International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 02, February-2015

A Novel Methodology to Predict Centrifugal Pump Characteristics Through Navier-Stokes Exact Solutions

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

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

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

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

***Abstract*—A novel method for centrifugal pump performance prediction through exact solutions of the Navier-Stokes equations is presented. Velocity components and the static pressure were obtained using the Bessel functions. The volume flow rate is obtained using empirical relations taking into account geometrical and operational parameters of the pump. Extensive validation of the numerical predictions against experimental data for various centrifugal pumps has revealed that the proposed method is reliable and it produces accurately the pump's characteristic line.**

Keywords — Centrifugal pump; exact solution; characteristic line; manometric head; flow rate.

I. Introduction

A wide variety of centrifugal pump types have been constructed and used in industrial and residential applications. Centrifugal pumps are widely used for irrigation, water supply plants, steam power plants, sewage, oil refineries, chemical plants, hydraulic power service, food processing factories and mines. Moreover, they are also used extensively in the chemical industry because of their suitability in practically any service related to it. Due to the vast applications it is very important that centrifugal pumps should work efficiently. In this context there have been continuous efforts to improve the performance of centrifugal pumps. The significant cost and time of the trial-and-error process by constructing and testing physical prototypes reduces the profit margins of the pump manufacturers. In recent years, a growing availability of computational resources and progress in the a accuracy of numerical methods brought Computational Fluid Dynamics (CFD) methods from pure research work into the competitive industrial markets. However, advanced CFD commercial software is not suited for a quick assessment of characteristic lines of a series of pumps, due to the detailed impeller blade camber, thickness and height distribution required for the grid generation. Alternatively, numerical models based on one-dimensional approach can provide very fast and accurate predictions of the pump performance.

II. Literature Survey

Many researchers developed simple one-dimensional computational models trying to predict the characteristic curves of centrifugal pumps. The one-dimensional model in [1] is simple and robust, giving accurate performance predictions. It is the basis for comparison of numerical predictions of characteristic lines, since is shows good agreement to manufacturer's data and experimental results. It is important that prediction models should include correlations accounting for loss mechanisms, since it was demonstrated in [2] that predictions based on the Euler's method and airfoil theory, sometimes produce unrealistic results.

The effect of blade exit angle variation on the head and efficiency of a centrifugal pump was investigated in [3] using the numerical model of [1]. It was concluded that the exit blade angle is influencing mainly at high volume flow rates the pump head and efficiency.

A one-dimensional performance analysis, [4], has proved to be an effective and important approach on pump design. It was concluded that pump characteristics depend on geometrical data and on losses in different parts of pump. Various types of losses of the basic model [1] were introduced in [5]. The results obtained show a good agreement to experimental results [6] for that test case.

A meanline pump flow modeling method has been developed in [7] aiming to provide a fast capability for predicting the performance of pumps at off-design operating conditions. The design-point rotor efficiency and slip factor were obtained from empirical correlations to rotor-specific speed and geometry.

A growing availability of computer power and progress in accuracy of numerical methods, brought turbomachinery CFD methods from pure research work into the competitive industrial markets. State of the art CFD solutions consist the solution of the Reynolds Averaged Navier Stokes Equations (RANS), for impeller flows as well as for complete pump flows including the impeller – volute interaction. In this context a promising alternative or supplement to standard Reynolds averaged Navier-Stokes (RANS) turbulence modeling is Large Eddy Simulation (LES).

Many software packages are available in the market for numerical analysis such as FLUENT, CFX, Fidap, Polyflow, Phoenix, Star CD, Flow 3d, ESI/CFDRC, SCRYU.

Many researchers, such [8], [9], [10], [11], using the CFX-TASCflow software carried out calculations of the flow inside the impeller, as well as considering the whole pump including the spiral casing. In these calculations, the standard *k-ε* two-equation turbulence model was used. The predictions were compared to performance experimental data showing a fairly good agreement.

Many researchers, such as [12], [13], [14], [15], [16], [17], [18], [19], [20] carried out the analysis of centrifugal pump flows using the ANSYS-CFX package solving the Reynolds Averaged Navier-Stokes (RANS) equations. The results obtained agree to available experimental data. The turbulence models used were the *k-ε* standard turbulence model and the *k-ω* based SST model.

The commercial FLUENT software was used by many researchers, such as [21], [22], [23], [24], [25], [26] to resolve the RANS equations for the whole pump. Numerical results compare well to experimental data and performance prediction data.

Numerical predictions of the flow in pumps were also performed using the Fine Turbo flow solver [27]. The computational grid was generated using IGG for the volute and Autogrid for the impeller. Reynolds Averaged Navier-Stokes equations in rotating frames of reference coupled with various turbulence models and near-wall treatment for low-Reynolds modeling.

Other researchers developed their own three-dimensional numerical tools to analyze the flow structure inside centrifugal pumps. A three-level optimization technique was developed in [28].

In very low specific speed range the efficiency of centrifugal pump, designed by the conventional method, becomes remarkably low. The recently used centrifugal pumps have a higher rotational speed and smaller size [29]. The internal flow of a very low specific speed centrifugal impeller has large influence on the pump performance, thus detailed information is needed for the relation between the internal flow and performance in order to develop high performance pumps.

A three dimensional incompressible thin-layer Navier-Stokes method has been developed in [30] for multistage unsteady turbomachinery flow calculations. The method is based on a combination between the pseudo-compressibility and the dual time stepping technique. Calculations were carried out for unsteady incompressible flow cases and the results show satisfactory agreement with well-established theoretical and experimental data. Sensitivity of the solution to various numerical variables was assessed for a centrifugal pump case.

A numerical model was developed in [31] for the numerical solution of the RANS equations in the impeller of a centrifugal pump. It was applied for direct flow analysis and parametric investigation of the effect of some impeller design details on its hydrodynamic characteristics.

A numerical simulation of the three-dimensional unsteady pump flow taking into account the impeller-volute interaction with different outlet blade angles using the SIMPLEC algorithm and the *k-ε* turbulence model, was done in [32]. Results show that when the blade outlet angle increases, the centrifugal pump performance handling viscous fluids improves.

In order to improve the accuracy of the numerical simulations and to be able to analyze and understand more thoroughly the flow in centrifugal pumps, it is essential to advance from a steady state to an unsteady simulation technique. In this context a promising alternative or supplement to standard Reynolds averaged Navier-Stokes (RANS) turbulence modeling is Large Eddy Simulation (LES).

LES method was applied in a Francis turbine [33] and in a centrifugal pump impeller, at design and off-design conditions.

Unsteady flow computations were done in a centrifugal pump impeller using the LES method [34]. It is thus found that LES provides an improved insight into the basic fluid dynamics with a satisfactory accuracy compared to experiments.

The present numerical method is an alternative to all those previously described numerical methods. It is based on the exact solution of the Navier-Stokes equations inside the pump impeller. The Bessel functions of the first order are used to provide approximate forms for the fluid velocity components and for the static pressure. In the following, an approximate form is used to derive the pump volume flow rate taking into account the inlet and outlet impeller diameter, the number of blades and the angular velocity of rotation of the impeller. Exhaustive testing of the model in ten different centrifugal pumps including comparisons with available experimental data and reference solutions, leads us to conclude that the present model can be used as a sound assessment tool to predict impeller characteristic lines.

III. GOVERNING EQUATIONS

A. *General Assumptions*

The present numerical model consists of the Navier-Stokes equations written in cylindrical coordinates, (r,θ,z), better suited for axisymmetric, rotating geometries than the Cartesian ones. Additionally, it is assumed 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 continuity equation is becoming:

$$\frac{u_r}{r}+\frac{\partial u_r}{\partial r}+\frac{\partial u_z}{\partial z}=0 \tag{1}$$

The system of the Navier-Stokes equations can be written:

r -Momentum:

$$u_r\frac{\partial u_r}{\partial r}-\frac{u_\theta^2}{r}+u_z\frac{\partial u_r}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial r}+\omega^2 r+2\omega u_\theta+\frac{\mu}{\rho}\left[\frac{\partial^2 u_r}{\partial r^2}+\frac{1}{r}\frac{\partial u_r}{\partial r}-\frac{u_r}{r^2}+\frac{\partial^2 u_r}{\partial z^2}\right] \tag{2}$$

θ -Momentum:

$$u_r \frac{\partial u_\theta}{\partial r} + \frac{u_r u_\theta}{r} = -\frac{1}{\rho}\frac{1}{r}\frac{\partial P}{\partial r} - 2\omega u_r + \frac{\mu}{\rho}\left[\frac{\partial u_\theta^2}{\partial r^2} + \frac{1}{r}\frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r^2}\right] \tag{3}$$

z -Momentum:

$$u_r \frac{\partial u_z}{\partial r} + u_z \frac{\partial u_z}{\partial z} = -\frac{1}{\rho}\cdot\frac{\partial p}{\partial z} + \frac{\mu}{\rho}\left[\frac{\partial^2 u_z}{\partial r^2} + \frac{1}{r}\cdot\frac{\partial u_z}{\partial r} + \frac{\partial^2 u_z}{\partial z^2}\right] \tag{4}$$

B. Non-dimensionalisation

The above system of partial differential equations can be written in non-dimensional form choosing the following parameters:

$$u_r^* = \frac{u_r}{\omega \cdot R_2}, u_\theta^* = \frac{u_\theta}{\omega \cdot R_2}, u_z^* = \frac{u_z}{\omega \cdot R_2} \quad z^* = \frac{z}{R_2} \quad r^* = \frac{r}{R_2}$$

where R_2 is a characteristic length of the geometry in consideration. The fluid pressure and the Reynolds number can then be written as:

$$p^* = \frac{p}{\rho \cdot (R_2 \cdot \omega)^2}, \qquad \mathrm{Re} = \frac{\rho \cdot R_2 \cdot \omega^2}{\mu}, \quad \omega = 2\cdot\pi\cdot n$$

C. Solution strategy

Resolving the system of equations (1) to (4) using the above non-dimensional quantities, it was found that the axial velocity u_z^*, the radial velocity u_r^* and the tangential velocity u_θ^*, can be expressed in terms of the functions:

$$u_z^* = J_0 \cdot e^{-b\cdot z} \tag{5a}$$

$$u_r^* = J_1 \cdot e^{-b\cdot z} \tag{5b}$$

$$u_\theta^* = -r \tag{5c}$$

where J_0 and J_1 are the Bessel functions of the First kind [35].

Substituting the non-dimensional velocity field to the Navier-Stokes equations (2) to (4), one obtains the static pressure field by solving the equation:

$$\frac{1}{2}\left(J_1^2 + J_0^2\right)\cdot e^{-2bz} = -p^*(r,z) - C - \frac{4\cdot z}{\mathrm{Re}} \tag{6}$$

where C is a constant. The determination of this constant involves some algebra and it will be omitted here. It is important to highlight that the methodology used, combines flow parameters (such as shut-off manometric height and maximum volume flow) and geometrical data of the pump impeller by means of an empirical relation which will be presented below.

Conservation of mass yields a relation between inlet and discharge meridional velocities that involves the cross-sectional areas of the streamtube at these locations. The volume flow rate through the pump is related to the velocity distribution at any location by the integral [36]:

$$\dot{V} = \int_{R_1}^{R_2} \dot{V}_i \, dr_i = \int_{R_1}^{R_2} 2\pi\, r\, u_m dr_i \tag{7}$$

where u_m is the meridional component of the flow velocity and $\dot{V}_i$ is the volume flow through a streamtube from an intermediate radius r_i to radius $r_i + dr$. Integrating for the whole length of each streamline between R_1 and R_2 gives the volume flow $\dot{V}$.

The limiting case that corresponds to $\dot{V} = 0$ leads us to determine the shut-off manometric head, $H_{shut-off}$.

As it is suggested in [1], the volume flow is proportional to the third power of the exit pump impeller diameter. Trying to account for the effect of the rotational speed of the impeller, the number of impeller blades and the ratio of the inlet hub diameter to the outlet diameter, the maximum volume flow is approximated here by:

$$\dot{V}_{\max} = 0,08\cdot n\cdot D_2^3 \cdot\left(\frac{u_2}{u_1}\right)\cdot\left[\left(\frac{g\cdot H_{shut-off}}{0,8\cdot u_2^2}\right)\cdot\left(\frac{n}{1200}\right)\cdot\left(\frac{z}{7}\right)\cdot\left(\frac{D_1}{D_2}\right)\right] \tag{8a}$$

for $D_1 > \dfrac{D_2}{2}$, while

$$\dot{V}_{\max} = 0,08\cdot n\cdot D_2^3 \cdot\left[\left(\frac{g\cdot H_{shut-off}}{0,8\cdot u_2^2}\right)\cdot\left(\frac{n}{1200}\right)\cdot\left(\frac{z}{7}\right)\cdot\left(\frac{D_1}{D_2}\right)\right] \tag{8b}$$

for $D_1 < \dfrac{D_2}{2}$

where u_2 is the tangential velocity at the pump impeller outlet:

$$u_2 = \pi \cdot D_2 \cdot n$$

D_1, D_2 is the impeller inlet and outlet diameter, respectively and z is the number of pump impeller blades.

Based on [3], the effect of the blade exit angle, β_2, which is not taken into account in the current work, has little influence to pump performance in the expected range of operation (close to the design point) - and it is limited mainly in high flow rates.

Having determined the pressure from equation (6), the manometric head can be calculated from:

$$p^* = \frac{\Delta p}{\frac{\rho}{2}\cdot\omega^2\cdot R_2^2} = \frac{\rho\cdot g\cdot H}{\frac{\rho}{2}\cdot R_2^2\cdot(2\cdot\pi\cdot n)^2} = \frac{g\cdot H}{\frac{1}{2}\cdot\frac{D_2^2}{4}\cdot 4\cdot\pi^2\cdot n^2}$$

$$\Rightarrow H = \frac{1}{2}\cdot\left[\left(J_1^2 + J_0^2\right)e^{-\frac{4b\Delta z}{D_2}} - \frac{8\cdot\Delta z}{\mathrm{Re}\cdot D_2}\right]\cdot\left(n\cdot D_2\right)^2 \qquad (9)$$

where Δz is the axial gap between the impeller disk and casing.

IV. Results

A first test case to assess the applicability of the present method is the prediction of the 7-bladed centrifugal pump in [37]. The hub diameter is $D_h = 0.3m$, suction diameter is $D_s = 0.405m$, impeller diameter is $D_2 = 0.169m$ and impeller width is $b_2 = 0.05m$. The rotational speed is *2900rpm*, the design point flow rate is *20m³/h*, the head is *35m*. The pump specific speed is $n_q = 15$, defined as:

$$n_q = RPM\,\frac{\sqrt{\dot{V}}}{H^{0,75}}$$

A comparison between prediction of manometric head obtained using the present method (illustrated with solid line) and experimental results (illustrated by dots) found in [37] is shown in Fig. 1. From this figure, one can see that the results obtained using the present method show a very good agreement with the available experimental data.

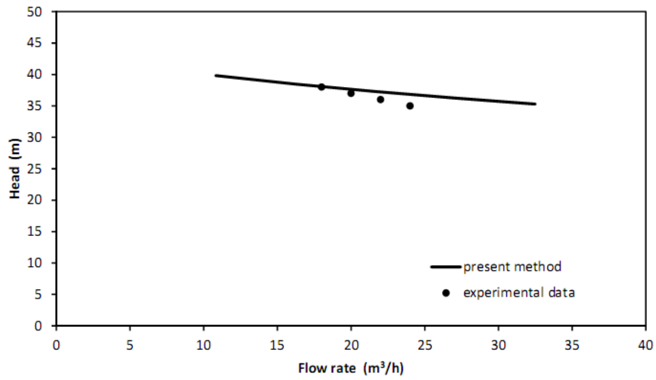

Fig. 1. Comparison between numerical results by the present method and experimental results in [37]

A second test case is the pump studied in [38]. The centrifugal impeller consists of six two-dimensioned curvature backward swept blades of constant thickness. The inlet height b_1 is *15.13 mm* and the outlet height b_2 is *8.11 mm* and the angular speed is *1000 rpm*. The centrifugal impeller was manufactured of acryl for PIV measurements and operates at flow rate of *2m³/h* at design point.

Fig. 2 shows the comparison between results of the present method and experimental data [38]. One can see that the present numerical data show the same trend as the experimental data. Slight differences occur at lower than the design point flow rates.

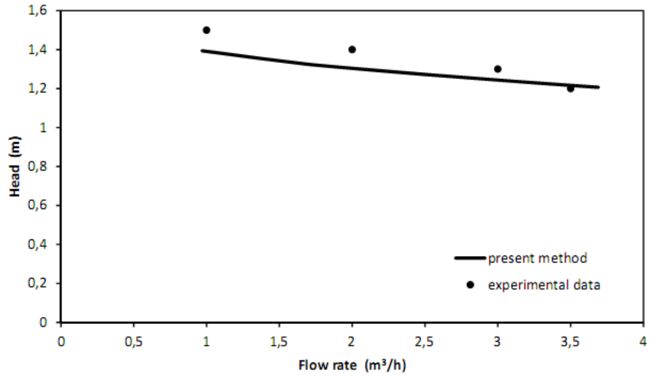

Fig. 2. Comparison between numerical results by the present method and experimental results in [38].

A third test case is the 6-bladed centrifugal pump impeller analyzed in [39]. The pump examined operates at high volume rates, having at design point manometric head of *31.5m* and a flow rate of *225.8m³/h*. Comparisons between prediction using the present method and experimental data are shown in Fig. 3. One can see a slight over-prediction between the present method and experimental data, without altering the behavior of the predicted distribution for this high flow rate centrifugal pump.

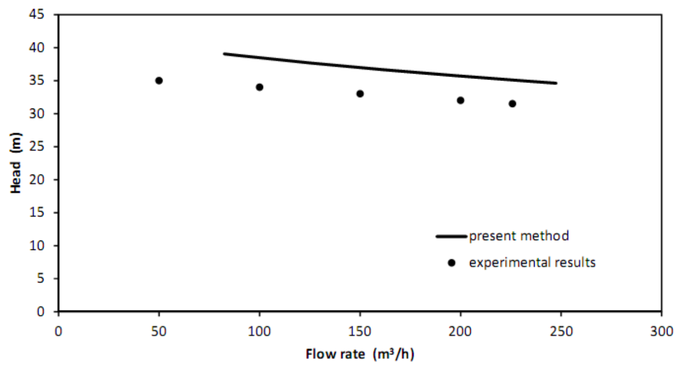

Fig. 3. Comparison between numerical results by the present method and experimental results in [39]

A fourth test case is a 6-bladed centrifugal impeller with specific speed $n_q = 38.47$ that studied in [3]. In Fig. 4, on can see numerical prediction obtained by the present method, compared using manufacturer data [3]. Prediction obtained using the present method show quite good agreement to manufacturer's data for all flow rates examined.

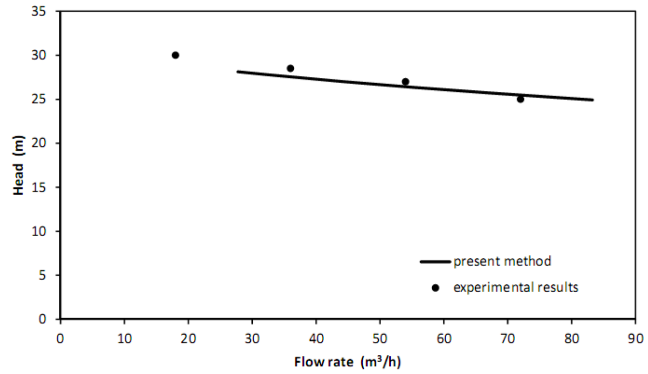

Fig. 4. Comparison between numerical results by the present method and experimental results in [3]

A fifth test case is a centrifugal oil pump analyzed in [40]. This is a single stage standard industrial centrifugal oil pump with side-suction entry, which has been applied to transport clean crude oil and other liquid petroleum products. The pump duty specifications are as follows: the nominal flow rate is $25m^3/h$, the head is *60 m*, the rotating speed is *2950 rpm* and the specific speed $n_q = 41.6$. The impeller was of closed type with five twisted three-dimensional blades. Fig. 5 shows the comparison between prediction of the present method and experimental data found in [40]. Once more, numerical data show a very good agreement to experimental data.

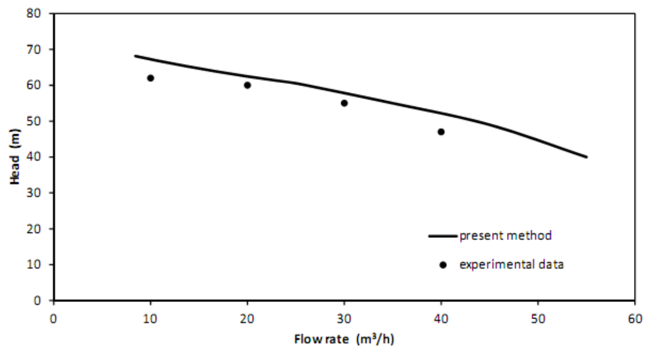

Fig. 5. Comparison between numerical results by the present method and experimental results in [40].

A sixth test case is one of the impellers analysed by Fard [41]. It is a 6-bladed centrifugal pump impeller with backwards curved blades, running at *1450 rpm*. Fig. 6 shows comparison between numerical results by the present method, experimental results for this pump [41]. From this figure, one can see that the present method shows a very good agreement to pump experimental data.

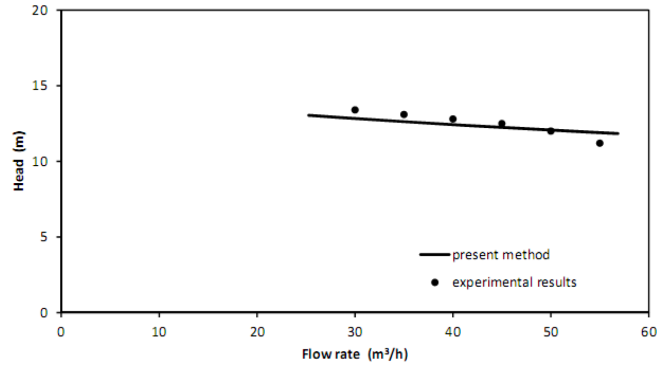

Fig. 6. Comparison between numerical results by the present method and experimental results in [41].

A seventh test case is a pump geometry found in the textbook of [42]. Figure 7 shows comparison between numerical results by the present method, experimental results and reference numerical results for the pump of [42]. The present method predicts accurately the characteristic of the pump. At high flow rates, one can observe numerical results predict slightly higher manometric height.

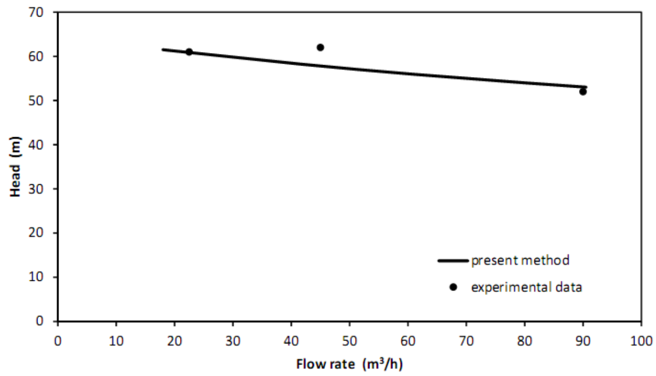

Fig. 7. Comparison between numerical results by the present method and experimental results in [42]

The FLUENT package was used in [43] to study the three-dimensional turbulent flow through a commercial water pump at design and off-design conditions. The selected 6-bladed commercial pump has a specific speed of $n_q = 22.86$.

Fig. 8 shows the comparison between numerical results obtained by the present method, experimental data and numerical results solving the Navier-Stokes equations using a standard *k-ε* two-equation turbulence model [43]. The predicted results are presented in terms of pressure profiles, velocity vectors and performance curves. The computed performance characteristics are in good agreement with the manufacturer curves and almost coincide in the region of the design volumetric capacity. Deflecting from this point either to lower or to higher flow rates, the head prediction is either underestimated or overestimated with reference value the value of head at volume flow of $90m^3/h$.

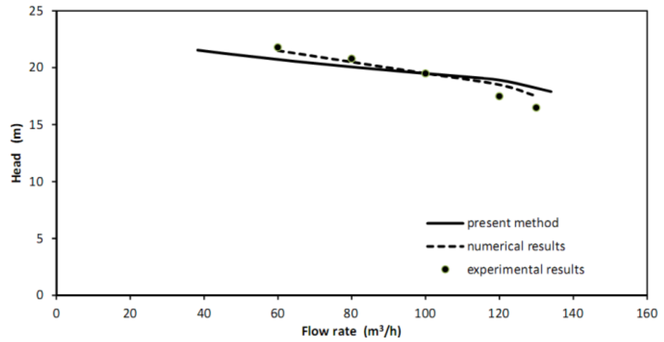

Fig. 8. Comparison between numerical results by the present method, experimental results and reference numerical results in [43]

Figure 9 presents the comparison between experimental data and numerical prediction for the 7-bladed pump analyzed in [44]. This is a centrifugal pump with a specific speed of $n_q = 24.17$. Results obtained by the present method show a very good agreement to experimental data.

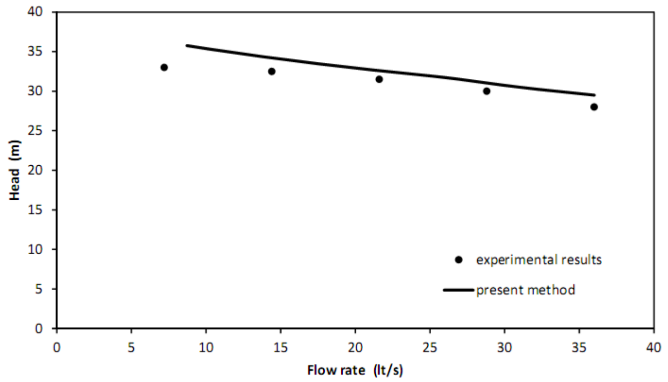

Fig. 9. Comparison between numerical results by the present method and experimental results in [44]

Figure 10 shows the comparison between numerical predictions and experimental data for a 5-bladed slurry pump, [45], with a specific speed of $n_q = 85$. One can see that the numerical results using the present method agree very well with experimental data over all flow regimes.

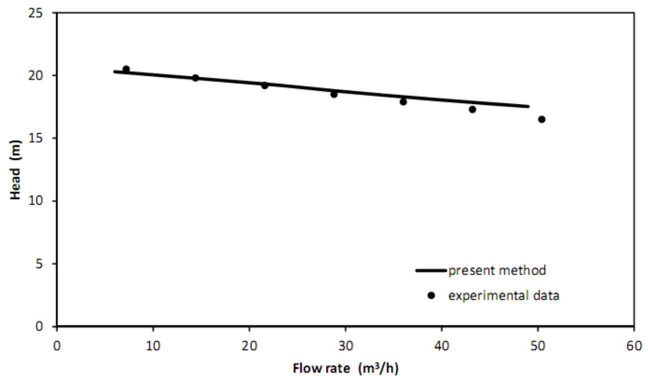

Fig. 10. Comparison between numerical results by the present method and experimental results in [45]

V. Conclusions

In the present article, a novel approach to predict characteristic lines for centrifugal pumps is presented. This approach is based on exact solutions of the incompressible, steady state Navier-Stokes for the case of centrifugal pumps. Velocity components and pressure are obtained using the Bessel functions of the first kind. The flow rate is found by integrating the meridional velocity from hub to tip. 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. Test cases of ten 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 and the number of blades from 5 to 7. 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 point, but also at higher and lower flow rates. This validates the method and makes it useful for industrial applications. An opportunity to improve the predictions could be to include the effect of the exit blade angle to the model. 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] Gülich J.F., "Centrifugal Pumps", 2nd edition, Springer, Berlin, 2010.
[2] Yedidiah, S. "An overview of methods for calculating the head of a rotordynamic impeller and their practical significance". Proc. Instn. Mech. Engrs, Part E, J. Process Mechanical Engineering. Vol. 217, No. 3, 2003, pp. 221-232.
[3] 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.
[4] Khin Cho Thin, Mya Mya Khaing, and Khin Maung Aye, "Design and performance analysis of centrifugal pump", World Academy of Science, Engineering and Technology, Vol. 36, 2008, pp. 422-429.
[5] 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.
[6] 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.
[7] Veres, J.P., "Centrifugal and Axial Pump Design and Off-Design Performance Prediction", NASA TM 106745, 1994.
[8] Praetor, R, Kecke H.J., "Unterschiedliche Berecnhungsmöglichkeiten von radialen Kreiselpumpen", Chemie Ingenieur Technik, Vol.73, 2001.
[9] Meakhail, T.A., Salem, M., Shafie, I., "Steady and unsteady flow inside a centrifugal pump for two different impellers", International Journal of Energy and Power Engineering, Vol. 3, No. 2, 2014, pp.65-76.

[10] Zhou W., Zhao Z., Lee T.S. and Winoto S.H., "Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics", International Journal of Rotating Machinery, Vol. 9, No. 1, 2003, pp.49–61.

[11] Spence R., Amaral-Teixeira J., "A CFD parametric study of geometrical variations on the pressure pulsations and performance characteristics of a centrifugal pump", Computers & Fluids, Vol. 38, 2009, pp.1243–1257.

[12] Rajendran, S., K. Purushothaman, K. , "Analysis of a centrifugal pump impeller using ANSYS-CFX". International Journal of Engineering Research and Technology, Vol.1, Issue 3, 2012.

[13] Munish Gupta, Satish Kumar, Ayush Kumar "Numerical Study of Pressure and Velocity Distribution Analysis of Centrifugal Pump" International Journal of Thermal Technologies, Vol.1, No.1, 2011.

[14] Budea Sanda, Carbune Varzaru Daniela, "The influence of the inlet angle over the radial impeller geometry design approach with ANSYS" Journal of Engineering Studies and Research, Vol. 18, 2012.

[15] Gaurav M, Vadaliya, A. "Parametric study of centrifugal pump impeller – A review", International Journal of Advanced Research and Technology, Volume 2, Issue 2, 2014, pp.1-4.

[16] Gantar, T., Sekavcnik, M. "Hydraulic Behavior of Unconventional Radial Pump Stages", Transactions of the CSME, Vol. 32, No.2, 2008, pp. 155-180.

[17] 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, 2008, pp. 1971-1976.

[18] Atif, A., Benmansour S. and Bois, G., "Numerical Investigation of Velocity Flow Field inside an Impeller Air Model of a Centrifugal Pump with Vaned Diffuser Interactions and Comparison with PIV Measurements", Hindawi Publishing Corporation, International Journal of Rotating Machinery, Volume 2010, Article ID 706043.

[19] Kaewnai, S., Chamaoot, M. and Wongwises, S., "Predicting performance of radial flow type impeller of centrifugal pump using CFD", Journal of Mechanical Science and Technology 23, 2009, pp.1620-1627.

[20] Cheah, K.W., Lee T.S., Winoto S.H. and Zhao, Z.M., "Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions", Hindawi Publishing Corporation, International Journal of Rotating Machinery, Volume 2007, Article ID 83641.

[21] Bacharoudis, E.C., Filios, A.E. Mentzos, M.D. and Margaris, D.P., "Parametric study of a centrifugal pump impeller by varying the outlet blade angle", The Open Mechanical Engineering Journal, 2008, pp.75-83.

[22] Mununga, L., Hourigan, K. and Thompson, M. "Numerical study of the effect of blade size on pumping effectiveness of a paddle impeller in an unbaffled mixing vessel." Third International Conference on CFD in the Minerals and Process Industries, CSIRO Melbourne, Australia, December 2003.

[23] Auvinen, M. "Computational analysis of a single-blade sewage pump with FINFLO." Helsinki University of Technology Laboratory of Applied Thermodynamics, CFD Group, 6th March 2006.

[24] González-Pérez J., Parrondo J., Santolaria C, Blanco E.,"Steady and unsteady radial forces for a centrifugal pump with impeller to tongue gap variation", ASME Journal of Fluids Engineering Vol. 128, 2006, pp.454–462.

[25] Georgiev, D., Vlaev, S., "Flow field characteristics of modified inclined-blade mixing impellers", Journal of the University of Chemical Technology and Metallurgy, Vol. 41, No.1, 2006, pp. 45-50.

[26] Kergourlay, G., Younsi, M., Bakir, F., and Rey, R., "Influence of Splitter Blades on the Flow Field of a Centrifugal Pump: Test-Analysis Comparison"., International Journal of Rotating Machinery, Vol. 2007, Article ID 85024.

[27] Derakhshan S., Nourbakhsh, A., "Theoretical, numerical and experimental investigation of centrifugal pumps in reverse operation", Experimental Thermal and Fluid Science, Vol. 32, 2008, pp.1620–1627.

[28] Thum, S., Schilling, R, "Optimization of hydraulic Machinery bladings by multilevel CFD Techniques", 9th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, Honolulu, Hawaii, February 10-14, 2002.

[29] Choi, Y-D, Kurokawa J., Matsui J., "Performance and internal flow characteristics of a very low specific speed centrifugal pump", ASME Journal of Fluids Engineering, Vol. 128, 2006, pp.341-349.

[30] He, L, Sato, K, "Numerical Solution of Incompressible Unsteady Flows in Turbomachinery", Journal of Fluids Engineering, Vol. 123, 2001, pp.680-685.

[31] 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.

[32] 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.

[33] Song, C.C.S., Chen, X., Ikohagi, T., Sato, J., Shinmei, K. and Tani, K., "Simulation of Flow through Francis Turbine by LES Method" Proceedings of 18th IAHR Symposium on Hydraulic Machinery and Cavitation, Iberdrola, 1996, pp.267–276.

[34] Byskov, R.K, Jacobsen C.B. and Pedersen, N., "Flow in a Centrifugal Pump Impeller at Design and Off-Design Conditions—Part II: Large Eddy Simulations", Journal of Fluids Engineering, Vol.125, 2003, pp.73-83.

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

[36] Brennen, C.H., "Hydrodynamics of Pumps", Concepts ETI and Oxford University Press, 1994.

[37] Derakhshan, S., Maryam Pourmahdavi, M., Abdolahnejad, E., Reihani, A., Ojaghi. A., "Numerical shape optimization of a centrifugal pump impeller using artificial bee colony algorithm", Computers & Fluids, Vol.81, 2013, pp. 145–151.

[38] Wu Y. L., Liu S.H., Yuan H.J., Shao J. "PIV measurement on internal instantaneous flows of a centrifugal pump", Sci. China Tech. Sci., Vol.54, No.2, 2011, pp. 270–276.

[39] 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.

[40] 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.

[41] Fard, M.S., Boyaghchi, F.A., "Studies of the influence of various outlet angles in a centrifugal pump when handling viscous fluids", American Journal of Applied Sciences, Vol. 4, No. 9, 2007, pp.718-24.

[42] Sigloch, H., "Strömungsmaschinen: Grundlagen und Anwendungen", Hanser, 1984

[43] 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.

[44] 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

[45] Shi, B. and Wei, J., "Numerical Simulation of 3D Solid-Liquid Turbulent Flow in a Low Specific Speed Centrifugal Pump: Performance Comparison of Four Geometric Models", Advances in Mechanical Engineering, Vol. 2014, Article ID 678271, Hindawi Publishing Corporation.