Centrifugal Pump Efficiency Prediction by Means of Empirical Methods and Exact Navier - Stokes Solutions

Fatsis Antonios
Mechanical Engineering Department
Technological University of Central Greece
34400 Psachna, Greece

Vlachakis Vassilios
ESM, Virginia Polytechnic Institute and State University
Blacksburg, VA, USA

Panoutsopoulou Angeliki
Hellenic Defense Systems S.A.
1, Ilioupoleos Avenue, Hymettus, Greece

Vlachakis Nikolaos
Mechanical Engineering Department
Technological University of Central Greece
34400 Psachna, Greece

***Abstract*—This article presents three different methods for centrifugal pump efficiency prediction. The first method is based on exact solutions of the Navier-Stokes equations. The second one is based on empirical laws from laboratory measurements and the third one is a new empirical law which takes into account geometrical and operating centrifugal pump parameters. All three methods are validated against experimental data for various centrifugal pump geometries found in the literature. It is concluded that the methods presented are reliable and they accurately estimate the pump efficiency distribution with respect to volume flow rate.**

Keywords—Centrifugal Pump; Hydraulic Efficiency; Pump Impeller; Exact Solution; Empirical Method; Experimental Data; Flow Rate.

I. INTRODUCTION

In recent years, centrifugal pumps have been increasingly used in industrial, agricultural and domestic applications. For the cost-effective design of pumps it is essential to predict their efficiency prior to manufacturing and placement in installation. To do so, a number of unknown issues associated to the enhancement of the pump efficiency need to be investigated. Since the impeller is an active part of the pump adding energy to the fluid, its geometry plays a major role in the centrifugal pump performance. Modern design practices demand a detailed understanding of the internal flow for design and off-design operating conditions. Today's computer competency as well as the progress of numerical methods' accuracy brought turbomachinery Computational Fluid Dynamics (CFD) methods from pure research work into the competitive industrial pump market. However, advanced CFD commercial software is not suited for a quick assessment of characteristic lines of a series of pumps, due to the detailed geometrical pump data required for the grid generation. Alternatively, very fast and accurate tools for the pump overall efficiency prediction prior to the detailed flow analysis and laboratory testing, can be given either by prediction methods based on exact solutions of the Navier-Stokes equations, or combination of traditional analysis and design approaches based on empirical correlations of model testing and engineering experience.

II. LITERATURE SURVEY

Researchers tried to investigate the centrifugal pump efficiency by means of numerical and experimental efforts.

An accurate one-dimensional flow method for centrifugal pump efficiency prediction was proposed in [1]. It was based on the Euler and energy equation, including various types of losses. The predicted hydraulic pump efficiency was found consistent with experimental data in [2]. For one test case examined, the predicted efficiency was overestimated at low mass flow rates. This indicates the difficulty of such methods to provide an accurate prediction of pump efficiency over the whole range of volume flows.

Theoretical analysis on pump efficiency in [1] revealed that the output hydraulic power of the pump can be divided into three parts based on the effect of the pump, being the effective output power (to produce the static head), the necessary output (to overcome the system loss), and the actual output of the pump.

The blade number of impeller is an important design parameter, affecting the characteristics of centrifugal pumps.

Calculations and experiments with centrifugal impellers having different number of blades [3], [4], revealed that the blade number has an important effect on the flow field inside the impeller and on its jet-and-wake structure, affecting the pump efficiency. In general, the efficiency is increased by increasing the number of blades from 2 to 5. Further increase of blade number does not necessarily mean increase in efficiency. It was found that there is a critical number of blades that corresponds to the maximum pump efficiency.

The influence of pump impeller inlet geometry on hydraulic performance of centrifugal pumps has been studied experimentally and numerically in [5]. Five impellers have been considered by extending the blade leading edge or applying much larger blade angle at impeller inlet compared with the original impeller. The 3-D turbulent flow inside those pumps has been analyzed basing on RNG k-ε

turbulence model. Based on the experimental test and numerical simulation, it was concluded that extending the blade leading edge and applying large blade angle at impeller inlet improve hydraulic pump performance.

The effect of the blade exit angles was studied by means of a simple numerical model in [6]. It was concluded that when the blade exit angle is increasing, the efficiency of centrifugal pumps based on the model of [7] is also increasing. However, in some cases, depending on the value of the specific speed, when the blade exit angle is decreasing, the pump efficiency is increasing.

The effects of flow viscosity play also an important role to the pump efficiency. Having investigated experimentally and numerically the influence of viscosity and of blade exit angle, it was concluded in [8] that the pump efficiency is significantly increasing as the viscosity is decreasing, but when increasing the discharge angle, it just slightly increases as well.

The pump efficiency is also influenced by the pump specific speed [9]. For the case of low specific speed pumps, an increase in specific speed causes also an increase in efficiency. An increase in the impeller blade height at trailing edge, b_2, makes the pump attain higher efficiencies.

Calculations have been performed in [10] by means of the finite volume method to solve the unsteady 3D incompressible Navier Stokes equations. Standard *k*-ε turbulence model with enhanced wall function and pressure-implicit with splitting of operators was chosen for turbulence model and pressure-velocity coupling respectively.

Calculations were performed in [11], [12] and [13] using the FLUENT package. The code uses the finite volume method and solves the fully 3D incompressible Navier-Stokes equations, including the centrifugal and Coriolis force source in the impeller channel. The pressure-velocity coupling is performed using the SIMPLE algorithm. Second order, upwind discretization is used for convection terms and central difference scheme for diffusion terms. Results are similar to those reported in earlier literature and also confirm the theory of [7] regarding the effects of the meridian curvature on the span-wise, as well as those of the blade forces in the pitch-wise velocity distributions.

The fully 3D incompressible Navier-Stokes equations are performed in ANSYS CFX 13.0 code in [14]. The finite volume method has been used for the discretization of the governing equations, and a high resolution algorithm has been employed to solve the equations. Turbulence is simulated with the shear stress transport (SST) *k*-*ω* turbulence model. The space and pressure discretization schemes are second order accurate. Five main impeller geometric characteristics were chosen as the research target to carry out calculations: The impeller blade height at trailing edge, b_2, the impeller inlet diameter, D_1, the impeller blade warp angle, φ, the impeller inlet blade angle, β_1 and the impeller outlet blade angle, β_2. It was found that the impeller blade outlet width is the most important efficiency determinant.

Numerical investigation on the effects of pump design parameters was carried out in [15] including the blade height, the blade number, the outlet blade angle, the blade width, and the impeller diameter. Three cases were considered: impeller, impeller and volute, and combined impeller and diffuser. The continuity and the Navier-Stokes equations with the *k*-*ε* turbulence model and the standard wall functions were used by means of ANSYS-CFX code. Results revealed that some design parameters have an impact on the centrifugal pump performance describing the head, the brake horsepower, and the overall efficiency. Numerical results were compared against experimental data considering the case of combined impeller and diffuser showing good agreement.

A numerical investigation of very low specific speed impellers of centrifugal pumps was done in [16] using the FLUENT software solving the Reynolds Averaged Navier-Stokes equations, using the *k*-*ε* turbulence model. It was shown that thick trailing edges suppress local eddies in the blade channels and decrease energy dissipation due to excessive swirling.

A numerical model for the simulation of the 3-Dimensional turbulent flow in centrifugal pump impellers solving the Reynolds Averaged Navier-Stokes equations was presented in [17]. The numerical results showed a fair comparison to experimental data.

Experimental and numerical performance investigation on the effect of the outlet blade angle of mini centrifugal pumps was studied in [18]. For these small-sized fluid machines the internal flow condition is not clarified and conventional theory is not suitable to predict performance characteristics. Numerical predictions using the FLUENT software showed good agreement when compared against experimental data.

Experimental data and numerical simulation of the three-dimensional unsteady pump flow taking into account the impeller-volute interaction with different outlet blade angles was done in [19]. Results obtained using impellers with different outlet angles showed that when the blade outlet angle increases, the centrifugal pump efficiency is improving.

An analysis and design method using the commercial package FLUENT was proposed in [20]. Experimental results verified that the pumps designed by the direct and inverse iterative design method have better hydraulic performance than the pump designed by the traditional design method.

The effect on efficiency due to the number and shape of recirculation channels located at the impeller outlet, was examined in [16] using the FLUENT software. It was found that L-shaped channels provided the highest efficiency, which was also experimentally confirmed. On the contrary T-shaped channels at the impeller outlet produce a drop in efficiency.

A generalized approximate formula of pump efficiency by means of an exponential function was done in [21]. This function represents the pump efficiency in terms of the impeller rotational speed and the pump volume flow.

Prediction of pump efficiency using the CFD software package FLUENT using two different turbulent models was presented in [22]. It was found that the predictions using SST *k*-*ε* showed better agreement to experimental data than when using the RNG *k*-*ε* turbulence model.

The effect of mesh style and grid convergence on numerical simulation accuracy of centrifugal pump was studied in [23]. When compared to experimental data, the structured, unstructured or hybrid meshes were found to have certain differences in the impeller velocity distributions by varying the number of grid cells.

Three different original numerical methods are presented here, aiming to predict centrifugal pump efficiency. The first one is based on exact solution of the Navier-Stokes equations. The second one is empirical, based on correlations resulting from experiments in the laboratory. The third method is an empirical one, based on geometrical and operating pump parameters. These methods are applied on various pump geometries and the results are compared with experimental data available in the literature, leading to the conclusion that can be useful tools to assess centrifugal pump performance.

III. Pump Efficiency Prediction Methods

A. Exact Solution of the Navier Stokes Equations

This approach consists of the exact solution of the Navier-Stokes equations for the case of centrifugal pump impellers. This methodology, described in detail in [24], assumes that: (a) the impeller flow field has reached steady state conditions; (b) the circumferential component u_θ does not depend on the axial coordinate, z. Thus, the impeller head is found to be:

$$H = \frac{1}{2}\cdot\left[\left(J_1^2 + J_0^2\right)e^{-\frac{4b\Delta z}{D_2}} - \frac{8\cdot\Delta z}{\mathrm{Re}\cdot D_2}\right]\cdot\left(n\cdot D_2\right)^2 \qquad (1)$$

where J_0 and J_1 are the Bessel functions of the First kind [25], Δz is the axial gap between the impeller disk and casing, D_2 is the impeller outlet diameter and Re is the Reynolds number.

The hydraulic pump efficiency of the impeller is defined as the ratio of the net power added to the passing fluid, divided by the electric power given to the impeller shaft, [17], i.e.

$$n_{PUMP,I} = \frac{N}{N_{el}} = \frac{1}{N_{el}}\int\frac{dW}{dt} \qquad (2)$$

where N denotes the power delivered from the pump to the fluid and N_{el} denotes the electric power given to the pump. The work added to the fluid, dW, can be calculated as

$$dW = \rho\cdot g\cdot H\cdot dV \Rightarrow \frac{dW}{dt} = \rho\cdot g\cdot H\cdot\frac{dV}{dt} = \rho\cdot g\cdot H\cdot d\dot{V}$$

The pump efficiency then is calculated from the integral:

$$n_{PUMP,I} = \rho\cdot g\int H\,d\dot{V} \Rightarrow n_{PUMP,I} \approx \int H\,d\dot{V} \qquad (3)$$

Integrating the above equation results to:

$$n_{PUMP,I} = 1,2\cdot\left[J_0^2\left(\frac{\dot{V}}{\dot{V}_{\max}}\right)^{0,33} + J_1^2\left(\frac{\dot{V}}{\dot{V}_{\max}}\right)^{0,33}\right]\cdot \left[\left(\frac{\dot{V}}{\dot{V}_{\max}}\right)^{0,33} - 2\cdot\ln^3\left[1,5\cdot\left(\frac{\dot{V}}{\dot{V}_{\max}}\right)^{0,33}\right]\right] \qquad (4)$$

Expressions for the impeller volume flow $\dot{V}$ as well as for the maximum volume flow are given in [24].

B. Prediction method resulting from experimental data

Empirical laws can be developed from experimental data obtained from extensive testing of various pumps with different geometrical characteristics running at different flow conditions. According to this approach, gathering all experimental data and examining them it was concluded that the pump efficiency is influenced by two categories of efficiencies that depend principally on the ratio of the operating volume flow divided by the maximum volume flow, i.e. $\dot{V}/\dot{V}_{\max}$ and on pump geometrical characteristics.

$$n_{PUMP,II} = \left(n_I\cdot n_{II}\right)^{1/2} \qquad (5)$$

The first category is a function of: (a) the ratio $\dot{V}/\dot{V}_{\max}$, (b) the blade width at the impeller outlet and (c) the number of impeller blades. So the following empirical relation was derived, trying to model the influence of the three aforementioned effects:

$$n_I = 2,8\cdot\left(\frac{\dot{V}}{\dot{V}_{\max}}\right)\left[1{,}15 - 0,8\cdot\left(\frac{\dot{V}}{\dot{V}_{\max}}\right)\right]\cdot\left(b_2\cdot z^{\frac{n}{1000}}\right)^{0,2} \qquad (6)$$

where

b_2 is the blade width at the impeller outlet

z is the number of impeller blades

n is the rotational speed of the impeller expressed in *rpm*

The second category depends on: (a) the ratio $\dot{V}/\dot{V}_{\max}$ and (b) the impeller outlet diameter. The following function was derived trying to model the two aforementioned effects:

$$n_{II} = \left\{1 - 0,9\cdot\left[\left(\frac{\dot{V}}{\dot{V}_{\max}}\right) - 0,8\right]^{0,2} - 0,09\cdot\left(\frac{\dot{V}}{\dot{V}_{\max}} - 0,3\right)^{0,3}\right\}\cdot\left(\frac{D_2}{330}\right)^{0,75} \qquad (7)$$

where
D_2 is the impeller outlet diameter

C. New Empirical method

Modifying the approach adopted in [2], it is proposed that the pump efficiency can be analyzed in two categories of parameters:

(a) The first category is influenced by geometrical parameters:

$$n_G = \left(n_{b2} \cdot n_{D1} \cdot n_{\varphi} \cdot n_{\beta 2} \cdot n_{\beta 1} \cdot n_t \cdot n_z \cdot n_{D2}\right)^{1/8} \qquad (8)$$

where

n_{b2} refers to the effect on efficiency due to the blade height at the impeller outlet

n_{D1} refers to the effect on efficiency due to the impeller inlet diameter

n_{φ} refers to the effect on efficiency due to the difference of peripheral angle between trailing edge and leading edge

$n_{\beta 2}$ refers to the effect on efficiency due to the blade angle at trailing edge

$n_{\beta 1}$ refers to the effect on efficiency due to the blade angle at trailing edge

n_t refers to the effect on efficiency due to the influence of the blade thickness

n_z refers to the effect on efficiency due to the outlet impeller blade angle

n_{D2} refers to the effect on efficiency due to the impeller outlet diameter

(b) The second category is influenced by flow parameters, given by the following equations:

$$n_F = \left(n_v \cdot n_n\right)^{1/2} \qquad (9)$$

where

n_v refers to the effect on efficiency due to the volume flow

n_n refers to the effect on efficiency due to the impeller speed of rotation

Once the efficiencies which depend on these parameters are determined according to [1], the pump efficiency is:

$$n_{PUMP,III} = \left(n_F \cdot n_G\right)^{1/2} \qquad (10)$$

IV. RESULTS

A first test case to assess the applicability of the present methods described previously, is the prediction of the 6-bladed centrifugal pump impeller with backwards curved blades, running at *1450 rpm*, [19]. Figure 1 shows comparison between numerical results obtained by the three different approaches analyzed in the present article and experimental data found in [19]. The continuous line represents numerical predictions obtained of the exact solution of the Navier-Stokes equations using equation (4). Dash-dotted line represents the efficiency prediction based on experimental data, using equations (5), (6), (7). Dotted line represents the efficiency prediction using the new empirical law, using equations (8), (9), (10). Finally, squares represent experimental data found in [19].

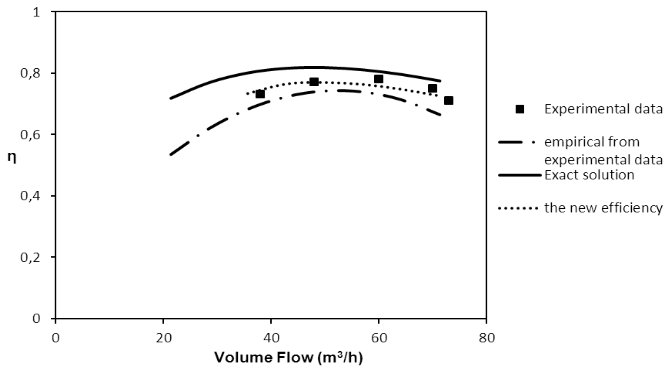

Fig. 1. Comparison between numerical results by the present method and experimental results in [19]

One can see from figure 1 that all numerical predictions follow the trend of experimental data. The exact solution overpredicts the pump efficincy, while the prediction based on experimental data under predicts the efficiency. The new empirical prediction stands in between showing a very good agreement to experimental data.

Experimental data regarding the efficiency of a 6-bladed impeller with outlet diameter $D_2 = 0.2\ m$ are found in [23] for a second test case. The rotational speed is *1450rpm*, the design point flow rate is *25.2m³/h*, the specific speed is $n_s = 73.6$ and the maximum efficiency measured as $53{,}9\%$.

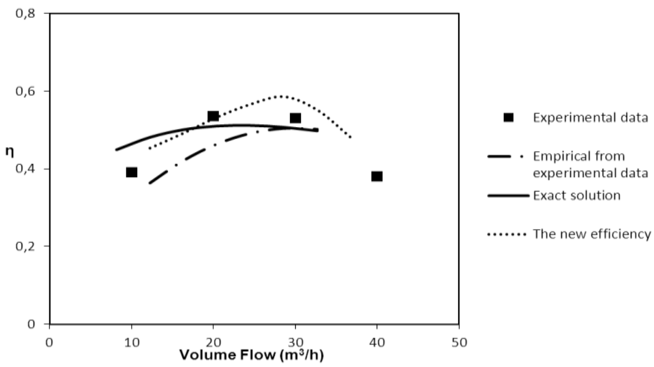

Fig. 2. Comparison between numerical results by the present method and experimental results in [23]

From figure 2, one can see that the exact solution (continuous line) passes among the experimental points and gives a good estimation of the efficiency behavior, under-predicting the maximum efficiency. Empirical law from experimental data also agree with measurements, but the maximum efficiency predicted with this method occurs at higher flow rate than the experimental one. A rather interesting shape has the curve representing the efficiency using the new empirical law presented in this article. This method gives the prediction of the maximum efficiency at higher volume flow than the experimental one.

A third test case is a 6-bladed centrifugal impeller examined in [6]. This centrifugal pump impeller has an outlet diameter $D_2 = 0.1\ m$, rotating at *2900 rpm* with specific speed $n_s = 38.47$.

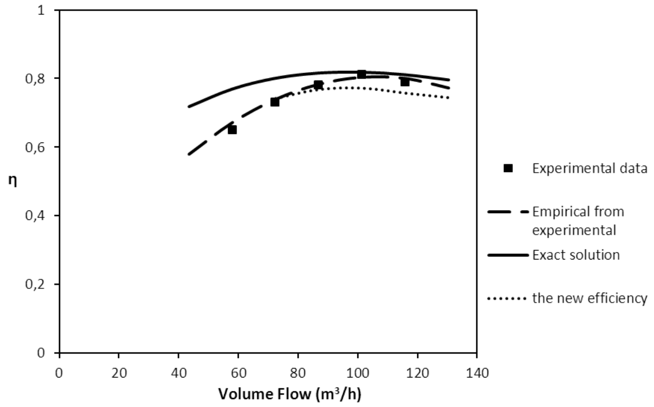

Fig. 3. Comparison between numerical results by the present method and experimental results in [6]

Figure 3 shows comparisons of numerical predictions obtained with the three different models presented in this article, against experimental data. Predictions obtained using the present method show quite good agreement to reference data from [6]. Empirical laws from experimental data agree very well to measurements for all the range of volume flows. Efficiency predictions using the exact solution method agrees well only in high flow rates and predicts higher efficiency at low and medium flow rates. Efficiency prediction using the new empirical method agrees also well to measurements showing better agreement to low flow rates.

Another test case where experimental data are available, concerns the centrifugal pump impeller discussed in [20]. This impeller has an outlet diameter $D_2 = 0.322\ m$, rotating at *1480 rpm*. Predictions obtained using the present method are compared in figure 4 against experimental data found in [20].
From the above figure one can see the comparison between numerical predictions and experimental data for this impeller. Numerical results using the exact solution of the Navier-Stokes equations follow the trend of the experimental data and agree very well with them for all flow regimes.
Results obtained using empirical laws from experimental data show also good agreement with measurements, but discrepancies can be observed in low flow rates. Numerical predictions using the new efficiency method lie between the curve of the exact solution and the curve using empirical laws from experimental data. The comparison is satisfactory except the area of high flow rates.

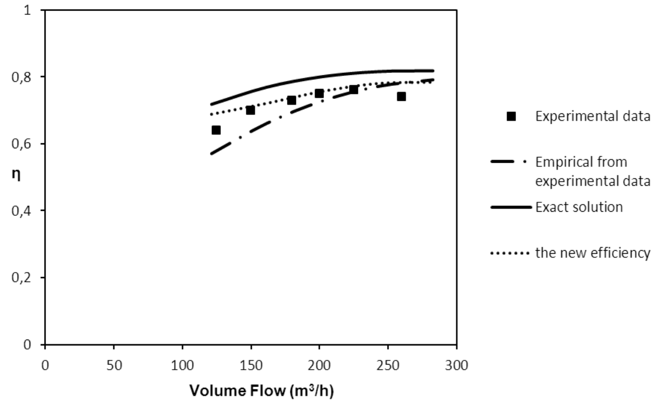

Fig. 4. Comparison between numerical results by the present method and experimental results in [20]

A fifth test case concerns the centrifugal pump impeller analyzed in [12]. Figure 5 presents the comparison between the three numerical models presented here and experimental data as well as numerical results obtained using 3D sophisticated numerical package including the effect of impeller-volute interaction.

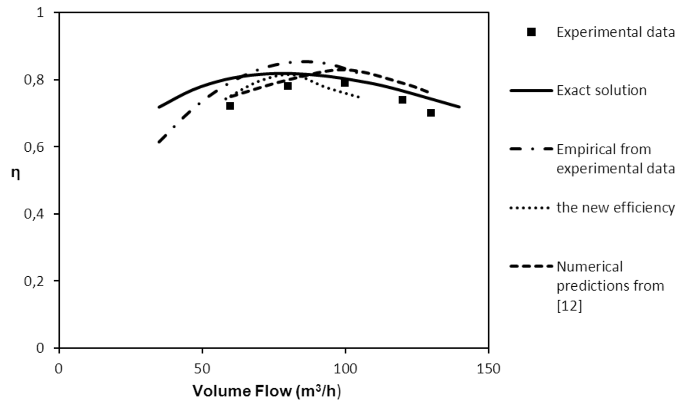

Fig. 5. Comparison between numerical results by the present method and experimental results in [12]

Observing this figure, one can see that the exact solution (solid line) shows a very good agreement to experimental data (squares), especially in medium and high mass flow rates. Numerical results according to empirical laws from experimental data (dash-dotted line) predict higher values of efficiency for all the range of volume flows. Predictions using the new empirical method agree well to experimental data. In the same figure, it is illustrated using dashed lines the prediction of pump efficiency from [12]. It can be seen that numerical predictions of characteristic lines considering the complete pump and solving the full Navier – Stokes equations with modeling of turbulence, leads to the most accurate numerical results, if one has the numerical tools for the grid generation and if he can afford the CPU time for these calculations.

Another test case is the centrifugal pump impeller analyzed in [10]. This is a 6-bladed centrifugal pump impeller with an inlet diameter $D_1 = 0.203\ m$, outlet diameter $D_2 = 0.489\ m$, rotating at *1490 rpm* and specific speed $n_s = 14.2$.

Figure 6 presents the comparison between numerical prediction using the three numerical models presented in this study and experimental results. One can see that the exact solution curve passes through the experimental points. Numerical prediction from experimental data follows better the trend of experimental data. The new efficiency method also predicts accurately the experimental data.

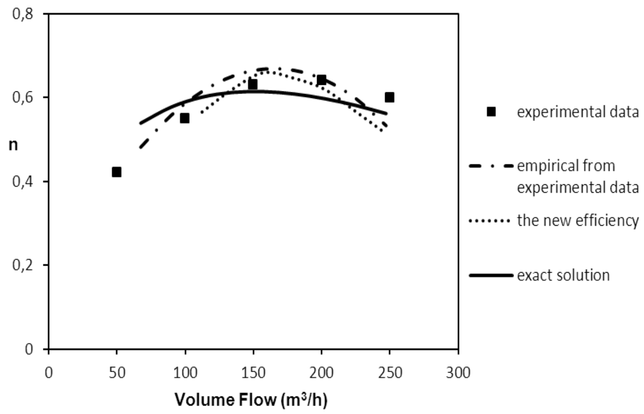

Fig. 6. Comparison between numerical results by the present method and experimental results in [10]

V. Conclusions

In this contribution, three different approaches to predict hydraulic efficiency of centrifugal pumps were presented. The first approach is based on exact solutions of the incompressible, steady state Navier-Stokes for the case of centrifugal pumps, employing the Bessel functions of the first kind. In order to express the pump head in terms of the flow rate, empirical relations are used. In these relations, the most important parameters are the pump impeller rotational speed, the number of impeller blades and the ratio of the inlet hub diameter to the outlet diameter. The pump efficiency is proportional to the integral of the manometric head. When performing the integration, the maximum volume flow is used as the integration constant.

The second method proposed here is an empirical equation derived from experiments done in the laboratory. This method

takes into account the maximum volume flow, the blade width at the impeller outlet, the number of impeller blades and the impeller outlet diameter.

The third method is a new empirical method proposed in the current study. The pump efficiency is expressed as the product of the efficiency depending on geometrical parameters and the one depending on volume flow and rotational speed of the pump impeller.

Test cases of six different centrifugal pump impellers with two-dimensional and three-dimensional blades are chosen to illustrate the ability of the method to predict the variation of the pump head in terms of the flow rate. In these test cases the specific speed is varying from 15 to 85. For all the cases examined, a very satisfactory agreement between numerical predictions and experimental data is found. The predicted results are in agreement to experimental data not only at the best efficiency poi nt, but also at higher and lower flow rates. This validates the method and makes it useful for industrial applications. The advantages of the present method are: (a) it requires a minimum of geometrical data and (b) it is simple and accurate and it can be used as a very quick global pump performance assessment tool, prior to a detailed investigation of the three-dimensional pump flow field, either numerically or experimentally.

References

[1] El-Naggar, M.A., "A One Dimensional Flow Analysis for the Prediction of Centrifugal Pump Performance Characteristics", International Journal of Rotating Machinery, Vol. 2013, Article ID 473512.

[2] Baun D.O. and Flack R.D., "Effects of volute design and number of impeller blades on lateral impeller forces and hydraulic performance", International Journal of Rotating Machinery, Vol. 9, No.2, 2003, pp.145-152.

[3] Yin, L., Yuan, S., Yue, T., and Jianping Y., "Research on the Energy Consumption Evaluation and Energy Saving Technical Reconstruction of Centrifugal Pump System Based on Actual Demand", Hindawi Publishing Corporation Advances in Mechanical Engineering Volume 2013, Article ID 423107.

[3] Liu H., Wang Y., Yuan S., Tan M. and Wang K., "Effects of Blade Number on Characteristics of Centrifugal Pumps", Chinese Journal of Mechanical Engineering, Vol. 23,No. *, 2010.

[4] Rababa, K.S., "The Effect of Blades Number and Shape on the Operating Characteristics of Groundwater Centrifugal Pumps", European Journal of Scientific Research, Volume 52 No.2, 2011, pp.243-251.

[5] Luo, X., Zhang, Y., Peng, J., Xu, H., and Yu, W., "Impeller inlet geometry effect on performance improvement for centrifugal pumps", Journal of Mechanical Science and Technology, Vol.22, pp.1971-1976, 2008.

[6] M.G.Patel M.G. and Doshi, A.V., "Effect of Impeller Blade Exit Angle on the Performance of Centrifugal Pump", International Journal of Emerging Technology and Advanced Engineering, Volume 3, Issue 1, January 2013.

[7] Gülich J.F., "Centrifugal Pumps", 2nd edition, Springer, Berlin, 2010.

[8] Li W-G, "Blade Exit Angle Effects on Performance of a Standard Industrial Centrifugal Oil Pump", Journal of Applied Fluid Mechanics, Vol. 4, No. 2, Issue 1, 2011 pp. 105-119.

[9] Kurokawa, J., Matsumoto, K., Matsui, J, Kitahora, T., "Performance of low specific speed pumps", 11th Australasian Fluid Mechanics Conference, Univeristy of Tasmania, Hobart, Australia, 14-18 December 1992.

[10] Fouaad, M., Adel, M. and Ashmawy, A., "CFD parametric simulation of low specific speed centrifugal pump", Journal of American Science, Volume 10, No. 12, 2014.

[11] Mentzos, M.D., Markopoulos, A.P., Galanis, N.I., Margaris, D.P., Manolakos, D.E., "Design, Numerical Analysis and Manufacture of Radial Pump Impellers with Various Blade Geometries", International Review of Mechanical Engineering (I.RE.M.E.), Vol. 9, No.1, pp.104-111, January 2015.

[12] Mentzos M., Filios, A., Margaris, D., Bacharoudis , E. and Vrachopoulos, M.Gr., "The use of CFD for flow analysis and performance prediction of centrifugal pumps", South-Eastern Europe FLUENT Event 2005, 11-13 May 2005, Greece.

[13] Mentzos, M., Markopoulos, A., Galanis, N., Margaris, D. and Manolakos, D., "Design, Numerical Analysis and Manufacture of Radial Pump Impellers with Various Blade Geometries", International Review of Mechanical Engineering, Vol.9, No. 1, pp. 104-111, January 2015.

[14] Zhou, L., Shi, W., and Wu, S., "Performance Optimization in a Centrifugal Pump Impeller by Orthogonal Experiment and Numerical Simulation", Hindawi Publishing Corporation Advances in Mechanical Engineering Volume 2013, Article ID 385809.

[15] Djerroud, M., Ngoma, G.D. and Ghie, W., "Numerical Identification of Key Design Parameters Enhancing the Centrifugal Pump Performance: Impeller, Impeller - Volute and Impeller - Diffuser", International Scholarly Research Network ISRN Mechanical Engineering Volume 2011, Article ID 794341

[16] Klas, R., Pochylý, F., and Rudolf, P. "Analysis of novel low specific speed pump designs" Presented at the 27th IAHR Symposium on Hydraulic Machinery and Systems (IAHR 2014), IOP Conf. Ser.: Earth Environ. Sci. 22 012010.

[17] Anagnostopoulos, J.,"CFD Analysis and Design Effects in a Radial Pump Impeller" , WSEAS Transactions on Fluid Mechanics, Issue 7, Vol. 1, 2006, pp.763-770.

[18] Shigemitsu T., Fukutomi J. and Kaji K.,, "Influence of Blade Outlet Angle and Blade Thickness on Performance and Internal Flow Conditions of Mini Centrifugal Pump", International Journal of Fluid Machinery and Systems, Vol. 4, No. 3, July-September 2011.

[19] Shojaee Fard, M.H., Boyaghci, F.A., "Studies on the influence of various blade outlet angles in a centrifugal pump when handling viscous fluids", American Journal of Applied Sciences, Vol. 4, No. 9, 2007, pp.718-724.

[20] Tan, L., Cao S., Wang Y. and Zhu, B., "Direct and inverse iterative design method for centrifugal pump impellers", Proc. IMechE. Part A: J.Power and Energy Vol. 226, No. 6, 2012, pp.764–775.

[21] Shekun, G.D., "Approximating the Efficiency Characteristics of Blade Pumps", Thermal Engineering, Vol. 54, No. 11, 2007, pp. 886–891.

[22] Sivakumar, V.R., Kumar, S., Mathews, J., "Analysis of Centrifugal Pump in Diffuser Vane By Using CFD", International Journal of Engineering Sciences & Research Technology, Vol. 3, No. 3, March 2014.

[23] Liu H., Liu M., Bay, Yu, Dong L., "Effects of mesh style and grid convergence on numerical simulation accuracy of centrifugal pump", J. Cent. South Univ. Volume 22, 2015, pp. 368−376.

[24] Fatsis, A., Panoutsopoulou, A., Vlachakis V. and Vlachakis N., "A Novel Methodology to Predict Centrifugal Pump Characteristics Through Navier-Stokes Exact Solutions", International Journal of Engineering Research & Technology (IJERT), Vol. 4 Issue 02, February 2015.

[25] Fatsis, A., Panoutsopoulou, A., Vlachakis, V. and Vlachakis, N. "A Practical method to predict performance curves of centrifugal water pumps", ASABE Transactions, Applied Engineering in Agriculture, Vol. 24(2), 2008.