CFD based Improvement of Thai Irrigation Pump

Kittipass Wasinarom[1], Dachdanai Boonchauy[2] and Jarruwat Charoensuk[3]

[1]*School of International and Interdisciplinary Engineering Programs Faculty of Engineering, King Mongkut's Institute of Technology Ladkrabang Bangkok, Thailand, kittipass.wa@kmitl.ac.th*

[2]*Seagate Technology (Thailand) Ltd. Teparak, Samutprakarn ,Thailand, dachdanai.boonchauy@seagate.com*

[3]*Department of Mechanical Engineering Faculty of Engineering, King Mongkut's Institute of Technology Ladkrabang Bangkok, Thailand, jarruwat.ch@kmitl.ac.th*

Abstract.

Evaluation phase which was partial fulfilment of the beginning phase of "Development of Performance test rig and Efficiency improvement of impeller in Thai irrigation pump project" is presented in this paper. Overall flow field in the pump system that consisted of inlet, impeller and stator vane of the available pump was analyzed using commercial Computational Fluid Dynamics (CFD) code. The goal of this investigation is to obtain more understanding of energy dissipation which results from shear stress that developed within the flow field in each section of the pump. The improvement measure is then conducted with the concern of manufacturing difficulties. High dissipation flow structure was observed around the impeller outlet. Jet-wake and recirculation flow were observed. The first improvement measure was conducted by adding the bluff body in the flow channel to alleviate jet-wake structure and delay flow separation. After the implementation of the optimized bluff body around the impeller exit, CFD results indicated around 3-8% improvement compared with the CFD results of the available pump for the entire range of operating conditions.

Keywords. CFD, turbo-machine, pump, improvement.

1. INTRODUCTION

The irrigation pump is a small portable pump unit (Figure1). It is typically developed by local technicians. The improvement focused on the increasing volume flow rate rather than the energy efficiency. Therefore, generally the energy efficiency of this kind of pump is very low. The test of the available pump shows that the obtained peak efficiency was around 40% [1] which there is high potential to conduct energy efficiency improvement projects. There are a lot of different conditions between development of Thai irrigation pumps and the industrial pumps. Irrigation pumps are forced to use simple manufacturing techniques because of its low capital cost. Domestic human resources are used in this development rather than the world's leading turbomachines

In this project progress, CFD simulation of the flow inside the pump system was conducted. Energy dissipation which results from shear stress that developed within the flow field of each section of the baseline pump was analyzed in order to point out the improvement opportunity in specific sections. The results showed that there was the jet-wake, recirculation and flow separation at the impeller outlet area which were considered high internal dissipation flow structure at the impeller outlet. The first improvement measure was implemented by adding the bluff body profile at the impeller outlet (behind the impeller). By this action, it was expected to alleviate jet-wake structure and delay the separation flow. After performing CFD simulation of the improved version, it was found that recirculation disappeared and the jet-wake flow structure was significantly reduced.

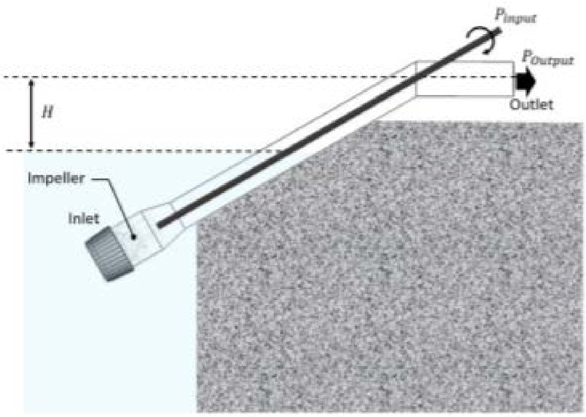

Figure 1 Thai irrigation pumps used in flooding application

2. METHODOLOGY

Energy dissipation is the process that leads to the conversion of useful mechanical energy (flow energy) to non-useful heat energy (internal energy). It can be described by the first law of thermodynamics for SteadyState, Steady Flow Process (SSSF). The equation can be developed for turbo-machine equipment as shown in (4). The isentropic process which is the ideal case, it considers non-dissipation occurs between inlet and outlet of turbomachine. The process occurs following the line that starts at point 1 to point 2s as seen in Figure 3. The energy efficiency is 100 percent in the isentropic process. In the other words, all of the input mechanical shaft energy can be converted to useful flow energy at outlet (state2). While in actual situations, some input shaft energy is converted to heat by fluid friction or so called energy dissipation which resulted in less useful flow energy at outlet regarding to the equation (4). This process is also known as internal irreversibility since the dissipation occurs inside the fluid medium itself.

The efficiency can be calculated as the equation (1)

$$n = \frac{p_{output}}{p_{input}} \tag{2.1}$$

Where η is the calculated efficiency, Poutput is the flow energy at the pump outlet which can be calculated as the equation (2)

$$P_{output} = p \times g \times Q \times H \tag{2.2}$$

Where ρ is water density, g is the gravitational acceleration, Q is volume low rate and H is lifted head as indicated by Figure 1.

P_{input} can be calculated by equation (3)

$$P_{input} = Tshaft \times \omega \tag{2.3}$$

P_{input} is the shaft mechanical energy. ω is the shaft rotational speed and *Tshaft* is the torque.

$$P_{output} = P_{input} + Internal\ dissipation \tag{2.4}$$

Flow dissipation is generated by the presence of velocity gradient (shear layer) which has a specific magnitude in particular location of the flow domain. The developed shear stress is directly proportional to the velocity gradient in Newtonian fluids [5]. The most basic illustration of the situation is the laminar flow in a simple circular pipe. Shear stress is proportional to the velocity gradient magnitude which in this case, it depends on y location as shown in Figure 2. It has only shear stress in x direction in this case. Flow structure in a turbomachine is much more complicated than the flow within a circular pipe. The fluid elements usually possessed 3D of viscous shear force.

In order to conduct energy efficiency improvement activity in turbo machinery, it is necessary to be able to visualize detailed fluid dynamics within the flow passage. Therefore, viscous shear stress at specific locations in the system can be quantified or at least estimated. Then the improvement measure is implemented with the expectation of reduced shear flow. The optimization process may incorporate the validated CFD simulation code to minimize cost and time consumed and also allow high ability to analyze. Finally, the experimental test must be performed. Flow visualization technique is the key to this process. CFD is an interesting method to obtain detailed flow visualization. Many researchers employed CFD as a tool [2, 3, 4]. However, it has to calibrate and validate with the experimental result in order to justify the obtained flow field.

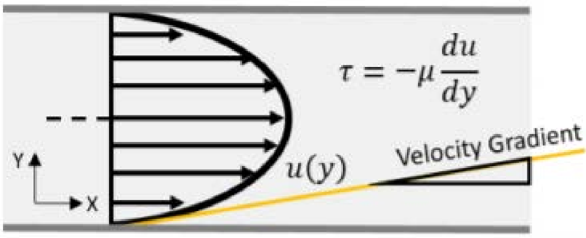

Figure 2 Viscous shear stress relates to velocity profile of the Newtonian fluids in circular pipe

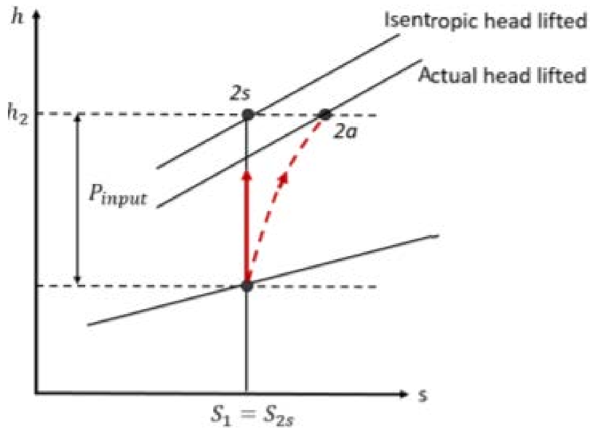

Figure 3 Example of Ideal (isentropic) and actual pump process of the same flow rate and input shaft energy

CFD has become an important tool to explore the flow field inside the flow channel of turbomachinery. The obtained results then analyzed by focusing on shear stress occurred in the fluid element at the particular location of flow domain

3. MODELING

Commercial CFD code was used in this work. The simulation of available Thai irrigation pumps was performed throughout the flow rate of 0.01 to 0.1 m^3/s while speed was kept constant at 900 rpm. CFD results were validated with experimental results from a test rig at King Mongkut's University of Technology Thonburi (KMUTT). The test rig was conforming to grade 1 according to the applicable JIS standard (JIS B 8301, 8302) [1]. Flow channel, impeller and stator vane of the available pump was reproduced by CAD program as shown in Fig. 4. The flow channel was divided into 8,000,000 control volumes. Then this geometry file was imported to the ANSYS CFX solver [6]. The flow model was RANS (Reynolds Averaged Navier Stokes) with SST turbulence model. The boundary condition and all necessary parameters were set regarding Table 1. Iterative solution procedure of inter-link between discretized momentum equation, continuity equation, turbulence kinetic energy and turbulence dissipation equation was executed. It continued until calculated normalized residual was below 10^{-4} Finally the last updated pressure and velocity field was considered as the simulation results.

Figure 4 CAD geometry of the irrigation pump system

Table 1 Boundary condition and input parameter used in CFD simulation

Analysis type	Steady Stat
Mesh	8,000,000 elements
Turbulence model	SST
Fluid domain	Water
Moving Reference Frame (MRF)	900 RPM
Inlet	Total pressure 1 atm
Outlet	Desire flow rate
Convergence residual	10^{-4}

Figure 5 Cross Section A-A of the pump system

4. RESULT

Model validation of the available Thai irrigation pump showed the acceptable agreement with the experimental results throughout the flow rate between 0.01 to 0.1 m^3/s which covered all operation range of the pump [1].

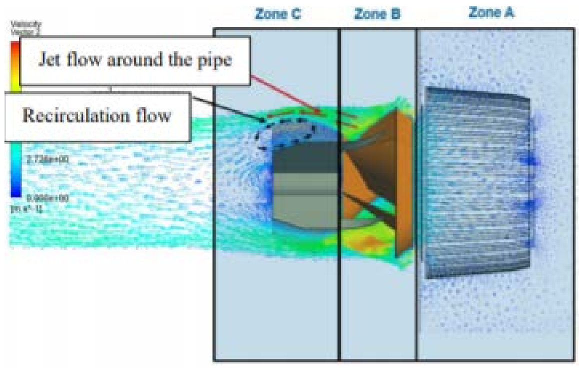

Figure 6 Jet flow and recirculation flow at the peak efficiency condition of available version

General flow structure around peak efficiency condition (0.05 m^3/s) was analyzed. The flow domain was divided into 3 sections as shown in Figure 6. They were composed of 1.) Inlet region 2.) Impeller and 3.) Behind impeller. All of the demonstrated flow fields in this paper were on the cross-section in A-A planar as shown in Figure 5. It was found that, at the inlet region, flow entered the impeller with considerable low velocity as indicated by blue velocity vector and the flow was nearly uniform with only low velocity gradient presented. The flow inside the impeller was highly complex but it was beyond scope of discussion in this paper. At section 3 (behind impeller), evidence of high shear flow was observed. High velocity flow at the impeller exit impinged on the pipe wall which resulted in the formation of high velocity-jet flow around the pipe rim, while the low velocity-wake flow was found at the inner core of section C as shown in Figure 6 Highly shear flow of this jet-wake structure will result in massive energy dissipation in this section.

First improvement measures focused on the area around the impeller exit (section 3 or Zone C in Figure 6 even though there was also considerable shear flow observed inside the impeller passage. This is because any change in impeller manufacturing is a complicated task. The impeller improvement will be performed in the later phases.

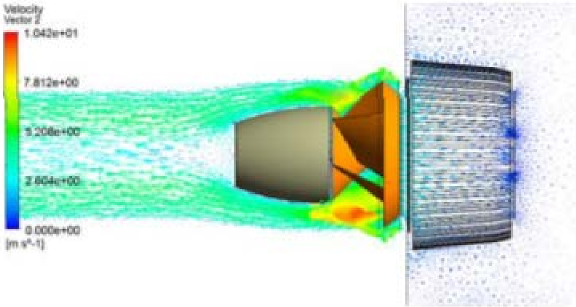

Figure 7 An improved version A

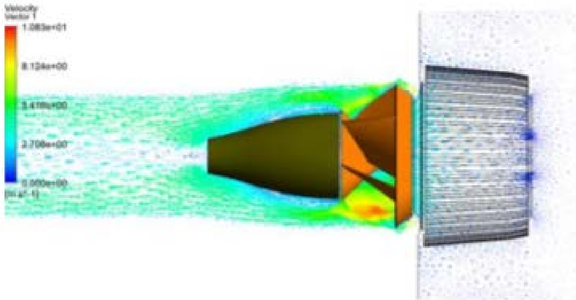

Figure 8 An improved version B

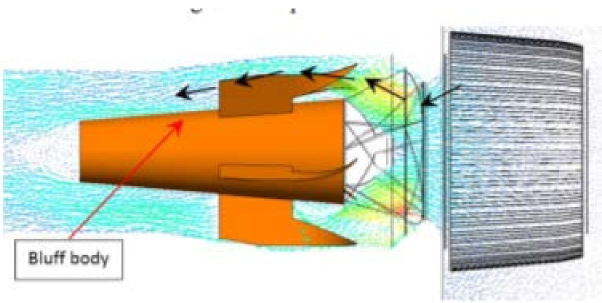

Figure 9 The final improved version

The first improvement measure is to add the bluff body profile behind the impeller. It proposed to prevent abrupt expansion of the flow cross-sectional area at the impeller exit. This expansion flow cross-sectional area will result in a wake region at the inner core behind the impeller as shown in Figure 6. Where both recirculation and separation flow were observed. Various profile shapes were trialed by CFD simulation (Figure 7 and 8). The profile shape was optimized in this process. It was improved by analyzing the velocity field which related to the cross-area of the flow channel that developed by the bluff body.

The profile of the bluff body should be allowed as low as possible of the averaged flow velocity while still absent of jet-wake flow or any flow separation. However, the curved profile was transformed to conical shape at the end (Figure 9) for the ease of manufacturing reason. After this action, CFD results indicated around 3-8% improvement compared with the CFD results of available pumps for the entire range of operating conditions as shown in Figure 10.

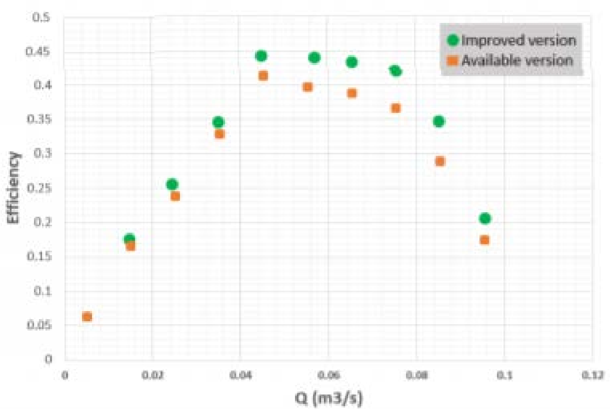

Figure 10 Efficiency of available pump and improved pump (bluff body installed) by CFD

5. CONCLUSION

Overall flow field in the available irrigation pump system that consisted of inlet, impeller and stator vane was analyzed using commercial Computational Fluid Dynamics (CFD) code. The first improvement measures have been conducted by adding the bluff body in the flow channel to alleviate jet-wake structure and delay flow separation. After the implementation of the optimized bluff body around the impeller exit, CFD results indicated around 3-8% improvement compared with the CFD results of the available pump for the entire range of operating conditions.

6. ACKNOWLEDGEMENT

This work was supported by Thailand National Metal and Materials Technology Center (MTEC) project code P1851268

7. REFERENCES

[1] “Development of performance test rig and efficiency improvement of impeller in Thai irrigation pump full report (P1851268),” National Science and Technology Development Agency, 2019, Chapter 3, pp. 37- 66.

[2] L. Ji, W. Li, W. Shi, H. Chang and Z. Yang. "Energy characteristics of mixed flow pump under different tip clearances based on entropy production analysis," Energy, vol.199, 2020, pp. 117447.

[3] A. Patil, S. Sundar, A. Delgado and J. Gamboa, "CFD based evaluation of conventional electrical submersible pump for high-speed application," Journal of Petroleum Science and Engineering, vol.182, 2019, pp. 106287.

[4] S. Kim, K. Y. Lee, J. H. Kim and Y. S. Choi, "A Numerical Study on the Improvement of Suction Performance and Hydraulic Efficiency for a Mixed-Flow Pump Impeller", Mathematical Problems in Engineering, vol.2014, Article ID 269483

[5] J. Charoensuk, "Computational Fluid Dynamics and its Application in Engineering Problems", Ladkrabang, 2018, Chapter 2, pp 25-87.

[6] ANSYS, ANSYS Academic Research, Release 19 R3, Help System, CFX-Documentation 2019. ANSYS, Inc.