2016 Experimental Investigation
on Unstable Flow in Multi-stage Centrifugal Blower with Inlet Guide Vane J. Fluid Science
and Tech. Vol. 11 No. 2 Paper No. 15-00566

[11] Gulich J F 2008 Centrifugal Pumps Second Edition Springer-Verlag Chapter 2. ISBN: 978-3-
642-12823-3

[12] Software Cradle Co. Ltd. 2016 User’s Guid Basic of CFD Analysis SC/Tetra Version13

[13] Gulich J F 2008 Centrifugal Pumps Second Edition Springer-Verlag Chapter 9. ISBN: 978-3-
642-12823-3