Parametric CFD Study of the Hydraulic and Energetic Performance in Centrifugal Pumps as a Function of Geometrical Parameters

Jorge Duarte¹, Guillermo E. Valencia² and Luis G. Obregón³

¹ Design of Mechanical and Robotic Systems for Industrial Production – DIMER Research Group, Universidad del Atlántico, Carrera 30 Número 8 - 49, Puerto Colombia – Colombia

² Mechanical Eng., Efficient Energy Management Research Group – Kaí Universidad del Atlántico, Carrera 30 Número 8 – 49 Puerto Colombia – Colombia

³ Sustainable Chemical and Biochemical Processes Research Group Universidad del Atlántico, Carrera 30 Número 8 – 49 Puerto Colombia – Colombia

Copyright © 2018 Jorge Duarte, Guillermo E. Valencia and Luis G. Obregón. This article is distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium provided the original work is properly cited.

Abstract

In this work, the hydraulic and energetic performance of a centrifugal pump is studied in CFD (Computational Fluid Dynamics), to achieve optimization in its performance through a parametric analysis. Variations were made in pump parameters such as diameter, number of blades and speed of rotation, to study their influence over the general performance of the pump, by using a CFD package; the results obtained were validated in a test bench, to guarantee the prediction capacity of the simulations. In the same way, a theoretical analysis was carried out, comparing the results with the Euler equation and corrections with Stodola equation. An analysis of mesh independence was carried out, to optimize the simulation process and computational resources. The results allowed to conclude that the reduction in the thickness of the pump casing, which has the effect of a larger diameter of the impeller, leads to an increase in the effective head of the pump; similar results are obtained with the increase in the number of blades and the
increase in the speed of rotation, although in the latter case, it is less accentuated due to the reduction in volumetric efficiency.

**Keywords**: CFD, Optimization, Performance, Pump, Simulation

### 1. Introduction

Centrifugal pumps are the basis for a vast amount of applications that involve transport of fluids, such as agriculture, industry, and domestic use. Because of that, this kind of machines is the main focus of design optimization processes, with obtaining a more efficient design as their goal. However, to reduce manufacturing costs and resources, it’s crucial to predicting their performance in advance, which requires an understanding of the flow behavior in different parts of the pump. Experimental model testing is one of the solutions for prediction of performance, but it is tedious, time-consuming and costly; in the other hand, a theoretical approach gives a result, but it is unable to determine the origin of a poor pump performance [1].

To solve that problem, the development of computer technology allowed the growth and expansion of CFD (Computational Fluid Dynamics), as a third alternative to predict pump performance and to obtain accurate results with a relatively low cost [2]. The basis of CFD, involve the numerical solution of the equations that describe the flow field in a control volume (as the Navier-Stokes and turbulence equations), in order to calculate fluid properties as pressure, temperature, density, etc., in a certain point of interest, achieving results in less time than experimental measurements, with a comparable precision.

Applied to pump performance prediction, the importance and potential of CFD resides in the parametric design, a tool which allows to study the effects of various geometrical parameters on pump performance, without the need to do experimental tests [1]; this represents a saving of time and production costs, and its widely used for two purposes: to evaluate a brand new design, and to optimize existing designs. Since the impeller geometry plays a major role in the centrifugal pump, any modification in the impeller geometry would have an impact on the impeller inlet or exit velocity triangles, which may result in significant changes in the performance. Therefore, many investigations use geometrical parameters of the impeller, like the outlet angle passage [3], wrap angle [4], width of the impeller blade at exit [5], blade thickness [6], and others [7]-[12], as a starting point for parametric designs [2]; however, due to the huge variety of designs and parameters of centrifugal pumps, the parametric design still has a lot of combinations to evaluate, in order to accurately predict pump performance in any of these.

In this article, the influence of three parameters on the pump performance was studied: the impeller speed of rotation and diameter, and the number of blades. The results of the simulation were validated using data from the fabricant and compared with a couple of theoretical models, with the purpose of evaluating the precision of the data obtained by these models, and to make visible the importance of the CFD numerical methods in the parametric design.
2. Methodology

2.1. Development of the study

CAD model for a radial pump was generated and geometrically characterized, by using the SOLIDWORKS® package. Following that, a control volume was defined in the vicinity of the pump impeller, which is the region where the energy transfer takes place; due to the impeller geometry, a counter-clockwise rotation was defined, to emulate the best condition of operation. The CAD model generated, was used in the SOLIDWORKS® Flow Simulation add-on to simulate a set of fixed points, which are used to plot the pump head against the volumetric flow. A level 4 mesh was used in the simulations, and regarding the boundary conditions, an inlet pressure of 101.325 kPa was used, while the second condition (volumetric outlet flow) was defined for each point of the study, according to the information provided in the pump datasheet. To ensure the convergence of the solution, the outlet pressure was monitored, with a convergence criterion of $10^{-4}$ was considered for the normalized sum of the residuals of the discretized equations.

To validate the results from the simulation, the results obtained were plotted against the characteristic curve given by the manufacturer, together with the characteristic curves obtained with the Euler and Stodola equations [13]. After validated, the simulation was used in a parametric study, changing the values of three parameters of the impeller: impeller speed of rotation, impeller diameter, and the number of blades. For each modified parameter, the characteristic curve was obtained, to compare the influence of the changes in the parameter, over the curve.

2.2. Fundamental Equations

The fundamental equations used in the study have been widely studied in literature. For the development of a theoretical model of a centrifugal pump, it’s required to make some assumptions to simplify the real case, like the following:

- The fluid is incompressible
- The fluid enters the pump in the radial direction
- There are infinite blades in the impeller
- The thickness of impeller blades can be considered negligible

With this assumption in mind, the next step is to define the velocity triangles of both inlet and outlet; taking in account the assumptions mentioned, the triangles of velocity for the actual case, are illustrated in the figures 1a and 1b, where $c$, $u$, $w$ are the absolute, circumferential and relative velocity vectors, respectively, and $\beta$ is the exit angle of the fluid. It’s possible to apply the main equation of pump analysis, the Euler equation:

$$H_{u\infty} = \frac{u_1 \cdot c_1 - u_2 \cdot c_2}{g}$$

(1)
Figure 1. Velocity triangles for the inlet (a) and the outlet (b).

Though the Euler equation is commonly used in the study of centrifugal pumps, one of its main assumptions (the infinite number of blades) generates a huge difference between the calculated pump head and the theoretical result, because that assumption implies a greater momentum transmission to the working fluid. Therefore, to reduce this effect, the Stodola correction was proposed, which applies a correction factor to estimate the right value of pump head for a finite number of blades; this head relates to the head obtained by (1) by the equation

\[ H_u = e_z \cdot H_{u\infty} \] (2)

The term \( e_z \), also called the work reduction coefficient, is calculated with the expression

\[ e_z = 1 - K_R \cdot \frac{\pi \cdot \sin(\beta_2)}{z} \cdot \frac{u_2}{c_{2u}} \] (3)

Where \( z \) represents the number of blades in the pump, and \( K_R \) is the correction coefficient of the model [13].

3. Results and discussion

Below are the results of the main points of the present study:

3.1. Mesh independence study

To optimize the time and computational resources, a mesh independence check was carefully carried out before analyzing the simulation results. For each point of the simulation, the convergence criterion of \( 10^{-4} \) was achieved, with the monitor point for the outlet pressure. Three different meshes were used to satisfy the grid requirements for the solution approaches; a mesh with a total of 1755523 elements or cells was utilized as the best compromise between solution-accuracy requirements and computational resources as shown in Figure 2.
3.2. Validation of the simulation

For validation purposes, Figure 3 shows the characteristic curves obtained from the simulation along with the experimental values and the predictions from the Euler/Stodola equations, for the parameters \( \omega=3450 \) rpm, \( d=145 \) mm and \( z=7 \) blades. It is evident that the results from the simulation resemble the experimental data closely, while the theoretical models gave results too separated from the real curve. However, this can be explained as a consequence of the assumptions of each model, which doesn’t take account for the blade thickness and the difference of velocity between the frontal and posterior faces of the impeller.

3.3. Case study: Variation of the speed of rotation

To study the influence of the speed of rotation on the pump performance, the characteristic curve was obtained at three values of speed: 2800, 3450 and 4100 rpm, maintaining constant the other parameters (\( d=145 \) mm and \( z=7 \) blades). The results, shown in Figure 4, made evident that the increase in speed of rotation, generates a displacement of the characteristic curve in the same direction as the change in speed, which goes by the dynamic similitude expressions; however, to higher flows, the magnitude of the displacement reduces considerably.
3.4. Case study: Variation of impeller diameter

To study the influence of the impeller diameter on the performance, the characteristic curve was obtained at three values of diameter: 141, 145 and 149 mm, maintaining constant the other parameters. The results, shown in Figure 5, made evident that the increase in the diameter, increases the total head of the system, due to the change in circumferential velocity of the fluid, and goes in concordance with the similitude expressions for this kind of machines.

3.5. Case study: Variation of number of blades

To study the effect of the number of blades, three values were analyzed: 6, 7 and 8 blades. The results, shown in Figure 6, state the existence of an inverse proportional relationship between the number of blades and the performance; contrary to the assumption of infinite blades, a finite number of these generate variations of pressure in the fluid, with a low effect on the outlet velocity. These pressure differences are lower with a higher number of blades, which is in concordance with the Stodola equation.
4. Conclusions

The parametric study developed with CFD allowed to study the performance of a centrifugal pump at different geometric values and operating conditions, to obtain accurate results without experimental tests, which involves a considerable reduction in resources, keeping the quality in the design and manufacture of these machines. Besides, the simulation in CFD represents an advantage before the theoretical models due to its ability to integrate a complete model in a numerical solution scheme, compared to the simplifications required by the theoretical models for their formulation. For the same design, the pump performance varies considerably depending on the adjusted parameter, being the speed of rotation the most influencing, mainly at low volumetric flows (though the impeller diameter and the number of blades also have some effect on the performance). For the pump analyzed, it was found that the performance varies proportionally with the changes in the parameters. However, with the number of blades, an inverse proportional relationship was found, where the performance reduces with the increment in the number, which was in total concordance with the finite blade number theoretical models.

References


Received: April 10, 2018; Published: May 4, 2018