Oral Presentation

The 7th TSME International Conference on Mechanical Engineering
13-16 December 2016

CST0005

CFD Analysis of Thai-made Axial Flow Low-lift Pump

Noppong Sritrakul[1*], and Teerasak Hudakorn[1]

[1] Department of Mechanical Engineering, Faculty of Engineering and Industrial Technology,
Silpakorn University Sanam Chandra Palace Campus, Amphoe Muang Nakhon Pathom,
Nakhon Pathom Province, Thailand, 73000
[*] Corresponding Author: n_pp_ng@yahoo.com, Telephone: +6634-257 218

Abstract

Thai-made axial flow low-lift pumps (Tor-Payanak) are widely used in the agriculture sector of Thailand. Simple design, easy to move and low maintenance cost of this type of pump are prominent causing widely use in farms, vegetable gardens, orchards, and aquaculture farms. Thai-made axial flow low-lift pumps are suited for pumping water at low lift level, about 1-4 meters, with high capacity. Various designs of these pumps make different performance. Design for better performance and efficiency of axial flow low-lift pump requires knowledge and understanding of the mechanism of flow that occurs within the pump. The advance in computer science and fluid dynamics simulation techniques that can calculate faster and more accurately is used to simulate fluid flow behavior within the pump. The application of computational fluid dynamics (CFD) analysis to improve pump performance leads to reduce cost and time of design procedure. The investigation of the fluid flow field inside the axial flow low-lift pump was based on the Reynolds time-averaged Navier-Stokes equations. The k-ε turbulent flow model with the multiple reference frames technique was used to analysis. A pump with pipe diameter of 6 inch and 20 feet long was chosen as a case study. The simulation and experimental results were compared. The study revealed that simulation results were in fair agreement with the experimental results. However, the simulation results tended to overestimate the pump performance. The simulation of fluid flow field by CFD analysis gives an understanding of flow behavior within pump that will be beneficial to improve performance and efficiency of Thai-made axial flow low-lift pumps. This technique can give correction and accurate result in order to reduce both difficulty and cost of pump design procedure.

***Keywords**:* axial flow pump, Thai-made axial flow pump, Tor-Payanak , pump performance, CFD

1. Introduction

Thai-made axial flow low-lift pumps (Debriddhi pump or Tor-Payanak) are widely used in the agriculture sector of Thailand. Simple design, easy to move and low maintenance cost of this type of pump are prominent. These pumps are widely used in farms, vegetable gardens, orchards, and aquaculture farms. The first Thai-made axial flow pump was designed and demonstrated in 1941 by the late M. R. Debriddhi Tavakul [1]. In 1955, Tavakul modified and produced the second design. The main pipe and the discharge pipe were made of sheet steel. The discharge pipe was attached to the main pipe at right angles. The drive shaft was totally inside the main pipe. The single impeller was attached to the suction end. A screener was attached to the inlet end. A small gasoline engine was the power source of pump. The second design was not commercially available until 1957. After that, the Thai-made axial flow pumps have contributed a lot to Thai agriculture. Many designs of pump were modified and adapted by manufacturers such as changing the attached angle and sizing of the main pipe, impeller shape, shaft speed, inlet portion, etc. Thai-made axial flow pumps are shown in Fig. 1 and various modified pump impellers are shown in Fig. 2.. Although more than one-million pumps have been fabricated and used in Thailand [2], but lack of engineering knowledge for pump design of Thai manufacturers caused low operation of both mechanical and hydraulic efficiency of pumps.

Fig. 1 Thai-made axial flow pump (Tor-Payanak).

Source: [13-14]

Fig. 2 Various type of Thai-made axial flow pump impeller and guide vane.

Source: [14-15]

There have been only a few researches on improvement and evaluation of Thai – made axial flow low-lift pumps. Kaewprakaisaengkul [3] studied on 5 aspects, i.e. i) testing on the local made axial flow, mixed flow and radial flow pumps; ii) testing on various mechanical losses of Thai - made pumps; iii) the study on external drive shaft configuration, which has effected on mechanical power loss; iv) the study on impeller, diffusion vanes and pump casing geometry that affect on hydraulic efficiency; and v) the study on the effect of surface coating on hydraulic efficiency. Furthermore, Kaewprakaisaengkul used 3D – CFD technique to predict cavitation phenomena, which occurred in Thai – made pumps. Another research by Kasantikul [4] investigated the details of flow phenomena in Thai – made pumps (Tor-Payanak) on computer with Ansys CFX Code and k-ω SST turbulence model. The CFD simulation result had correlation with experimental data and could give a good understanding of flow phenomena in pumps. However, CFD modeling demonstrated the over estimation result in all studied cases. Subsequently, Kasantikul and Laksitanonta [5] used Ansys CFX software to numerically analyze and study the flow phenomena and also to determine the effects of geometric factors on the Thai-made irrigation pump (Tor-Payanak) efficiency. The simulation results showed that the properly designed impeller could increase the performance of the pump.

Thai-made axial flow low-lift pumps are suited for pumping water at low lift, level about 1-4 meters, with high capacity [6]. Various designs of pump make different performance. Design for better performance and efficiency of axial flow low-lift pump requires knowledge and understanding of the flow behavior that occurs within the pump. In the past, the design of pumps used to start with the design and construction of a small pump model to test in laboratory. The analysis test result was obtained to update the model and then test it again. The process was repeated until the desired performance was satisfactory. After that, a pump prototype was created. Such procedure was costly and time-consuming. Nowadays, the advance in computer science and fluid dynamics simulation techniques that can calculate faster and more accurately is used to simulate fluid flow behavior within the pump. Erik et al. [7] used commercial program Fluent Code with realizable $k-\varepsilon$ to predict the capacity of pumps by presenting various forecasting techniques, e.g., Multi Reference Frame (MRF), Mixing Plane (MP), and Sliding Mesh (SM), to compare them with the experimental results. It was found that Steady Method (MRF and MP) carries less validity compared with that of Unsteady Method (SM) which can precisely predict the capacity of pump and clearly reveal its internal flow behavior. Subsequently, Rui and Hong-xun [8] used the CFD code ANSYS CFX to estimate the hydraulic and cavitation performance of the slanted axial-flow pump under different operation conditions. Compared with the experimental hydraulic performance curves, the numerical results show that the filter-based model is better than the standard $k-\varepsilon$ model to predict the parameters of hydraulic performance. The above-mentioned is the background of research. The previous researches showed that the CFD flow simulation could reveal the flow behavior within pump and it was also a great tool for pump design. However, the experimental results would be measured and compared to the simulation results.

This research aims at using CFD to simulate flow in a Thai – made pump (Tor-Payanak) and compare the result with the experimental result from pump test rig. The application of CFD technique improving the pump performance can lead to reduce cost and time of design procedure. The obtained knowledge and the details of flow behavior will be beneficial for the further design of Thai-made axial flow low-lift pumps.

2. Background Concept

In the past decade, many researchers applied CFD for checking flow pattern in several kinds of pump. In ordinary simulation scheme, the shape of pump which is drawn in CAD must be imported into commercial code of CFD and then grids will be generated in order to create many cells with a tiny volume for calculation. The equation of motion which is derived from the principle of conservation of mass and momentum will be solved by finite difference method or finite volume method. In real situation, the pump works by some parts such as blade, hub, shaft, etc. being rotated with angular velocity and some parts such as shroud, guide vane, casing surface, duct wall, etc. being stationary part. Thus, to achieve accuracy, flow of fluid has to be analyzed by multiple reference frames (MRF) capability.

2.1 Mathematical Models

To describe the turbulent phenomenon, the modification of Navier-Stokes equations (NSEs) was established by Reynolds averaging method (RANS), generally used to transform NSEs [9 - 11].

By taking time average over the characteristic time of mean values, the instantaneous quantities of direct numerical simulation (DNS) equations were replaced by mean quantities and the additional unknown terms, describing the turbulence, were introduced.

2.1.1 Governing Equation

The modified NSEs, called Reynolds-Averaged Navier-Stokes (RANS) equations, can be written in the tensor form as:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho \cdot u_i)}{\partial x_i}=0 \tag{1}$$

$$\frac{\partial(\rho \cdot u_i)}{\partial t}+\frac{\partial(\rho \cdot u_i \cdot u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu \cdot \left(\frac{\partial u_i}{\partial u_j}+\frac{\partial u_j}{\partial u_i}-\frac{2}{3}\cdot \delta_{ij}\cdot \frac{\partial u_l}{\partial x_l}\right)\right]+\tau_{ij} \tag{2}$$

The additional unknown terms were named Reynolds Stress and defined as:

$$\tau_{ij}=\frac{\partial\left(\rho \cdot \overline{u_i' \cdot u_j'}\right)}{\partial x_j}$$

2.1.2 Boussinesq Hypothesis

The common fashion, used to model Reynolds Stress term, employs Boussinesq hypothesis, relating the Reynolds Stress with the velocity gradients:

$$\tau_{ij}=\mu_t \cdot \left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\cdot\left(\rho \cdot k+\mu_t \cdot \frac{\partial u_i}{\partial x_i}\right)\cdot \delta_{ij} \tag{3}$$

2.1.3 Modeling turbulent eddy viscosity

For the calculation of turbulence flow, the effect of turbulent eddy motions was described and incorporated into the turbulent viscosity term (μ_t). In contrast with the molecular viscosity, the eddy viscosity depends strongly on the flow property. Therefore, selecting the turbulent model, accommodated the flow behavior of each application, is very important. The standard $k-\varepsilon$ turbulent models for eddy viscosity were briefly described as follows.

2.1.4 Standard $k-\varepsilon$ Turbulent Model

For the standard $k-\varepsilon$ Model, the turbulent viscosity is computed by the combination of the turbulence kinetic energy k and its dissipative rate ε as follows:

$$\mu_t=\rho \cdot C_\mu \cdot \frac{k^2}{\varepsilon} \tag{4}$$

The k and ε were obtained from the following equations:

$$\rho \cdot \frac{Dk}{Dt}=\frac{\partial}{\partial x_i}\cdot\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\cdot \frac{\partial k}{\partial x_j}\right]+\mathrm{P}-\rho \cdot \varepsilon \tag{5}$$

and

$$\rho \cdot \frac{D\varepsilon}{Dt}=\frac{\partial}{\partial x_j}\cdot\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\cdot \frac{\partial \varepsilon}{\partial x_i}\right]+C_{1\varepsilon}\cdot \frac{\varepsilon}{k}\cdot P-C_{2\varepsilon}\cdot \rho \cdot \frac{\varepsilon^2}{k} \tag{6}$$

Variable P represents the production of turbulent kinetic energy. σ_k , σ_ε , $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are constant.

2.2 Physical model and CFD methodology

The physical model of Thai-made axial flow pumps is shown in Fig. 3. The pump consists of a 5-blades impeller and a 5-blade guide vane. The diameter of the impeller is 245 mm. The total length of pump is about 20 feet. The rotation speed of impeller is 700 – 930 rpm. A 5-hp electric motor was used to drive the pump.

Fig. 3 The 6 inch. Tor-Payanak configuration

The commercial CFD code, ANSYS FLUENT was used to solve three dimensional problems in steady state incompressible flow through the pump. The time-independent incompressible Navier-Stokes equations [12] and the standard $k-\varepsilon$ turbulent models was discretized using the finite volume method. QUICK and central differencing flow numerical schemes were applied for convective and diffusive terms, respectively. The discrete nonlinear equations were implemented implicitly. To evaluate the pressure field, the pressure-velocity coupling algorithm SIMPLE (Semi Implicit Method for Pressure-Linked Equations) was selected. The linearized equations were solved using multi grid method. Due to the geometrical complexity of the impeller and guide vane, the numerically approximated equations were performed on the collocated tetrahedral grid. The grid domains of guide vane, impeller, inlet and outlet regime were constructed separately, comprising approximate total of 1,441,069 cells (approx. 768,157 cells of impeller domain) as shown in Fig. 4. Generating mesh at the interface between the domains is non-conformal. The tip clearance of impeller is ignored in this simulation.

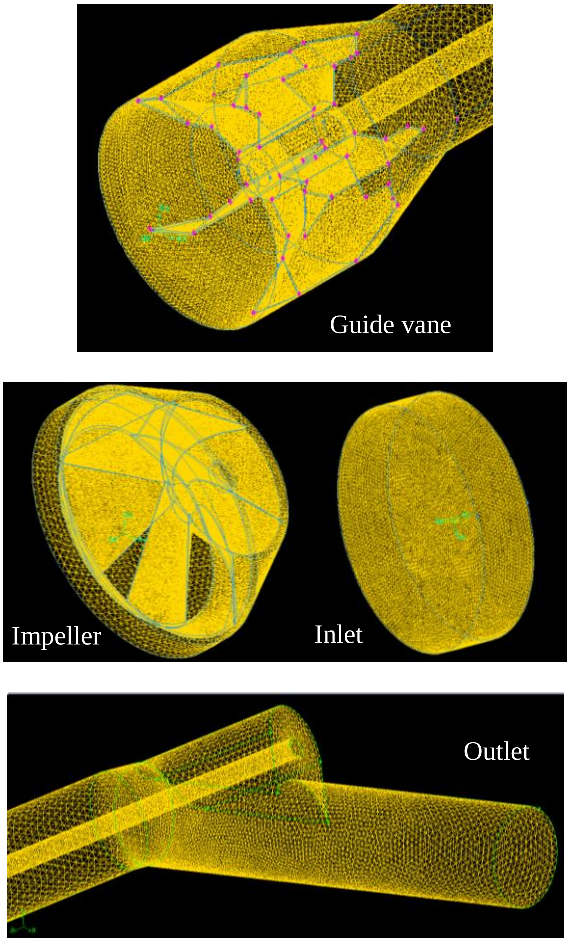

Fig. 4 Grid distribution of computational domain consists of guide vane, impeller, inlet, and outlet

This simulation was using moving reference frame (MRF) technique. The pressure inlet condition was set for upstream inlet and the pressure outlet condition was set for outlet tube boundary. The boundary condition on connected surface of each sub domain was set to be the interface as shown in Fig. 5. This simulation was set to be steady flow and standard $k-\varepsilon$ was selected for turbulence model. The pressure inlet condition was set to head (h3) equal to 1.29, 1.45, 1.57, 1.07, and 1.2 m. on the inlet boundary and pressure outlet condition was set to head equal to 1.44, 1.83, 2.23, 3.23, and 3.57 m. with turbulent intensity of 5%. No slip condition was set at wall section. The rotation speed of impeller was 700 – 930 rpm.

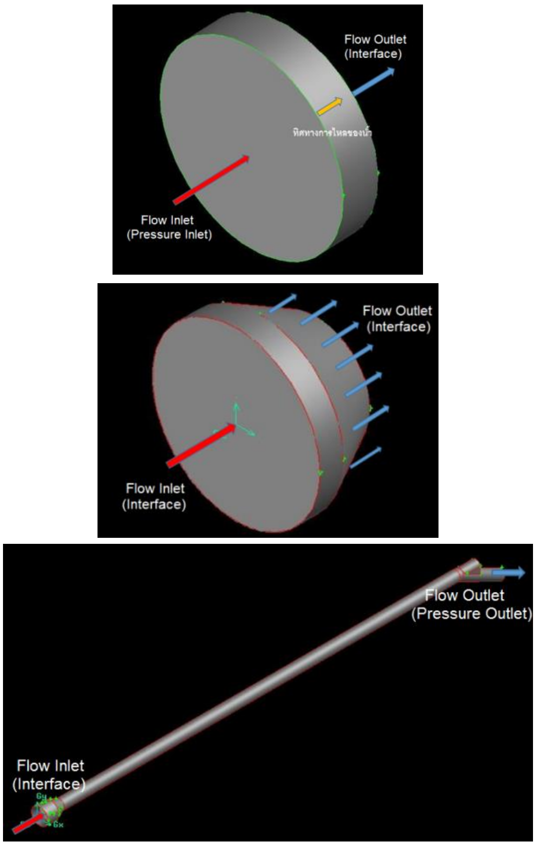

Fig. 5 Boundary condition of each domain

2.3 Pump test section

The pump test rig was set up to test a Thai-made axis flow pump as shown in Fig. 6. Pump test condition was set up as shown in Table 1 and the pressure data should be used for CFD simulation.

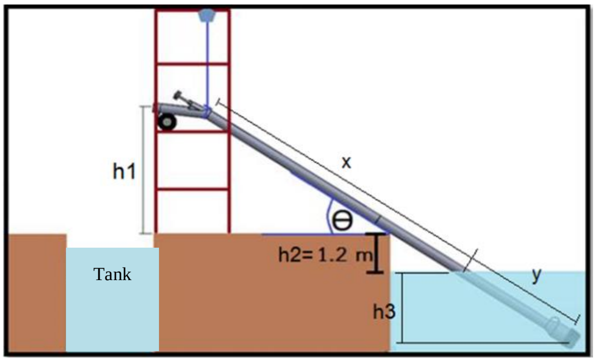

Fig. 6 Pump test rig

Table. 1 Setting up condition for pump test

Pressure for calculation (6 inch dia.)					
Angle θ (degree)	h1+h2 (m)	h3 (m)	h1+h3 (m)	Pressure inlet (Pa)	Pressure outlet (Pa)
25	1.44	1.29	2.73	12678.44	26804.84
30	1.83	1.45	3.28	14244.12	32196.42
35	2.23	1.57	3.81	15419.36	37344.71
40	3.23	1.07	4.30	10521.23	42207.53
45	3.57	1.20	4.77	11726.87	46748.57
50	4.45	0.74	5.19	7280.00	50934.50

The experimental result of Thai-made axis flow pump was shown in Fig. 7.

CST0005

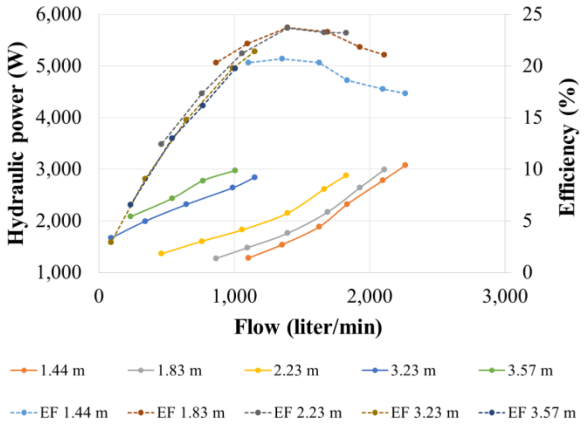

Fig. 7 The experimental result

Fig. 7 showed performance curves of the pump. The maximum efficiency was about 23.7 % at flow rate of 1,396 liter/min, head 2.23 m. and impeller speed of 850 rpm.

3. Results and discussion

3.1 Simulation result

The solution was converged when the residual value less than 1x10^{-4}. The velocity flow field within the pump and pressure flow field on pressure side of impeller surface are shown in Fig. 8. The highest velocity was found nearby leading edge of blades and then the speed slightly slowed down after passing through impeller. After the water flowed through the impeller, the wide circulation flow had occurred in the guide vane section. The circulation flow causes losses in the pump.

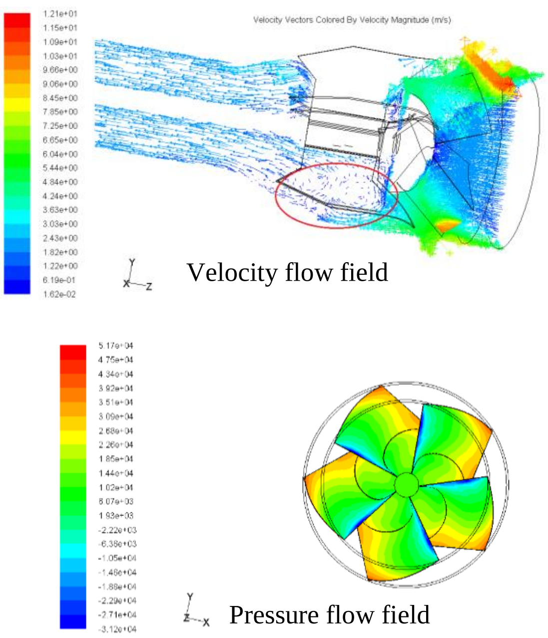

Fig. 8 Flow field within the pump at a speed of 930 rpm and a head of 1.44 m.

Performance curves of pump are shown in Fig. 9. The maximum efficiency is about 23 % at the flow rate of 2,300 liter/min. and the head of 1.83 m. The high efficiency range occurred between flow rates of 2,000 to 2,500 liter/min. with the impeller speed of 850 to 930 rpm.

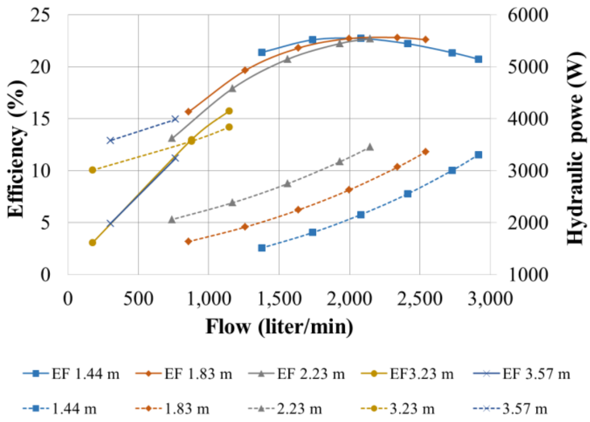

Fig. 9 The performance curves from simulation result

3.2 Compared result

The comparison of hydraulic performance curves between simulation and experimental approaches in the same operating conditions is shown in Fig.10 and Fig. 11. As it can be seen from Fig.10, both efficiency results from simulation and experiment are similar at the maximum value but different in flow rate. The simulation predicts more flow rate than the experiment coincident with a previous research by Kasantikul [4]. The simulation with RANS equation tends to overestimate turbo-machinery performance because RANS equation omits unsteady loss occurred within pump. Fig.11 shows that, the simulation results provide larger percentage of discrepancy comparing with the experimental results at high head operation.

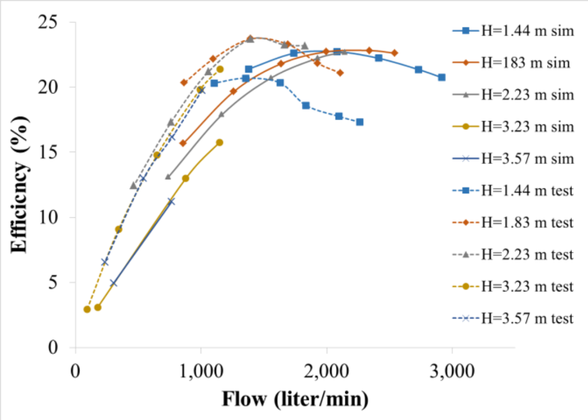

Fig. 10 Comparison of the efficiency result

CST0005

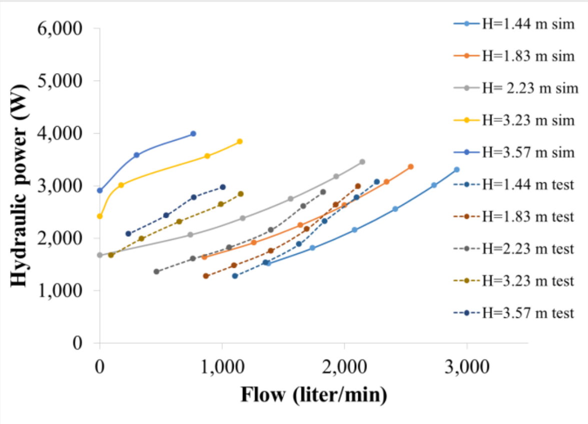

Fig. 11 Comparison of the hydraulic power result

4. Conclusion

Thai-made axial flow low-lift pumps with a pipe diameter of 6 inch and a length of 20 feet was chosen as a case study. CFD simulation with RANS and standard $k-\varepsilon$ turbulent model was employed. The simulation and experimental results were compared. The CFD simulation results were in fair agreement with the experimental result. However, CFD simulation tended to overestimate pump performance. The simulation of fluid flow field by CFD gave an understanding of flow behavior within pump that will be beneficial to improve performance and efficiency of Thai-made axial flow low-lift pumps. This technique can give correction and accurate result in order to reduce both difficulty and cost of pump design procedure.

5. Acknowledgement

The author would like to thank Research and Development Center for Sustainable Engineering (RDSE-SU), Department of Mechanical Engineering, Faculty of Engineering and Industrial Technology, Silpakorn University, and my work companion for all support to this work.

6. References

[1] Chinsuwan, W. and Cochran, B. J. (1986). Small Farm Equipment for Developing Countries: Past Experiences and Future Priorities: The axial-flow low-lift pump, ISBN: 971-104-157-X, Int. Rice Res. Inst., United States. Agency for International Development

[2] Kaewprakaisaengkul, C. (1996). Evaluation and improvement of Thai-made irrigation pumps. Thesis (Ph.D.)., AsianInstitute of Technology.

[3] Kasantikul B. and Laksitanonta S., (2011). Cavitations Analysis on Impeller Blades of Thai-made Irrigation Pump by Computational Fluid Dynamic technique, paper presented in *The Second TSME International Conference on Mechanical Engineering* , Krabi, Thailand .

[4] Kasantikul B. (2013). Numerical Simulations of Thai-made Irrigation Pump (Tor-Payanak), *Journal of Science and Technology*, Vol. 2, No. 3, August 2013, pp. 56 - 65

[5] Kasantikul, B. and Laksitanonta, S. (2014). Study and develop the thai-made irrigation pump system (Tor Payanak) in a large aquaculture pond, *Journal of Agricultural Technology*, Vol. 10(5), August 2014, pp. 1115 - 1138

[6] Udomkitmongkol, C. (2011). Study Reports of Agricultural Machinery on February 2011, The Office of Industrial Economics, Ministry of Industry, Thailand.

[7] Erik Dick, Janvierendeels, Sven Serbruyns and John vande voorde (2001). Performance prediction of centrifugal pumps with CFDTools, *Scientific Bulletin of the Academic Computer Center in Gdansk*, Vol. 5(4), July 2001, pp. 79 - 94.

[8] Rui, Z. and Hong-xun, C. (2013) Numerical analysis of cavitation within slanted axial-flow pump, *Sciencedirect Journal of Hydrodynamics*, Vol. 25(5), May 2013, pp. 663 - 672

[9] Tiaple, Y. et al. (2005). The development of bulb turbine for low head storage using CFD simulation, paper presented in *ME-NETT National Conference 19th*, Phuket, Thailand.

[10] Anderson, J. D. et al. (1992). Introduction to computational fluid dynamics, Edited by Wendt, John F, Spriger-Verlag, New York.

[11] Sritrakul, N. et al. (2013). Improving low head bulb turbine design through simulation technique, paper presented in *The 4th TSME International Conference on Mechanical Engineering*, Chonburi, Thailand.

[12] Chung, T.J. (2002). Computational fluid dynamics, Cambridge University press, Cambridge.

[13] http://gimzeng-ud.blogspot.com/2013/07/blog-post_7137.html

[14] http://www.torsubnam.com/?page_id=39

[15] http://topicstock.pantip.com/wahkor/topicstock/2011/10/X11152352/X11152352.html