VOL. 12, NO. 8, APRIL 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

FLOW PATTERN STUDY OF A CENTRIFUGAL PUMP USING CFD METHODS CONCENTRATING ON VOLUTE TONGUE ROLE

N. Pourmahmoud and S. Majid Taleby
Faculty of Engineering, Urmia University, Urmia, Iran
E-Mail: majid.taleby@gmail.com

ABSTRACT

In this paper, a 3-D simulation of complex flows in a centrifugal pump (EN 80-400, Pumpiran) was performed utilizing computational fluid dynamics methods. The standard k -ε model with standard wall functions and SIMPLE algorithm were chosen for turbulence model and pressure-velocity coupling respectively. The moving reference frame was used to calculate the interaction between impeller-volute in steady condition. Also grid independency study was performed. Flow field inside impeller in the static pressure contour, path lines and velocity vector plot were shown. The head coefficients and radial force at different flow rates were predicted and they agree well with the experimental data of this pump. In all simulation results the effect of volute tongue on the flow field was described. Finally the interpretations of results indicated that for efficiency enhancement, volute requires to redesign.

Keywords: computational fluid dynamics, centrifugal pump, MRF method, CFD, volute tongue.

INTRODUCTION

Centrifugal pumps are used in a widespread range of applications to enhance energy content of a liquid flowing through them. They transforms energy of an initial mover (an electric motor or turbine) first into velocity or kinetic energy, so into pressure energy of a pumped liquid. Centrifugal pumps have two main parts: impeller and volute or diffuser. The impeller is the rotating part that transforms driver energy into the kinetic energy. The volute or diffuser is the stationary part that converts the kinetic energy of the liquid into pressure energy.

Due to the extensive complication of flow in the centrifugal pumps, because of the three dimensional developed structures, turbulence, secondary flows, unsteadiness, etc., there exist still many unknown issues relevant to the complete flow field in these pumps, which require to be studied. Also the conduction of experimental investigations on models with various volute and impeller geometries is time-consuming and costly, and due to the complicated geometry, it is not feasible to accomplish a thorough study of the flow pattern for a vast number of operating conditions. Therefore, Computational fluid dynamics (CFD) has recently become a suitable method of study of the flow patterns and losses.

Computational fluid dynamics (CFD) have successfully accomplished the prediction of the flow through the pumps and the improvement of their design. Various researchers have considerably contributed to evaluating the flow field inside centrifugal pumps with considering the interaction between impeller and volute or vane diffuser, in order to reach the design of high performance centrifugal turbo machines. For instance, Shojaeefard *et al.* [1, 2], Asuaje *et al.* [3], Majidi [4], Huang *et al.* [5] and Zang *et al.* [6] investigated flow field inside centrifugal pump and performed parametric study utilizing computational fluid dynamics methods. Nevertheless necessity to the serious works that result in correction of the existence design or new design of centrifugal pumps is severely sensible.

In this article, the procedure of 3-D investigation of flow in centrifugal pumps includes the sections of geometry definition, mesh generation, mesh independency study, boundary condition, solver formulation, and processing of results. The obtained numerical results are compared with the experimental ones, and acceptable correlation is found between the two sets of results. The flow field inside pump was investigated and concentrated on the volute tongue role in the performance of pump.

MODEL DEVELOPMENT

In this study, a commercial centrifugal pump (EN 100-400 manufactured by PUMPIRAN) was investigated. The main pump parameters are presented in Table-1.

Table-1. Geometrical parameters of the pump.

Pump EN 80-400 (PUMPIRAN)	
Parameter	**Value**
Impeller	
Inlet flange diameter	100 mm
Impeller inlet diameter	122 mm
Impeller outlet diameter	404 mm
Impeller width	12 mm
Outlet blade angle	28°
Blade number	7
Volute	
Base volute Diameter	420 mm
Volute width	21mm
Outlet flange diameter	60 mm
Operational Condition	
Angular Velocity	1450 rpm
Flow rate at BEP	92.5 m^3/hr.
Head at BEP	51 m
Specific Speed	12.2