International Journal of Advance Research in Science and Engineering
Volume No.06, Special Issue No.(02), December 2017
www.ijarse.com
IJARSE
ISSN: 2319-8354

# NUMERICAL SIMULATION OF FLOW FIELD IN A CENTRIFUGAL PUMP

**Prakash Kumar[1*], P.K Chandra[1] , Yogesh Kumar Yadav[1]**

[1]*Skyline Institute of Engineering & Technology, Greater Noida*

**ABSTRACT**

*In this paper a general three-dimensional simulation of turbulent fluid flow is presented to predict velocity and pressure fields for a centrifugal pump. Computational fluid dynamics (CFD) is being increasingly applied in the design of the centrifugal pumps. 3-D numerical computational fluid dynamics tool can be used for simulation of the flow field characteristics inside the turbo machinery. Numerical simulation makes it possible to visualize the flow condition inside a centrifugal pump and provides the valuable hydraulic design information of the centrifugal pumps. Present work is aimed to analyse the pressure and velocity distribution inside the pump passage and evaluate the pump performance using the Fluent, a computational fluid dynamics simulation tool. A numerical model of an impeller and casing has been generated and the complex internal pressure and velocity distribution are investigated by using the fluent computational code.*

*Simulation results in the form of characteristic curves, using these characteristics curves the performance of pump can be predicted that operate under design and off-design conditions at different rotational speeds.*

***Keywords: Centrifugal pump, Turbulence modeling, Computational fluid dynamics (CFD), Numerical Simulation***

## I INTRODUCTION

Centrifugal pumps are prevalent for many different applications in the industrial and other sectors. Nevertheless, their design and performance prediction process is still a difficult task, mainly due to the great number of free geometric parameters involved. On the other hand 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. For this reason, CFD analysis is currently being used in hydrodynamic design for many different pump types [1–3].

Numerical simulations can provide quite accurate information on the fluid behaviour in the machine, and thus help the engineer to obtain a thorough performance evaluation of a particular design. However, the challenge of improving the hydraulic efficiency requires an inverse design process, in which a significant number of alternative designs must be evaluated. Despite the great progress in recent years, even CFD analysis remains rather expensive for the industry, and the need for faster mesh generators and solvers is imperative [4]. Some of the recent investigations in this field are mentioned in the following.

Guleren and Pinarbasi [5] analysed a centrifugal pump by solving Navier–Stokes equations, coupled with the standard k–e turbulence model. Their pump consisted of an impeller having five blades and a low rotating speed of 890 rpm. Numerical simulations were performed on a commercial FLUENT package assuming steady flow. Asuaje et al. [6] performed a 3D-CFD simulation of impeller and volute of a centrifugal pump using CFX code with a specific speed of 32. In this simulation, structured grid was used in the impeller and unstructured grid in the volute. k–e, k–x and SST turbulent models were used. They found velocity and pressure fields for different flow rates and radial thrust on the pump shaft. Cui et al. [7] investigated the effect of number of splitting blades for long, mid and short blades using a one-equation turbulent model. Their computations were performed using commercial FINE/TURBO 6.2 at a specific speed of 18. Their results show that the bulk flow in the impeller has an important influence on the pump performance. Anagnostopoulos [8] simulated 3D turbulent flow in a radial pump impeller for a constant rotational speed of 1500 rpm based on the solution of the RANS equations. The flow equations were discretized using the control volume approach, and the standard k–e model was adopted for the turbulence closure. The computations for the steady flow field in a particular impeller were presented. The characteristic performance curves for the entire load range of the impeller were constructed, and their pattern was found reasonable and in agreement with theory. None of the previous works includes study of 3D modelling within a full domain considering interaction between rotor and stator of a high centrifugal pump using various turbulence models.

In the current study the effect of various turbulence models (k–e, RNG and RSM) on the flow field and efficiency of a high speed centrifugal pump has been carried out. Using the most suitable model the effect of blade number on the specific characteristic of the pump has been investigated.

## II CFD METHODOLOGY

In order to obtain better design in CFD, following methodology is used for designing of any components.

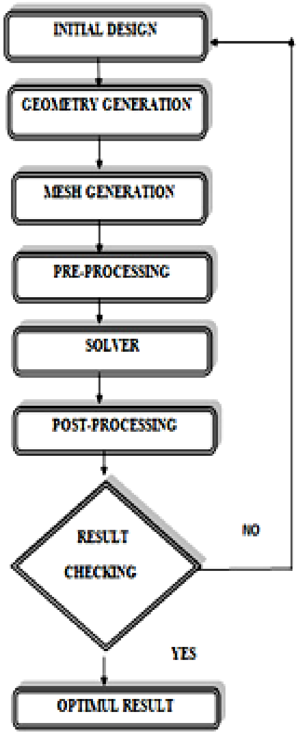


**Fig. 2.1 Flow Chart Of CFD**

## III GOVERNING EQUATION

The physical aspects of any fluid flow are governed by the following two fundamental Principles:

a) Conservation of mass

b) Conservation of momentum

Since the fluid surrounding the impeller rotates around the axis of the pump the equations must be organized in two reference frames, stationary and rotating reference frames. To accomplish this, the Multiple Reference Frame (MRF) model has been used. In this approach, the governing equations are set in a rotating reference frame, and coriolis and centrifugal forces are added as source terms. Continuity and momentum equations of an incompressible flow are as the following:

$$\nabla.u = 0 \qquad \text{…………..} \; 3.1$$

$$\nabla.(\rho uu) = -\nabla_p + \nabla_\tau + s \qquad \text{………………} 3.2$$

In the above equations u is the relative velocity of fluid, τ the stress tensor and s is the source term, which consists of Coriolis and centrifugal forces.

$S = 2\rho N\times u - \rho N\times(N\times r)$ ……………3.3

Here N is rotational speed and r position vector

### 3.1. Boundary Conditions

The following boundary conditions at the walls are used with the equations of motion.

- No slip conditions

At fluid wall interface, there must be no slip.

$V_{fluid} = V_{wall}$

- Inlet condition
- Outlet condition,
- Pressure and velocity
- Rotation about the axis direction
- Amount of mass flow at inlet and outlet

## IV MODELING OF THE PUMP COMPONENTS

To study the numerical analysis on the pump, the dimension data of the pump was required to generate a model in the software. A sample data of centrifugal pump is taken which is shown in table 4.1.

**Specification parameters of the original pump**

| Description | Parameter | Value |
|---|---|---|
| Design flow rate (m3/h) | $Q_d$ | 40 |
| Head (m) | $H$ | 30 |
| Specific speed (rpm) | $N_s$ | 23.4 |
| Impeller diameter (mm) | $D_i$ | 168 |
| Impeller blades number | $Z_i$ | 8 |
| Impeller blade outlet width (mm) | $b_2$ | 11 |
| Impeller inlet diameter (mm) | $D_1$ | 70 |
| Impeller blade warp angle (∘) | $\varphi$ | 120 |
| Impeller outlet blade angle (∘) | $\beta_2$ | 12 |

International Journal of Advance Research in Science and Engineering
Volume No.06, Special Issue No.(02), December 2017
www.ijarse.com
IJARSE
ISSN: 2319-8354

Table 4.1

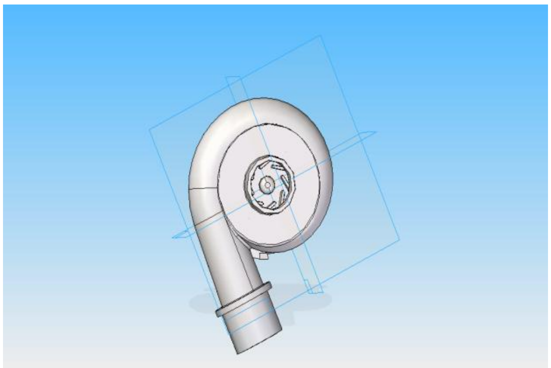

**Fig. 4.1 Volute casing with impeller**

## 4.1 Meshing of Pump Assembly

ANSYS Turbo grid is used for the meshing of the domains. During the pre-process stage the grid generation constitutes one of the most important steps after creation of the geometry. It is required to subdivide the flow domain into smaller sub-domains which are not overlapping each other. The flow physics are solved within the domain geometry that has been created. The result is the generation of the mesh which is made from small cells.

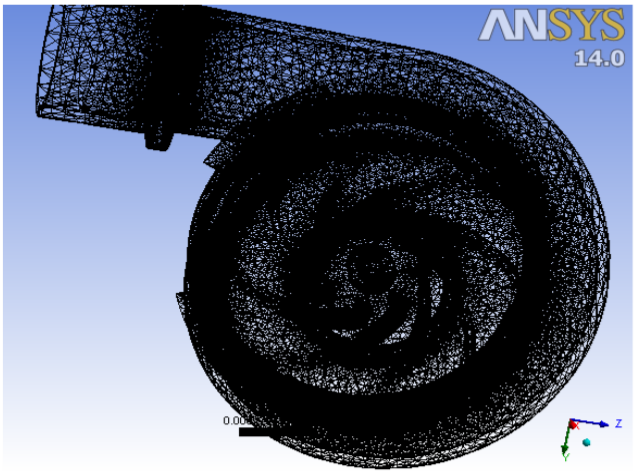


**Fig. 4.2 coarse mesh with grid interval size 4**

**Statistics of mesh**

| | |
|---|---|
| Number of nodes | 479304 |
| Number of element | 2514682 |

## V COMPUTATIONAL EVALUATION OF PUMP PERFORMANCE BY USING FLUENT TOOL OF ANSYS

### 5.1 Simulation of Pump

After meshing of the model of pump assembly commercial CFD code FLUENT is used for simulation of the pump performance .The boundary conditions of mass flow rate given at pump inlet. The performance results are obtained at different mass flow rate conditions with constant operating speed by taking different turbulent modelling. Numerical performance results are compared with the experimental results at the same operating conditions, i.e at constant rotational speed.

#### 5.1.1 Assumptions

The following assumptions are taken for simulation:

- The walls of the casing are assumed to be smooth hence any disturbances in flow due to roughness of the surface is neglected.
- The friction co-efficient for all surfaces is set to 0, hence friction between the walls and fluid is neglected.
- Steady state conditions and incompressible fluid flow.

#### 5.1.2 Solution Parameters

- 3-D double precision solver is used to solve for simulation.
- Multiple reference frame technique is used to simulate the pump performance.
- Clear water used is taken as working fluid.
- Standard K-Epsilon, K-omega, Eddy large simulation model are used for turbulence modeling.
- Convergence criteria for continuity, velocity and turbulence parameters is set to
- Second order scheme is used for pressure correction as well as for solving momentum, turbulent kinetic energy and turbulence dissipation rate.

### 5.2 Performance Characteristic of Pump

The performance characteristic of the centrifugal pump has been predicted numerically handling clear water. Head vs. discharge characteristic of the pump are predicted by CFD analysis at 1200 rpm and 1000 rpm and explained graphically in Figures 5.1 and 5.2 The Numerical results on pump performance handling clear water using three different turbulent modelling namely as K-epsilon ,K-Omega and Eddy Large Simulation at the different rated speed of 1200 and 1000 rpm are given in Table 5.1 and 5.2

International Journal of Advance Research in Science and Engineering
Volume No.06, Special Issue No.(02), December 2017
www.ijarse.com
IJARSE
ISSN: 2319-8354

**Table 5.1 Result of pump performance by experimental and simulation at 1200 rpm.**

| Sr No | Discharge Q (lps) | Head (m) experimental | Simulation (k-e) | Simulation (k-w) | Simulation (ELS) |
|---|---|---|---|---|---|
| 1 | 8.71 | 7.93 | 8.7563 | 8.8563 | 8.6523 |
| 2 | 7.57 | 8.50 | 9.4568 | 9.6023 | 9.3023 |
| 3 | 6.69 | 9.04 | 9.9236 | 10.0230 | 9.8232 |
| 4 | 5.32 | 9.70 | 10.5698 | 10.6850 | 10.4532 |
| 5 | 4.50 | 10.14 | 10.9236 | 11.0200 | 10.8236 |
| 6 | 3.44 | 10.50 | 11.3698 | 11.4632 | 11.2630 |
| 7 | 2.29 | 10.82 | 11.8089 | 11.9200 | 11.7123 |
| 8 | 1.18 | 11.06 | 11.8793 | 11.9765 | 11.7765 |
| 9 | 0 | 11.51 | 12.1652 | 12.2360 | 12.0236 |

**Table 5.2 Result of pump performance by experimental and simulation at 1000 rpm**

| Sr. No | Discharge Q (lps) | Head (m) Experimental | Simulation (k-e) | Simulation (k-w) | Simulation (ELS) |
|---|---|---|---|---|---|
| 1 | 7.051 | 5.4488 | 6.3969 | 6.5000 | 6.2360 |
| 2 | 6.135 | 5.8573 | 6.8070 | 6.9070 | 6.7254 |
| 3 | 6.135 | 6.1685 | 7.1625 | 7.2633 | 7.0234 |
| 4 | 5.552 | 6.5216 | 7.5236 | 7.6236 | 7.3596 |
| 5 | 3.573 | 6.9593 | 7.9360 | 8.0360 | 7.8234 |
| 6 | 2.664 | 7.2781 | 8.2978 | 8.3960 | 8.0236 |
| 7 | 1.658 | 7.4950 | 8.5023 | 8.6123 | 8.3256 |
| 8 | 0.95 | 7.5800 | 8.5600 | 8.6632 | 8.4365 |
| 9 | 0 | 7.6612 | 8.6600 | 8.7600 | 8.5200 |

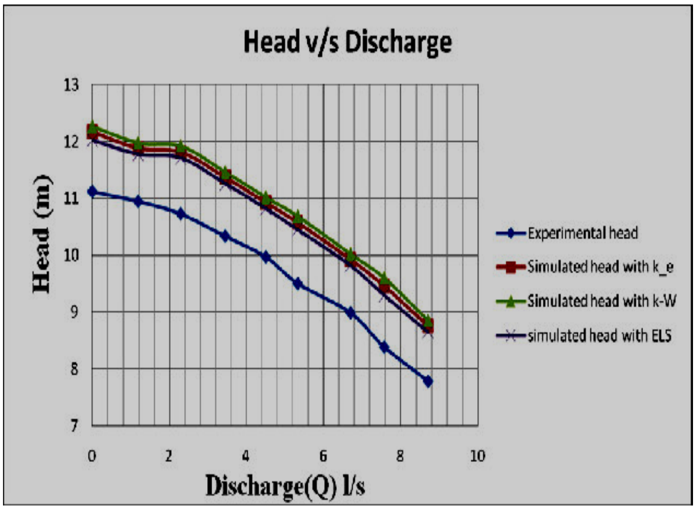


**Fig. 5.1 Head- Discharge comparisons of pump at 1200 with different Turbulentmodelling**

International Journal of Advance Research in Science and Engineering
Volume No.06, Special Issue No.(02), December 2017
www.ijarse.com
IJARSE
ISSN: 2319-8354

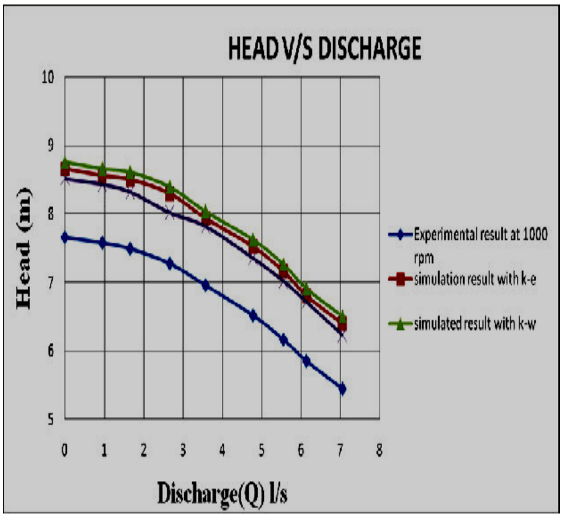


**Fig. 5.2 Head- Discharge comparisons of pump at 1000 rpm withmodelling different Turbulent**

### 5.3 Discussion for Curves Obtained

The computed head Vs discharge curves are compared with the experimental data at different rotational speeds. The simulation results are obtained at the operating speed 1200 rpm and 1000 rpm with different mass flow rates for transportation of clear liquid. The Simulation is performed by using three types of turbulent modelling such as k-Epsilon, K-ω, Eddy large simulation. The performance results show that total static head is the function of the mass flow rate with constant operating speed. A good agreement between numerical and experimental result is achieved over the entire flow range. However, it is also found that the present numerical results show a little higher prediction than the experimental data. This is because the flow losses in the pipes and volute are neglected in the current calculations.

## VI CONCLUSION

This paper discussed with the numerical simulation of centrifugal pump using FLUENT tool of Ansys software. From the present work, it can be concluded that: (1) Computational fluid dynamics is an important and useful tool for pump designers.Traditionally, centrifugal pump design depends on too many factors. The designer can not accurately predict the pump performance before it is tested. The pattern cost, manufacturing of prototype and testing are quite expensive and it may take many trials to obtain satisfactory or desirable results. With the help of Computational Fluid Dynamics (CFD) thedesign of centrifugal pumps are easier and cheaper. (2)The computed head-flow H-Q curves are compared with the experimental data at various rotational speeds. A good agreement between numerical and experimental result is achieved over the entire flow range. However, it is also found that the present numerical results show a little higher prediction than the experimental data. This is because the flow losses in the pipes and volute are neglected in the current calculations. It is also found that the best result appears to be obtained by K-ω model.

**REFERENCES**

[1] C. Hornsby, CFD – driving pump design forward, World Pumps 2002 (2002) 18–22.

[2] S. Cao, G. Peng, Z. Yu, Hydrodynamic design of rotodynamic pump impeller for multiphase pumping by combined approach of inverse design and CFD analysis, ASME Trans. J. Fluids Eng. 127 (2005) 330–338.

[3] F.A. Muggli, P. Holbein, CFD calculation of a mixed flow pump characteristic from shutoff to maximum flow, ASME Trans. J. Fluids Eng. 124 (2002) 798–802.

[4] M. Asuaje, F. Bakir, S. Kouidri, R. Rey, Inverse design method for centrifugal impellers and comparison with numerical simulation tools, Int. J. Comput.FluidDyn. 18 (2) (2004) 101–110.

[5] K.M. Guleren, A. Pinarbasi, Numerical simulation of the stalled flow within a vaned centrifugal pump, J. Mech. Eng. Sci. 218 (2004) 425–435.

[6] M. Asuaje, F. Bakir, S. Kouidri, F. Kenyery, R. Rey, Numericalmodelization of the flow in centrifugal pump: volute influence in velocity and pressure fields, Int. J. Rotat. Mach. 3 (2005) 244–255.

[7] Boaling Cui, Zuchao Zhu, Jianci Zhang, Ying Chen, The flow simulation and experimental study of low-specific-speed high-speed complex centrifugal impellers, Chin. J. Chem. Eng. 14 (4) (2006) 435–441.

[8] J.S. Anagnostopoulos, Numerical calculation of the flow in a centrifugal pump impeller using Cartesian grid, in: Proceedings of the Second WSEAS International Conference on Applied and Theoretical Mechanics, Venice, Italy, November 2006.

[9] John S. Anagnostopoulos,(2008) A fast numerical method for flow analysis and blade design in centrifugal pump impeller, *Journal of computer and fluid*, pp 284-289.

[10] Yang, C., Min-Guan, K., Dong, C., Liu, D. and Dong, X., (2007), Analysis of Turbulent Flow *Engineering Scienc eand Technology*, Volume: 3, pp 245-258.in the Impeller of a Chemical Pump,