# การจำลองเชิงตัวเลขการทำงานของเครื่องสูบน้ำไทย (ท่อพญานาค)

# Numerical Simulations of Thai-made Irrigation Pump (Tor-Payanak)

***Benya Kasantikul*[1*]**

***เบญญา กสานติกุล*[1*]**

**ABSTRACT**

The numerical simulations of mixed flow rotor functions were conducted to investigate the details of flow phenomena in mixed flow pump. The focus was on performance assessment of different operating conditions. Various pump speeds, such as 700, 800, 900, 1000 and 1,100 rpm, respectively and head operating conditions (i.e., 1.8, 2.0, 2.2, 2.4 and 2.6 m, respectively) were tested, and the obtained flow rates were recorded. Computation Fluid Dynamics (CFD) simulation of the rotor of a mixed flow pump was performed by using the commercial code ANSYS CFX with solving 3 dimension - Reynolds Averaged Navier–Stokes (3D - RANS). Then, flow rate was assumed to be three dimensions (viscous and incompressible).Shear Stress Transport (SST) turbulence model with standard wall function was used. The simulation and experimental results were compared and validated. The study showed that CFD result had correlation with experimental data. This simulation of pump flowing by CFD could give a good understanding of flow phenomena in pumps. However, CFD modeling demonstrated the over – estimation in all studied cases. The discrepancy was amplified at lower flow rate operation while at higher flow rate, CFD apparently obtained more accuracy of flow rate than at lower flow rate. Strong flow distortions were found at inlet, and separation flow were revealed at outlet.

**Keywords**: Numerical simulations, mixed flow pumps

**บทคัดย่อ**

วัตถุประสงค์ของการจำลองเชิงตัวเลขเกี่ยวกับการทำงานของเครื่องสูบน้ำไทยแบบไหลผสม คือเพื่อศึกษารายละเอียดของปรากฏการณ์การไหลของน้ำในเครื่องสูบน้ำ โดยมุ่งเน้นการประเมินประสิทธิภาพการทำงานของเครื่องสูบน้ำท่อพญานาคที่เงื่อนไขการทำงานต่างๆ โดยปรับตามความเร็วรอบของใบพัดที่ 700 800 900 1000 และ 1100 รอบต่อวินาที โดยทำงานที่เฮดความดัน 1.8 2.0 2.2 2.4 และ 2.6 เมตร และคำนวณอัตราการไหล เทคนิคทางด้านพลศาสตร์ของไหลเชิงคำนวณ (CFD) ถูกใช้ในการศึกษาทำงานของเครื่องสูบท่อพญานาค ด้วย ANSYS CFX และ 3-D Navier-Stokes (3D - RANS) โดยจำลองการไหลเป็น

---

[1*] ภาควิชาวิศวกรรมเครื่องกล คณะวิศวกรรมศาสตร์ กำแพงแสน มหาวิทยาลัยเกษตรศาสตร์ วิทยาเขตกำแพงแสน จ.นครปฐม 73140

Department of Mechanical Engineering, Faculty of Engineering, Kasetsart University, Khampaengsaen Campus, Nakornphathom, 73140, Thailand.

[*] Corresponding author: 0-3435-1407, Fax: 0-3435-1407, E-mail address: fengbyk@ku.ac.th

แบบสามมิติ มีความหนืดและอัดตัวไม่ได้ แบบจำลองความปั่นป่วนเป็นแบบ Shear Stress Transport (SST) ผลการจำลองและผลการทดลองที่ได้ถูกนำมาเปรียบเทียบและตรวจสอบ การศึกษาพบว่าผลที่ได้มีความสัมพันธ์กัน อย่างไรก็ตามผลที่ได้จากการจำลองจะมีค่าสูงกว่าเล็กน้อยในทุกเงื่อนไขการทำงาน ผลจะมีความแตกต่างที่อัตราการไหล ต่ำ ยิ่งอัตราการไหลสูง ผลจากการจำลองจะยิ่งใกล้เคียงกับผลจากการทดลอง นอกจากนี้ยังพบว่ามีการไหลแบบบิดเบือนที่ทางเข้าและมีการไหลแยกชั้นที่ทางออก

**คำสำคัญ:** การจำลองเชิงตัวเลข เครื่องสูบน้ำแบบไหลผสม

## Introduction

Thai - made pump is widely used in agriculture and aquaculture in Thailand. It is classified as low lift - high discharge pump. Although more than one-million pumps have been operated in Thailand, but a few of engineering disciplines have been applied in their design and development method (Kaewprakaisaengkul, 1996). The results of this study showed the considerable low operation of both mechanical and hydraulic efficiency. Locally developed - mixed flow pump was lower than 60% efficiency in operation (Kaewprakaisaengkul, 1996) due to lack of engineering knowledge for pump design of Thai manufacturers.

Apparently, there are few studies focusing on improvement and evaluation of Thai - made pumps (Kasantikul B. and Laksitanonta S., 2011). Kaewprakaisaengkul (1996) 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 was effected on mechanical power loss; iv) the study on impeller, diffusion vanes and pump casing geometry that effect on hydraulic efficiency; and v) the study on the effect of surface coating on hydraulic efficiency. Further, Kaewprakaisaengkul (1996) used 3D – CFD technique to predict cavitation phenomena, which occurred in Thai – made mixed – flow pump. The cavitation model of CFD code was validated by experimental data using NACA0012 airfoil. He reported that cavitation location was related to pressure distribution along entire airfoil, and that proper impeller dimension modification and surface coating obviously influenced the improvement of Thai - made pumps.

Presently, 3D - Computation Fluid Dynamics (CFD) is widely used in the fluid machinery industry (Ding *et al*., 2011). Further, this application can help the manufacturer save cost and time to design a pump by reducing the number of experimental tests. Moreover, this technique can show how the indexes of fluid properties work, presenting more understanding to the flow phenomena (Gao *et al*., 2008). However, the CFD model is needed to be validated and calibrated with experimental data of particular application before being applied as the efficient engineering tool (Bardina *et al*., 1997).

The purposes of this study were to validate this mixed - flow pump hydraulic flow rate prediction by CFD numerical simulation using RANS with SST turbulent closure model at the various pump speeds, i.e., 700, 800, 900, 1000 and 1,100 rpm, respectively while head operating conditions of 1.8, 2.0, 2.2, 2.4 and 2.6 m, respectively were varied, and the obtained flow rates were recorded.

**Impeller 3D Flow Simulation**

A 3D CFD flow simulation was carried out on an isolated impeller of a mixed - flow pump with the speed of 600 - 1100 rpm. The main pump parameters and geometry presented in Figure 1 and Table 1 were on the basis of Kaewprakaisaengkul, 1996 and Stepanoff, 1975.

In order to accomplish the numerical flow simulation and the most important first task is to define the geometry and grid generation. Regarding the study of each isolated impeller, assuming an axisymmetric flow simplifies the domain to a single blade passage. The simulation domain of the impeller mixed - flow pump is schematized in Figure 2 (based on Kaewprakaisaengkul, 1996 and Kasantikul and Laksitanonta , 2011). A structured grid was created with the ANSYS Turbogrid software.

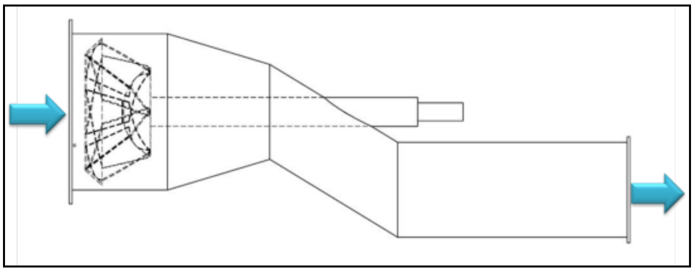

**Figure 1** Visualization mixed-flow pump

**Table 1** Geometrical parameters of the pump

| Mixed-flow pump | | |
|---|---|---|
| Parameter | Value | Description |
| Impeller | | |
| $R_0$ | 117mm | Inlet flange radius |
| $R_1$ | 97.5mm | Mean impeller inlet radius |
| $b_1$ | 85.9mm | Inlet impeller width |
| $\beta_1$ | 59° | Inlet blade angle |
| $\theta_1$ | 46° | Blade LE inclination angle |
| $R_2$ | 117mm | Mean impeller outlet radius |
| $b_2$ | 85mm | Outlet impeller width |
| $\beta_2$ | 68° | Outlet blade angle |
| Na | 6 | Blade number |
| $e$ | 3 mm | Blade thickness |

**Simulation parameters and boundary conditions**

The general parameters and boundary conditions used for 3D flow simulation of the impeller are summarized in Table 2. Regarding all simulations, the boundary conditions were as follows:

i) Inlet: total pressure applied in the rotation axis direction;

ii) Outlet: imposed mass flow;

iii) Periodic: two symmetry surfaces positioned in the middle of the blade passage; and

iv) Wall: general boundary conditions by default. The simulation domains at the inlet and outlet sections were extended enough to allow inlet recirculation and the elliptic influence of the flow.

v) The flow was assumed to be steady and incompressible flow.

vi) The density of water was assumed to be constant at 998.2 kg/m$^3$.

vii) The viscosity was assumed to be constant at 0.001003 kg/ m·s.

viii) Turbulent intensity was 3 %.

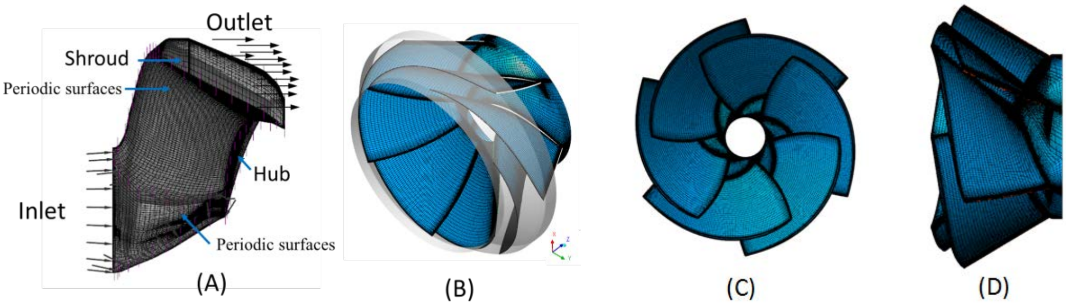


**Figure 2** Impeller flow simulation domain and structured grid

**Table 2** Impeller simulation parameters

| Parameters | ANSYS CFX |
|---|---|
| Flow simulation domain | Single impeller flow channel (periodic interface ) |
| Grid | Structured |
| Inlet | Total pressure = 101325 (Pa) |
| Outlet | Mass flow = variable (kg/s) |
| Turbulence model | $K$ - $\omega$ SST |
| Maximum residual convergence | $10^{-4}$(RSM) |

**Numerical methods**

ANSYS CFX code was used as a solver in this study. The conservative forms of Navies Stokes Equations and k-ω SST turbulence model were used as code to solve equations for ideal gas (Menter *et al*., 2003 and Lifante *et al.*, 2008). A part of solver strategy used in ANSYS CFX was multigrid method. The particular variant used in ANSYS CFX was based on principle conservation, which was already implicit in the Finite Volume discretization that was called Additive Correction Multigrid. The linearized discrete algebraic equations arose from most finite volume methods, which were sufficiently diagonally dominant to permit solution by simple relaxation methods.

**Computational grid**

The computational grid is generated with a multiblock grid generator based on transfinite interpolation. The computational grid is a structure of 8 blocks including additional blocks at inlet and outlet of the computational domain. This grid is relatively complex topology, which establishes a high quality and avoids strongly skewed cells. The *O-mesh* around the blade provides a good resolution of the leading and trailing edges. Total cells of 950,000 hexahedral elements of structured grid are used to model the computational domain.

**Results and discussions**

**Validation of analysis results**

Prior to the analysis of a mixed-flow pump, the optimal number of grids had to be determined. Hence, a preliminary grid dependency was tested with numbers of nodes ranging from 200000 to 800000 (Figure 3), and 600000-nodes were selected as the optimum number of grids. The performance test was set up as in Figure 4. The static pressure was measured at the outlet of the mixed-flow pump to determine the total head, and the dynamic pressure was calculated by measuring the discharging mass flow rate (Koumoutsos *et al*., 2000 and Youcef AIT, 2006).

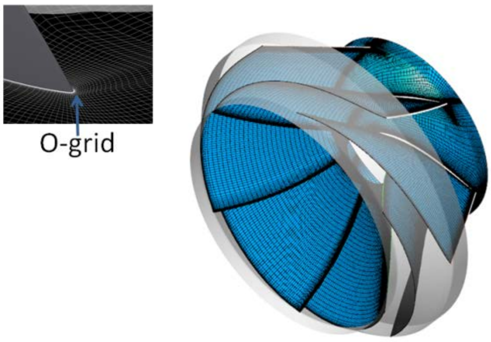


**Figure 3** Computational grids

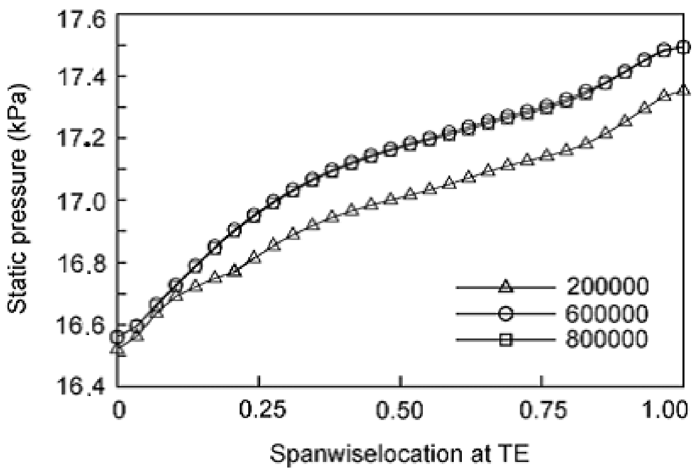


**Figure 4** Grid dependency test results

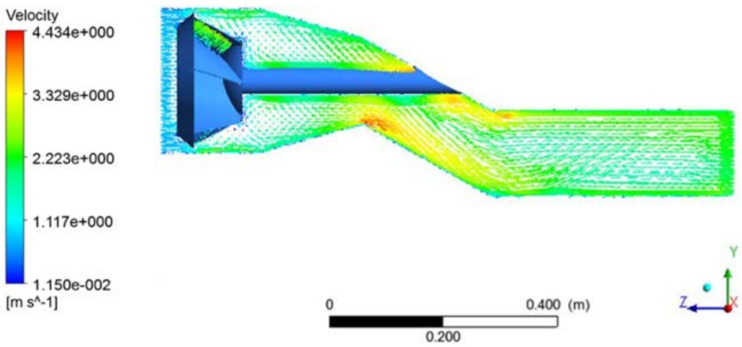


**Figure 5** Flow structure inlet to outlet

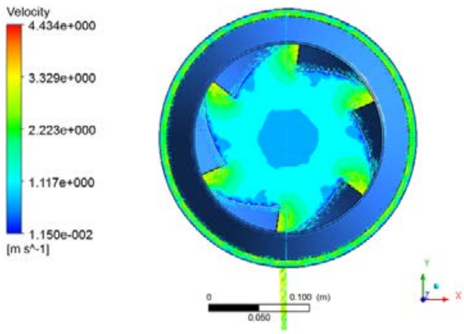


**Figure 6** Flow structure inlet

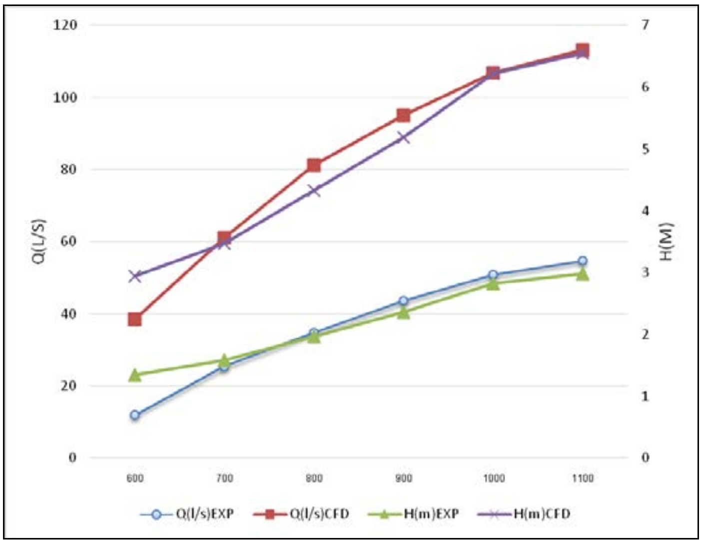


**Figure** 7 Comparison between experimental Q-H and CFD Q-H at various RPM

The selected CFD model predicted that flow rate and head were fairly in agreement with the experimental data as in Figure 7. At lower flow rate operation, the CFD model provided larger percentage of discrepancy comparing to that at high flow rate operation. The model predicted more flow rate than the experimental data along 13-50 l/s operation range. Coincide with previous research (Goto *et al.*, 2008), the performance of two different numerical techniques, RANS and URANS were compared with this study to determine their suitability in the prediction of flow phenomena in pumps. Oh *et al.* (2005) confirmed from his work that RANS model tends to over predict turbo-machinery performance. Since RANS model was omitted unsteady loss in contrast with URANS that large time scale of unsteady motion are captured. The flow energy (differentiation of the inlet and outlet enthalpy) plus power, which done by impeller on fluid that would balance with flow dissipation. Simply analogous could be made by focusing on dissipation of fluid element in arbitrary flow field drew on the force diagram (Figure 9). There were several forces acting on fluid element, e.g., pressure gradient, body force and shear force. However, this study focused only the shear force (F). Magnitude of shear force was depends on: 1) Turbulent modeling performance; 2) Near-wall modeling; and 3) Unsteady modeling (Omit for RANS). Some degree of inaccuracy shear force prediction was admitted by nature of turbulent modeling (Oh, H.W. and Yoon, 2007).

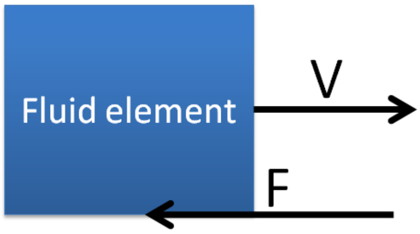


**Figure 8** Shear force acting on fluid element that moves with speed of V

Regarding Figure 8, dissipation was calculated by P = Shear force * V. While at low flow rate, V becomes small number. Resulting from any inaccuracy predicting of the shear force from turbulent model would compensate by larger percentage error predicting of V. Since V is directly linked to flow rate prediction. Regarding Figure 8, at the same percentage error of shear force would lead to: 1) larger percentage error of V when V is small; and 2) smaller percentage error of V when V is large. Coincidently with previous studies show that RANS modeling tends to predict lower shear force than URANUS and experimental result. This would lead to over prediction of (V) or flow rate. Since CFD model is trying to balance the dissipation with work done from impeller on fluid plus different of fluid enthalpy of inlet and outlet. Shear force inaccuracy prediction lead to over-predict of flow rate with larger percentage error at low flow rate and lower percentage error at high flow rate.

**General flow structure**

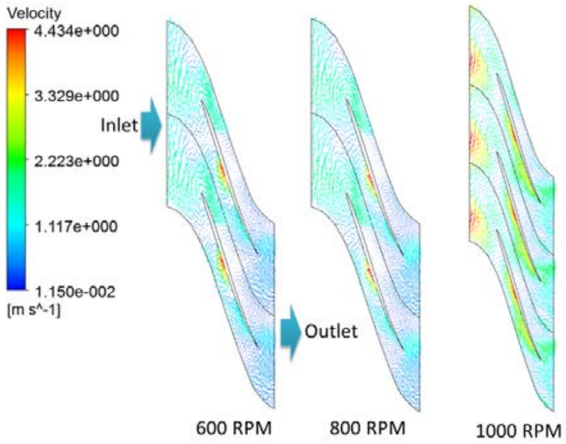


**Figure 9** flow structure Blade to Blade of various RPM

Regarding all cases, flow structures were similar as showed in Figure 9. Strong distortion flows were observed around the leading edge. The highest turbulent intensity occurred in this region because of the highly complex shear flow structure. The flows were well guided along with flow channel at around ¼ distance of chord length. It can reveal the faster flow at the pressure side comparing to suction side of the blade resulting in shear layer. Separation start at the trailing edge that the flow from pressure side was push toward to suction side (Brennen, 1994). Eventually, Complex shear flow along with high turbulent promotion commenced from this region through the outlet of the flow domain.

## Conclusions

Numerical simulation of Thai-made pump by CFD modeling, which employed RANS equation, would be over predicted the flow rates at all speeds, 600 – 1,000 RPM. Moreover, at lower RPM the modeling had larger error percentages of over-flow rate prediction than at higher RPM compared to the experimental results.

This simulation of pump flowing by CFD can give a good understanding of flow phenomena in pumps. From this simulation, all flow structures from CFD were similar to the data from experiments of all over 600-1,000 RPM. There was high flow gradient of serve at the inlet and outlet zones. Strong flow distortion could be revealed at the inlet and strong flow separation occurred at the outlet.

## References

Bardina, J. E.,Huang,P. G., and Coakley,T. J. (1997). Turbulence modeling, validation, testing and development. NASA TM 110446.

Brennen, C.E. (1994). Hydrodynamics of Pumps, Oxford University Press, London.

Goto, A., Nohmi, M., Sakurai, T. and Sogawa, Y., 2002, "Hydrodynamic Design System for Pumps Based on 3-D CAD, CFD, and Inverse Design Method," ASME J. Fluids Engineering, 124, published in this issue, pp. 329–325.

H. Ding, F. C. Visser,Y. Jiang and M. Furmanczyk (2011). Demonstration and Validation of a 3D CFD Simulation Tool Predicting Pump Performance and Cavitation for Industrial Applications. *ASME J. Fluids Engineering* ,133:1.

Hong Gao, W. Lin, and Zhaohui Du. (2008). Numerical Flow and Performance Analysis of a Water-Jet Axial Flow Pump, Ocean Engineering, Vol.35, No.16, pp. 1604-1614.

Kaewprakaisaengkul, C. (1996). Evaluation and improvement of Thai-made irrigation pumps. Thesis (Ph.D.)., Asian Institute of Technology.

Kasantikul B. and Laksitanonta S., (2011). Cavitations Analysis on Impeller Blades of Thai-made Irrigation Pump by Computational Fluid Dynamic technique. The Second TSME International Conference on Mechanical Engineering 19-21 October, 2011, Krabi.

Koumoutsos, A., Tourlidakis, A., and Elder, R. L. (2000). Computational studies of unsteady flows in a centrifugal compressor stage.*Proc. InstnMech.Engrs,Part A: J.Power and Energy*, 214(A6), pp. 611–633.

Lifante, C. and Frank, T.,(2008). Investigation of higher order pressure fluctuations and its influence on ship stern, taking into account cavitation at propeller blades, Final Report ANSYS / TR-08-04, Research Project No: 03SX202A.

Menter, F. R., Kuntz, M., Langtry, R.,( 2003) Ten Years of Industrial Experience with the SST Turbulence Model. In: Hanjali_c, K., Nagano, Y., Tummers, M. (Eds.), *Turbulence, Heat and Mass Transfer 4*, Begell House, pp. 625-632.

Oh, H.W., Yoon, E. S., Park, M. R., Sun, K., and Hwang,C. M.( 2005). Hydrodynamic design and performance analysis of a centrifugal blood pump for cardiopulmonary circulation. *Proc. IMechE, Part A: J. Power and Energy*, 219(A7), pp. 525–532.

Oh, H.W. and Yoon, E. S. (2007) Application of computational fluid dynamics to performance analysis of a Francis hydraulic turbine. *Proc. IMechE, Part A: J. Power and Energy*, 221(A4), pp. 583–590.

Stepanoff, A.J., (1975). Centrifugal and axial flow pumps Theory Design and Application, John Wiley, New York, pp.138-160.

Youcef AIT, B. (2006). Physical modeling of leading edge cavitation: Computational methodologies and application to hydraulic machinery. Thesis (Ph.D.), University of Paris.

**Received 25 April 2013**

**Accepted 13 August 2013**