Flow losses analysis in a mixed flow pump with annular volute by entropy production evaluation

Jian Cao, Ji Pei*, Yandong Gu, Wenjie Wang, Shouqi Yuan
Jiangsu University, National Research Center of Pumps, China

Corresponding author’s e-mail address: jpei@ujs.edu.cn

Abstract. Mixed flow pumps with annular volute are widely used in thermal and nuclear power plants for transporting or circulating the coolant, and require a high amount of power for which a large amount of electricity is required. Thus, improving its efficiency is absolutely important. The flow losses in the mixed flow pump with annular volute influence the hydraulic efficiency directly and are investigated in detail using computational fluid dynamics (CFD) in this paper. The Reynolds-averaged Navier–Stokes equation, coupled with the SST k-ω turbulent model, is employed to simulate the unsteady turbulent flow using ANSYS CFX. The numerical simulation results have been verified with the experimental measurements. The entropy production evaluation using CFD is applied to reveal the internal loss distribution and the amount of power losses. The results show that the power losses produced by turbulence dissipation are dominant, and the power losses caused by viscous dissipation can be neglected in each component. The turbulence dissipation losses in the annular volute are much larger compared to the other components, and a major loss distribution can be observed around the diffuser outlet and the shoulder between the circular flow channel and the discharge pipe, which is caused by the flow impact and separation. Therefore, the annular volute will be the key component for optimization and has the potential to reduce losses. This study can help understand the flow loss mechanisms and can provide theoretical guidance for improving the hydraulic efficiency of mixed flow pumps.

1. Introduction
Mixed flow pumps with annular volute are widely used in thermal nuclear power plants for providing forced coolant to remove and transfer the heat generated by fuels. Generally, the whole system of the mixed flow pump is very powerful, and it can consume a significant amount of electricity. Therefore, to reduce the energy consumption, the high efficiency is strictly considered an important performance parameter for the mixed flow pump with an annular volute.

The hydraulic efficiency is the key factor influencing the total efficiency of the pump, which is strongly related to the internal fluid flow conditions. The shapes of hydraulic components, influencing the hydraulic efficiency, can result in specific flow structures and flow loss behaviors. Previously, researchers have conducted some studies involving hydraulic performance optimization and unsteady flow behaviors in mixed flow pumps. Xiao et al.[1] employed an inverse design method and then selected the blade load as the design variable to improve the hydraulic efficiency and keep the head constant for a mixed flow pump. Kim et al.[2-4] proposed an optimization method, based on numerical calculations, to obtain the best geometric parameter combination of an impeller or a vane diffuser for improving the efficiency of a mixed flow pump. Long et al.[5] conducted an orthogonal test with seven
factors of diffuser geometric parameters and three levels of values to improve the performance of a reactor coolant pump, which is also known as a mixed flow pump. Zhou et al.\[6\] numerically investigated the effects of different staggered blades in a mixed flow pump used to cool the nuclear reactor. The results show that the staggered blades cannot improve hydraulic performance; however, they reduce the axial thrust and pressure fluctuation. Bing et al.\[7\] experimentally found that the rotating speed influenced the cavitation greatly at the part load condition. Safikhani et al.\[9\] applied a multi-objective optimization, aimed at increasing the efficiency and decreasing the necessary net positive suction head (NPSHr), in a centrifugal pump based on Pareto optimal solution. Siddique et al.\[10\] numerically analyzed the effects of rotating speed on the cavitation performance and flow pattern at the inlet in the nuclear reactor coolant pump with an annular volute, and pointed out that the rotating speed influenced the cavitation greatly at the part load condition. It can be seen that, compared to the traditional loss calculation method through which the loss distribution cannot be located precisely, the entropy production theory based on computational fluid dynamics (CFD) results is effective in revealing the loss distribution and evaluating the total amount of hydraulic losses.

In this study, the power of the mixed flow pump used in nuclear power plants is substantially high, consuming a large amount of electricity; thus, improving the efficiency is absolutely critical. In addition, some special structural components designed for security considerations, such as the annular casing rather than the spiral casing, restrict the improvement of efficiency. Consequently, it is of great importance to obtain the detailed loss distribution and propose a basic theory on reducing the flow losses in the premise of operation security. Because the power consumption and test rig of the mixed flow pump are significantly large during the design stage, the experimental investigation cannot be conducted. Hence, the analysis of internal flow cannot help improve the hydraulic efficiency and relate the hydraulic losses directly.

To reveal the flow loss distribution and mechanism clearly in each component in the mixed flow pump with annular volute, it is effective to apply the entropy production evaluation theory. Three-dimensional, unsteady numerical simulation of entire flow channels at different operating conditions is performed. Then, a comparison between simulations and experiments is conducted to verify the accuracy of the numerical calculations. Finally, the internal flow loss distribution and the amount of power losses are evaluated with the help of the entropy.

2. Entropy Production Theory

The working medium is generally considered to be incompressible in the hydraulic machinery, and the thermal transmission is negligible. Consequently, the flow losses caused by dissipation mainly derive from two factors: viscous dissipation and turbulence dissipation. In this study, the flow losses are evaluated using the entropy production theory\[19\] from the viewpoint of thermodynamics; the time-averaged entropy production rate \( \frac{\Phi}{T} \) is calculated indirectly by solving the Reynolds-averaged Navier–Stokes (RANS) equation. This term is composed as follows:
\[ \Phi \frac{(\cdots)}{T} = S_{\epsilon_{vr}}^{\nu} + S_{\epsilon_{vr}}^{\sigma} \quad (1) \]

Here, \( S_{\epsilon_{vr}}^{\nu} \) represents the entropy production rate derived from viscous dissipation and can be calculated directly in the CFD post-processing by using the following equation:

\[ \dot{S}_{\epsilon_{vr}}^{\nu} = \frac{\mu}{T} \left( 2 \cdot \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right) + \left( \frac{\partial u}{\partial x} \cdot \frac{\partial v}{\partial y} \right) + \left( \frac{\partial u}{\partial x} \cdot \frac{\partial w}{\partial z} \right) + \left( \frac{\partial v}{\partial y} \cdot \frac{\partial w}{\partial z} \right) \right) \quad (2) \]

Where \( \bar{u}, \bar{v}, \text{and} \bar{w} \) denote the time-averaged velocity components, \( \mu \) is the dynamic viscosity, \( T \) is the temperature. \( S_{\epsilon_{vr}}^{\sigma} \) represents the entropy production rate caused by turbulence dissipation, which is also called indirect dissipation. It can be written as the following equation:

\[ S_{\epsilon_{vr}}^{\sigma} = \frac{\mu}{T} \left( 2 \cdot \left( \frac{\partial u'}{\partial x} \right)^2 + \left( \frac{\partial v'}{\partial y} \right)^2 + \left( \frac{\partial w'}{\partial z} \right)^2 \right) + \left( \frac{\partial u'}{\partial x} \cdot \frac{\partial v'}{\partial y} \right) + \left( \frac{\partial u'}{\partial x} \cdot \frac{\partial w'}{\partial z} \right) + \left( \frac{\partial v'}{\partial y} \cdot \frac{\partial w'}{\partial z} \right) \right) \quad (3) \]

\( u', v', \text{and} w' \) represent the velocity fluctuation components. The expression cannot be calculated directly because the fluctuating variables are not available after the RANS equation is solved. To solve this problem, Kock\textsuperscript{[11,12]} proposed the following equation:

\[ \dot{S}_{\epsilon_{vr}}^{\sigma} = \frac{\rho \varepsilon}{T} \quad (4) \]

where \( \varepsilon \) represents the dissipation rate of turbulent kinetic energy, which is readily available during the post-processing.

According to Kock\textsuperscript{[11,12]}, the entropy production rates produced by viscous and turbulent effects, showed in Eqs. (2)–(4), are connected to the viscous and turbulence dissipation respectively through the following relations:

\[ \Phi_{\dot{\epsilon}_{vr}} = T \dot{S}_{\epsilon_{vr}}^{\nu} \quad (5) \]
\[ \Phi_{\dot{\epsilon}_{vr}} = T \dot{S}_{\epsilon_{vr}}^{\sigma} \quad (6) \]

Finally, the power losses produced by the viscous and turbulent effects can be calculated by integrating the dissipation rates over the domains, as shown in the following equations:

\[ P_{\epsilon_{vr}} = \int \Phi_{\dot{\epsilon}_{vr}} dV = \int T \dot{S}_{\epsilon_{vr}}^{\nu} dV \quad (7) \]
\[ P_{\epsilon_{vr}} = \int \Phi_{\dot{\epsilon}_{vr}} dV = \int T \dot{S}_{\epsilon_{vr}}^{\sigma} dV \quad (8) \]

3. Numerical Modeling

3.1. Prototype Model and Calculation Grids

Table 1 lists the main geometric parameters of the mixed flow pump. The rotating speed is \( n=1480\text{rpm} \) and the specific speed \( n_s = 3.65nQ^{0.5}/H^{0.75} \) is 386 at the best efficiency point (BEP).

| Parameters                              | Value   |
|-----------------------------------------|---------|
| Impeller inlet diameter, \( D_1 \) (mm) | 717.5   |
| Impeller outlet diameter, \( D_2 \) (mm)| 804     |
| Impeller outlet width, \( b_2 \) (mm)   | 212     |
| Number of impeller blades, \( Z_i \)    | 4       |
| Diffuser inlet width, \( b_1 \) (mm)    | 241     |
| Diffuser outlet width, \( b_4 \) (mm)   | 252     |
| Number of diffuser blades, \( Z_d \)    | 11      |
| Annular volute inlet width, \( b_3 \) (mm)| 356  |
| Annular volute inlet diameter, \( D_3 \) (mm)| 1340 |
As shown in Figure 1, the fluid domain of the entire flow passage is composed of the suction pipe, closed impeller, diffuser and annular volute. ICEM-CFD is applied to generate the structural grids for each component, and Figure 2 shows the detailed grids. The grid independence is analyzed with 9 different grid numbers, and approximately 10.7 million grids are selected until the head remained almost constant with the increasing the numbers, as shown in Figure 3. The maximum y+ value of the final scheme is around 150. Additionally, Figure 4 shows the definitions of the circumferential position and cross sections for analyzing the results in section 4. One cross section on the x-y plane of annular volute is marked VCS1. VCS2 is just near the inlet of the discharge pipe. VCS3 and VCS4 are circular surfaces in the annular volute, whose radii are 680 mm and 810 mm, respectively.

3.2. Boundary Condition Settings

The RANS equation is solved with the SST $k$-$\omega$ turbulent model using ANSYS CFX. The mass flow rate is employed in the outlet. The total pressure and turbulence intensity of 5% are imposed in the inlet. To determine the transient flow caused by rotor-stator interaction, the interface between the impeller and diffuser is set to be “transient rotor-stator”. Hence, their relative positions are refreshed for each time step. The time step for the unsteady simulation is $3.38 \times 10^{-4}$ s for a nominal rotating speed, which corresponds to a rotating angle of $\Delta \phi = 3^\circ$. Within each time step, the iteration stops when the maximum residual is less than $10^{-4}$; the convergence criterion for the transient problem is that the flow has reached its stable periodicity. The total calculation time is 0.41 s, which corresponds to 10 revolutions. Finally, all the variables used in Equations 2 and 4 for solving entropy productions are averaged over one revolution in the CFD-post process.
3.3. Experimental Validation
The accuracy of the numerical simulation results is validated by comparing them with the experimental pump performance results. Figure 5 shows the open test rig of the model mixed flow pump, whose test uncertainty is less than 1%. The model pump is manufactured instead of the prototype, and the linear ratio of the prototype to the model is 5.74. The static pressure sensor measures the pressure at the inlet and outlet, and the error level is 0.1%. The flow rate is measured by the turbine flow meter with a 0.5% measurement error. The shaft torque is measured by a torque meter with a 0.3% measurement uncertainty. For comparing the performances of the model pump and the prototype at the same scale, the flow rate coefficient $\phi$ and head coefficient $\psi$ are used to non-dimensionalize the performance, as shown in the following equations:

\[
\phi = \frac{4Q}{\pi^2 n D^3} \tag{9}
\]

\[
\psi = \frac{2gH}{\pi^2 D^2 n^2} \tag{10}
\]
Here, $Q$ is the flow rate, $n$ is the rotating speed of the impeller, $D_2$ is the outer diameter of the impeller, and $H$ is the head. Figure 6 presents the comparison of the performance curves between the numerical and experimental results. It is clear that both the simulated head and hydraulic efficiency curves have an acceptable agreement with experimental ones. The head and hydraulic efficiency deviations at the BEP (dimensionless flow rate is 0.19) are 4.08% and 4.6%, respectively. Overall, the maximum relative error is less than 5%. Therefore, the numerical simulation results are reasonable and can be used to perform detailed analyses.

4. Results and Discussions

4.1. Analysis of Power Losses

In Figures 7 and 8, the dissipation losses are normalized by the power of turbulent dissipation in the annular casing at the overload. Figure 7 shows the power losses produced by viscous dissipation at three different operating conditions. The power losses in the impeller caused by viscous dissipation are the highest while the power losses in the suction pipe are the least at each working condition. Although the volume of the annular volute is much larger than the other domains, its viscous dissipation power losses are not very high, standing in the middle position, because the Reynolds number in the impeller is high and the gradient of the time-averaged velocity is relatively large according to Eq. (2). In addition, the viscous dissipation power losses both in the casing and suction pipe increase with the rise in flow rate. On the other hand, such losses in the impeller and diffuser are lowest at the BEP.
However, it is obvious that the power losses produced by turbulence dissipation are much larger than the power losses caused by viscous dissipation, as shown in Figure 8. The flow rate has important effects on the turbulence dissipation power losses in each domain except for the suction pipe, whose power losses are the smallest. As the flow rate rises, the turbulence dissipation power losses in the annular casing increase rapidly. In contrast, the losses in the suction pipe increase slightly. The losses in the impeller and diffuser reach their minimum at the BEP; the losses are reduced by nearly two times from the part load condition to the overload condition. In the impeller, diffuser and annular volute, the losses caused by turbulence dissipation are almost the same at the part load condition, but the losses in the annular volute are much larger than the other two domains at the BEP and overload conditions. Because the annular volute is responsible for a large portion of the losses, more optimization should be done with respect to annular volute rather than the impeller or diffuser in the mixed flow pump; this helps improve the hydraulic efficiency significantly. Additionally, the turbulence dissipation distribution of the annular volute is the main focus of the analysis in the next section.

4.2. Analysis of Loss Distributions
To determine the locations where relatively high losses occur and why the losses in the annular volute are higher than the other domains, both the loss distribution contours and the flow patterns in different cross sections through the mixed flow pump are determined.

Figure 9 depicts the turbulence dissipation $\Phi_\rho$ distribution in the blade to blade view of the impeller and diffuser. The highest turbulence dissipation can be observed near the blade surface for both the impeller and diffuser. At the part load condition, the area of strong turbulence dissipation in the impeller, from $5.57 \times 10^5$ W·m$^{-3}$ to $6.00 \times 10^5$ W·m$^{-3}$, is the largest on the suction side (SS); however, on the pressure side (PS), it is the smallest in comparison with the other working conditions. As the flow rate rises, the area of strong turbulence dissipation diminishes on the SS and enlarges on the PS; therefore, the largest area of strong turbulence dissipation on the PS occurs at the overload condition. In Figure 10, flow angle is changing at the blade inlet from the part load condition to the overload condition, inducing the different flow separations on the two blade sides; this is the main
reason that the area of high turbulence dissipation varies on the blade surface. Further, the flow angle of the fluid moving out from the impeller, changing with the flow rate, leads to a different area of high turbulence dissipation on the blade surface of the diffuser. Additionally, the turbulent dissipation at the nominal condition in the diffuser seems larger than the overload condition, which is contradictory to Figure 8. However, the power loss calculated by Eq. (8) is a 3D integral of the turbulent dissipation calculated by Eq. (6), thus it is reasonable to understand the difference between Figures 8 and 9.

Figure 9. Blade to blade view of turbulence dissipation on mid-span at different working conditions

Figure 10. Velocity distribution around blade inlet at different working conditions
Figure 11. Contours on VCS1 at different working conditions

(a). Turbulence dissipation

(b). Velocity
Figures 11(a) and 12(a) show the turbulence dissipation $\Phi_\alpha$ distribution on VCS1 and VCS2 respectively. With the flow rate increasing, both the value and gradient of turbulence dissipation increase significantly. At each operating condition, a large turbulence dissipation surrounding the diffuser outlet, particularly peaking at A2, can be observed; this is because the fluid out from the diffuser affects the main flow of the annular volute, as shown in Figure 11(b). At the shoulder (A1 and A3) between the circular channel and the discharge pipe, high losses can be observed locally, shown in Figures. 11(a) and 12(a). The flow rate influences the turbulence dissipation evidently at A1 but insignificantly at A3. Figure 11(b) illustrates the fact that flow impact and flow separation appears at A1 and A3, respectively. Particularly at the overload condition, a fluid with high velocity crashes into shoulder A1 and then rebounds against the fluid coming out from the diffuser (showed with red arrows in Figure 11(b)); this leads to the highest losses as well as a weak velocity region at A1. Figure 12(b) shows the vortex region A4 varying with the flow rate, which results in a different loss area, as shown in Figure 12(a).

Figures 13 depicts the turbulence dissipation $\Phi_\gamma$ distribution on VCS3 and VCS4. The radius of VCS3 is 680 mm, encircling the diffuser outlet closely, and the radius of VCS4 is 810 mm. These two circular surfaces are used to show the loss distribution along the Z-axis direction. On VCS3, the turbulence dissipation is concentrated on a circular ring whose width is $b_5$ (the width of the inlet of the annular volute), and the loss is small in the other area. At the part load condition, the loss is inhomogeneous, irregular, and slightly larger than the other conditions but reduces rapidly on VCS4. It indicates that the loss region near the diffuser outlet is affected by the position of the diffuser flow channel. At the BEP and overload conditions on VCS3, the turbulence dissipation loss peaks can be observed at the edge of the diffuser shrouds. On VCS4, large a turbulence dissipation occurs in the circumferential range of $90^\circ$ to $180^\circ$, which corresponds to A2 shown in Figure 11(a). When compared to VCS3, high turbulence dissipation on VCS4 increases along the Z-axis with a rising flow rate.

**Figure 12.** Contours on VCS2 at different working conditions

**Figure 13.** Contours of turbulence dissipation on VCS3 and VCS4 at different working conditions
5. CONCLUSIONS
In this study, the entropy production theory is applied to analyze the flow losses in the mixed flow pump based on numerical simulation; the accuracy of the numerical results has also been verified by experimental tests. The mass flow rate definitely has a noticeable effect on the internal flow losses; thus, three typical working conditions are considered in detail. The following conclusions can be drawn from these studies:

1. The power losses produced by turbulence dissipation are dominant in all calculation domains and for all flow rate working conditions; the power losses caused by viscous dissipation can be neglected.

2. In the impeller and diffuser, the power losses produced by turbulence dissipation reach a minimum at the BEP point. The power losses in the annular volute are the highest among all hydraulic components in the mixed flow pump; they increase significantly with a rising flow rate. Therefore, the annular volute can be considered as the potential, key component for optimization.

3. In the impeller, as the flow angle varies with an increasing flow rate, large loss distributions diminish on the SS and become enlarged on the PS. Moreover, in the annular volute, the major loss distribution can be observed around the diffuser outlet because the fluid coming out from the diffuser crashes into the main flow. Furthermore, high losses occur at the shoulder (A1 and A3) between the circular flow channel and the discharge pipe owing to flow impact and flow separation, respectively. Therefore, by optimizing the shoulder shape, it may be possible to reduce the losses in the volute.

6. ACKNOWLEDGEMENTS
This study was supported by National Natural Science Foundation of China (Grant No. 51409123), Natural Science Foundation of Jiangsu Province (Grant No. BK20140554), Training Project for Young Core Teacher of Jiangsu University.

References
[1] Xiao, R.F., Tao, R., Wang, W.W., Yang, W.(2014). Inverse design and hydraulic optimization of mixed-flow pump impeller. Transactions of the Chinese Society for Agricultural Machinery, vol. 45, no. 9, p. 84-88, DOI:10.6041/j.issn.1000-1298.2014.09.014.
[2] Kim, J. H., Kim, K. Y. (2011). Optimization of vane diffuser in a mixed-flow pump for high efficiency design. International Journal of Fluid Machinery and Systems, vol. 4, no. 1, p. 172-178, DOI:10.5293/IJFMS.2011.4.1.172.
[3] Kim J. H., Kim K. Y. (2012). Analysis and optimization of a vaned diffuser in a mixed flow pump to improve hydrodynamic performance. Journal of Fluids Engineering, vol. 134, no. 7, DOI: 10.1115/1.4006820.
[4] Heo, M. W., Kim, K. Y., Kim, J. H., Choi, Y. S. (2016). High-efficiency design of a mixed-flow pump using a surrogate model. Journal of Mechanical Science and Technology, vol. 30, no. 2, p. 541-547, DOI: 10.1007/s12206-016-0107-8.
[5] Long, Y., Zhu, R.S., Wang, D.Z., Yin, J.L., Y., Li, T.B. (2016). Numerical and experimental investigation on the diffuser optimization of a reactor coolant pump with orthogonal test approach. Journal of Mechanical Science and Technology, vol. 30, no. 11, p. 4941-4948, DOI: 10.1007/s12206-016-1014-8.
[6] Zhou, F. M., Wang, X. F. (2016). Effects of staggered blades on the hydraulic characteristics of a 1400-MW canned nuclear coolant pump. Advances in Mechanical Engineering, vol. 8, no. 8, p. 1-21, DOI: 10.1177/1687814016657944.
[7] Bing, H., Cao, S. (2014). Experimental study of the influence of flow passage subtle variation on mixed-flow pump performance. Chinese Journal of Mechanical Engineering, vol. 27, no. 3, p. 615-621, DOI: 10.3901/CJME.2014.03.615.
[8] Fu, Q., Zhang B.Y., Zhu R.S., Cao, L. (2016). Effect of rotating speed on cavitation performance of nuclear reactor coolant pump. Journal of Drainage and Irrigation Machinery Engineering, vol. 34, no. 8, p. 651-656, DOI: 10.3969/j.issn.1674—8530.15.0170.
[9] Safikhani, H., Khalkhali, A., Farajpoor, M. (2011). Pareto based multi-objective optimization of centrifugal pumps using CFD, neural networks and genetic algorithms. Engineering Applications of Computational Fluid Mechanics, vol. 5, no. 1, p. 37-48, DOI: 10.1080/19942060.2011.11015351.

[10] Siddique, M. H., Mrinal, K. R., Samad, A. (2016). Optimization of a centrifugal pump impeller by controlling blade profile parameters. ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, DOI: 10.1115/GT2016-56604.

[11] Kock, F., Herwig, H. (2005). Entropy production calculation for turbulent shear flows and their implementation in CFD codes. International Journal of Heat and Fluid Flow, vol. 26, no. 4, p. 672-680, DOI:10.1016/j.ijheatfluidflow.2005.03.005.

[12] Kock, F., Herwig, H. (2004). Local entropy production in turbulent shear flows: a high-Reynolds number model with wall functions. International Journal of Heat and Mass Transfer, vol. 47, no. 10, p. 2205-2215, DOI: 10.1016/j.ijheatmasstransfer.2003.11.025.

[13] Herwig, H., Kock, F. (2007). Direct and indirect methods of calculating entropy generation rates in turbulent convective heat transfer problems. Heat and mass transfer, vol. 43, no. 3, p. 207-215, DOI:10.1007/s00231-006-0086-x.

[14] Herwig, H., Kock, F. (2006). Local entropy production in turbulent shear flows: A tool for evaluating heat transfer performance. Journal of Thermal Science, vol. 15, no. 2, p.159-167, DOI:10.1007/s11630-006-0159-7.

[15] Li, X.J., Zhu, Z.C., Li, Y., Chen, X.P. (2016). Experimental and numerical investigations of head-flow curve instability of a single-stage centrifugal pump with volute casing. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, vol. 230, no. 7, p. 633-647, DOI: 10.1177/0957650916663326.

[16] Böehle, M., Fleder, A., Mohr, M. (2016). Study of the losses in fluid machinery with the help of entropy. International Symposium on Transport Phenomena and Dynamics of Rotating Machinery.

[17] Li, D.Y., Gong, R.Z., Wang, H.J., Xiang, G.M., Wei, X.Z., Qin, D.Q. (2016). Entropy production analysis for hump characteristics of a pump turbine model. Chinese Journal of Mechanical Engineering, vol. 29, no. 4, p. 803-812, DOI: 10.3901/CJME.2016.0414.052.

[18] Gong, R.Z., Wang, H.J., Chen, L.X., Li, D.Y., Zhang, H.C., Wei, X.Z. (2013). Application of entropy production theory to hydro-turbine hydraulic analysis. Science China Technological Sciences, vol. 56, no. 7, p. 1636-1643, DOI: 10.1007/s11431-013-5229-y.

[19] Spurk, J. H., Aksel, N. (2008). Fluid Mechanics. Springer, Verlag Berlin Heidelberg, DOI: 10.1007/978-3-540-73537-3.