Computers & Fluids 60 (2012) 61–70

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid

M.H. Shojaeefard [a], M. Tahani [a,*], M.B. Ehghaghi [b], M.A. Fallahian [a], M. Beglari [a]

[a] School of Mechanical Engineering, Iran University of Science and Technology, Tehran, Iran
[b] Department of Mechanical Engineering, Tabriz University, Tabriz, Iran

ARTICLE INFO

Article history:
Received 17 July 2011
Received in revised form 30 November 2011
Accepted 27 February 2012
Available online 7 March 2012

Keywords:
CFD
Turbulence modeling
Hydraulic oil
Centrifugal pump
Geometry

ABSTRACT

The performance of centrifugal pumps drops sharply during the pumping of viscous fluids. Changing some geometric characteristics of the impeller in these types of pumps improve their performance. In this investigation, the 3-D flow in centrifugal pump along with the volute has been numerically simulated. This numerical solution has been carried out for different cases of primary geometry, and for the changes made to the outlet angle and passage width of the impeller, and also for simultaneous modifications of these parameters. The finite volume method has been used for the discretization of the governing equations, and the High Resolution algorithm has been employed to solve the equations. Also, the "$k-\omega$ SST" has been adopted as the turbulence model in the simulation. In the steady state, this simulation is defined by means of the multi-reference frame technique, in which the impeller is situated in the rotating reference frame, and the volute is in the fixed reference frame, and they are related to each other through the "Frozen Rotor". The obtained numerical results are compared with the experimental ones, and the outcome shows acceptable agreement between the two. The flow analysis indicates that with the modification of the original geometry of the pump, at the 30° outlet angle and the passage width of 21 mm, the pump head and efficiency increases compared to the other cases; this improvement is due the reduction of losses arising from the generation of eddies in the passage and outlet of the impeller.

1. Introduction

Due to the large complexity of flow and geometry in the radial flow pumps, there are still many unknown issues associated with the complete flow pattern in these pumps, which need to be investigated. Moreover, the conduction of experimental studies on samples with different volute and impeller geometries is time-consuming and costly, and because of the complicated geometry, it is not possible to carry out a thorough investigation of the flow field for a vast number of operating conditions. Therefore, the numerical flow analysis has recently become an appropriate method of investigation of the flow patterns and losses.

Based on the main pioneering researches on the centrifugal pump handling viscous fluid (since 1926), the performance as a function of oil viscosity was investigated, and the obtained results were used for the design and selection of these pumps [1–4]. Today, because of drastic changes in the design and structure of newer models of hydraulic pumps, the previously obtained results on older models cannot be used with high confidence.

Li [5–10] investigated the effects of fluids viscosity on the performance of centrifugal oil pumps experimentally and numerically; he presented the flow pattern inside the impeller. According to these researches, the high viscosity results in rapid increases in the disc friction losses over outside of the impeller shroud and hub as well as the hydraulic losses in flow channels of the pump. The viscosity of fluid not only affects the slip coefficient, but also causes the reduction of flow in the impeller and volute. Furthermore, he presented that there is a wide wake near the blade suction side of the centrifugal pump impeller. Also, there is not a jet near the blade pressure side, and the flow pattern is essentially different from the well-known jet/wake model. He obtained the optimum number of blades for the impeller when fluids with different viscosities are pumped.

In the year 2004 Asuaje et al. [11] conducted 3D flow solution by CFD tools. In this research, a design procedure was established. This method was based on the geometrical design and the performance analysis. Their design tool took into account models and correlations resulting from experimental data dealing with many ranges of industrial centrifugal pumps which constitute a significant database.

In 2007, Kergourlay et al. [12] investigated the effects of separated blades on the flow field of water in centrifugal pumps. According to this research, adding the splitters has negative and positive effects on the pump behavior. It increases the head rise compared to the original impeller that is mainly due to the

* Corresponding author. Tel.: +98 2177240360.
E-mail address: Tahani@iust.ac.ir (M. Tahani).

0045-7930/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2012.02.028

Nomenclature

b_2	passage width of Impeller (mm)
BEP	best efficiency point
CFD	computational fluid dynamics
D_H	hydraulic diameter (m)
g	gravity acceleration
H	head (m)
k	turbulent kinetic energy (m^2/s^2)
MRF	multiple reference frames
OL	over load performance
P	pressure (Pa)
P_d	discharge pressure (Pa)
P_s	suction pressure (Pa)
PL	part load performance
Q	flow rate (m^3/h)
r	position vector (m)
RNG	re-normalization group
S	source term
SDUSs	Skew Upwind Differencing Schemes
SST	shear stress transport model
t	time (s)
u	relative velocity of fluid (m/s)
V	capacity (l)
Z	altitude (m)

Greek symbols

β_2	outlet angle of blade(degree)
ρ	density of the fluid (kg/m^3)
μ	viscosity (Pa s)
μ_t	eddy viscosity (Pa s)
υ	kinematic viscosity (mm^2/s)
Ω	rotational speed (rpm)
τ	stress tensor
η	efficiency (%)

Subscripts and superscripts

Avg	average
d	discharge
elect	electrical
i, j	components
in	inlet
out	outlet
s	suction
t	total
—	time-averaged value

increase of the impeller slip factor which helps conduction of the flow.

Shojaeefard et al. [13,14] performed experimental and numerical investigations to obtain the effect of the impeller's outlet angle during the pumping of oil. As a result of these researches, when the blade outlet angle increases, the width of wake at the outlet of impeller decreases, this phenomenon causes the improvement of centrifugal pump performance when handling viscous fluids.

In 2008, Grepsas et al. [15] conducted the numerical study and optimized design of blades in centrifugal pumps by means of the evolutionary strategy (dynamic algorithm). In this research, commercial software for the analysis of flow was used to conduct parametric studies of the effect of some geometric parameters of a centrifugal pump impeller, and the results revealed that their modifications could have a significant impact on its performance.

In 2009, Anagnostopoulos [16] presented a quick numerical approach for the analysis of flow and the design of impeller blades. In this research, a numerical method for the two-dimensional and turbulent flows in the impeller of centrifugal pumps was developed. Spence and Amaral-Teixeira [17] studied the geometrical variations on the pressure pulsations and performance characteristics of a centrifugal pump by CFD method. The results of this paper presented by concentrating on the selected locations around the pump and provided the detailed information regarding the pressure pulsation close to the impeller outlet, in the volute and in the leakage flow region. There are many other valuable references covering the ongoing research and review of the computational fluid dynamics to study the flow in the impeller of a centrifugal pump [18–21].

In this article, the procedure of 3-D investigation of flow in centrifugal pumps includes the sections of geometry definition, mesh generation, analysis of equations, and processing of results. The finite volume method is used for the discretization of the governing equations of flow. The turbulence model used in the simulation is the " $k-\omega$ SST" model. At the steady state case, this simulation has been analyzed by using the multi-reference frame technique, in which the impeller is situated in the rotating reference frame and the volute is in the fixed reference frame. The obtained numerical results are compared with the experimental ones, and acceptable correlation is found between the two sets of results.

Numerically solving the complete 3-D geometry of the centrifugal pump, offering an impeller for improving the efficiency, simulating and comparing various possible geometries, and plotting of Static pressure contours and velocity vectors inside the pump are some of the significant features of this article.

2. Geometry of the centrifugal pump and the generation of mesh

In this report, for the numerical investigation of the flow field of centrifugal pump, the geometry of the pump model: 65–200 (made by the Pump Iran Co.) is used. This centrifugal pump has single axial suction and vane less volute casing; equipped with an impeller of 209 mm in outside diameter and six backwards curved blades. The blade outlet and wrapping angles of the impeller are 27.5° and 140° respectively. The shroud of the impeller made of metal is machined. The roughness of the impeller and volute is 100 μm. The pump tested is driven by a three-phase AC electric motor, whose rated power is 5.5 kW and speed is 1450 rpm. First, the initial geometry of this centrifugal pump (which has been designed for the pumping of water) is simulated by using the available technical specifications. In the next steps, in order to analyze the effect of changing the fluid viscosity, the outlet angle and the flow passage width are modified. In this analysis, the definition of geometry covers the three pump sections of volute, impeller, and outlet pipe, which are connected together for the analysis of the whole pump (Fig. 1).

Then, in the mesh generation part of this code, mesh configuration is produced based on the type of physics that is considered for the problem. For better conformity of the geometry with the computational domain, at the near-wall regions, the structured mesh is used for the boundary layer, and at regions away from the wall, the unstructured mesh configuration is employed to correctly cover the complex geometry. For producing the unstructured mesh configuration, six-sided, pyramid, and wedge-shaped elements are used in appropriate situations, which are shown in Fig. 1.