CFD Analysis of Centrifugal Pump in Sewerage System

J. Beston[1], G. Gopi[1], S. Gopi[1], M. Karthika[1], Dr. S. V. Suresh Babu[2]

[1]*Department of Mechanical Engineering, Adhiyamaan College of Engineering, Anna University, Hosur, India*
[2]*Professor, Adhiyamaan College of Engineering, Anna University, Hosur, India*

***Abstract*: This work aims at studying the influence of adding splitter blades on the performance of centrifugal pump for sewerage application. Centrifugal pump design is well facilitated by the CFD. Pump improvement performance can be achieved by making geometrical changes in design of an impeller like blade inlet and outlet angles, impeller inlet and outlet diameter, no of blades. Increase in no of blades for sewerage application leads to clogging inside the impeller due to reduced area and increased friction inside the casing. Addition of splitter blades decrease the blockage at impeller inlet, resulting in increase in pump performance.The crucial role played by the blade thickness blockage on the incidence flow angle at the leading edge of full blades was also investigated. Adding splitters has a positive effect on pressure fluctuations is also presented in the paper.**

***Keywords*: Splitter blades, Centrifugal pump, Pump performance, Impeller, Clogging.**

I. INTRODUCTION

A centrifugal pump is a kinetic device. Liquid entering the pump receives kinetic energy from the rotating impeller. The centrifugal action of the impeller accelerates the liquid to a high velocity, transferring mechanical (rotational) energy to the liquid. That kinetic energy is available to the fluid to accomplish work. In most cases, the work consists of the liquid moving at some velocity through a system by overcoming resistance to flow due to friction from pipes, and physical restrictions from valves, heat exchangers and other in-line devices, as well as elevation changes between the liquid's starting location and final destination. When velocity is reduced due to resistance encountered in the system, pressure increases. As resistance is encountered, the liquid expands some of its energy in the form of heat, noise, and vibration in overcoming that resistance. The result is that the available energy in the liquid decreases as the distance from the pump increases. The actual energy available for work at any point in a system is a combination of the available velocity and pressure energy at that point.

Centrifugal pumps are a sub-class of dynamic axisymmetric work-absorbing turbo machinery. Centrifugal pumps are used to transport fluids by the conversion of rotational kinetic energy to the hydrodynamic energy of the fluid flow. The rotational energy typically comes from an engine or electric motor. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward into a diffuser or volute chamber (casing), from where it exits. It also used to transport sewage and untreated waste water (e. g. raw waste water).

II. LITERATURE REVIEW

B. Jafarzadeh, et.al. (2011) conducted the flow simulation of a low-specific-speed high-speed centrifugal pump. It was observed that the head coefficient increases with an increase in the number of blades. It was also said that the position of blades with respect to the tongue of volute has great effect on the start of the separation.

Jie Jin. et.al. (2012) carried out design and analysis on Hydraulic Model of the Ultra-low Specific-speed Centrifugal Pump. When ultra-low specific-speed centrifugal pump is on high speed, the speed of the centrifugal impeller inlet flow is very high, the cavitation performance of centrifugal pump impeller will get bad.

Shah, et.al. (2013) had been performed a detail CFD analysis for centrifugal pumps, and described about the importance of CFD technique. And further said that k-ε turbulence model is appropriate to get a reasonable estimation of the general performance of the centrifugal pump, from an engineering point of view, with typical errors below 10 percent compared with experimental data.

Lomakin V.O, et.al., (2017) describes the multi-criteria optimization of the flow of a centrifugal pump on energy and vibroacoustic characteristics and proposed a technique which is based on the use of LP-tau optimization algorithm and it allows to decide on the technical solution to receive optimization criteria dependence from selected parameters, allowing to find an initial approximation closer to the optimal point in future.

Richard B. Medvitz, et.al. (2002) studied about performance analysis of cavitating flow in centrifugal pumps using multiphase CFD. Homogeneous multi-phase CFD method was applied to analyze centrifugal pump flow under developed cavitating conditions. Quasi-three-dimensional analysis was used to model a 7-blade pump impeller across a wide range of flow coefficients and cavitation numbers.

S. Rajendran and K. Purushothaman (2012) performed approach to the analysis of a centrifugal pump impeller using ANSYS-CFX. The performance of the pump is analysed by changing the pressure and blade angle and observed the continuous pressure rise from leading edge to trailing edge of the impeller due to the dynamic head developed by the rotating pump impeller.

Yu Zhang, et.al. (2014) carried out optimization and analysis of centrifugal pump considering Fluid-Structure Interaction. A set of centrifugal pumps with various blade shapes was studied using the FSI method, in order to investigate the transient vibration performance. The transient mechanical behavior of pump impeller has been investigated using the FSI method based on the optimized geometry parameters of the pump impeller.

Khin Cho Thin, et.al. (2008) performed design and performance analysis of centrifugal pump. Additionally, various losses like shock losses, impeller friction losses, volute friction losses, disk friction losses and recirculation losses of centrifugal pump have been said.

Raghavendra S Muttalli, et.al. (2007) describes the CFD simulation of centrifugal pump impeller using ANSYS-CFX. Ethylene Glycol mixture has been used as a working fluid and further concluded that the formation of cavitation on the blade is increasing with the increase of mass flow rate and rotating speed.

Alex George and P Muthu (2016) conducted a CFD analysis of the performance characteristics of centrifugal pump impeller to minimize cavitation. The effects of blade number, inlet and outlet pressures, and characteristics of centrifugal pump were researched by using the methods of numerical simulation.

P. Gurupranesh, et.al.(2014)Enhance the performance of the centrifugal pump through design modification of the impeller and predicted the performance of the pump along with comparative analysis is made for the entire control volume by varying meshing.

S. Kaliappan, et.al. (2016) studied numerical analysis of centrifugal pump impeller for performance improvement. Reduction of axial to radial turning and the associated passage curvatures in the meridional plane along with decreasing the impeller internal losses by the reduction of the secondary flows as well as the size and the location of the wake regions in the impeller passages have been studied.

III. PROBLEM IDENTIFICATION

The efficiency of pumps can be improved by increasing the number of blades, but for the sewerage application increasing the number of blades may lead to clogging in the impeller which gradually decrease the speed of the impeller.

So to increase the efficiency, splitter blades can be used. The analysis of impeller by using splitter is done.

IV. PUMP SPECIFICATION

The systematic research on the influence of the various design aspects of a centrifugal pump in its performance at various flow rates requires numerical predictions and experiments. The specifications of centrifugal pump undertaken in the current analysis are shown in Table No.1.

TABLE 1

SPECIFICATIONS OF PUMP

Blade width b	250 mm
Impeller Inlet diameter D1	550 mm
Impeller Outlet diameter D2	1300 mm
Total head H	26 m
Speed N	360 rpm
Total volume flow rate, Q	0.898 m3/sec
Efficiency	73%
Number of Blades	3

V. SIMULATION OF CENTRIFUGAL PUMP

After meshing of the model of pump assembly commercial CFD code CFX is used for simulation of the pump performance. The boundary conditions are applied. The performance results are obtained by mass flow rateconditions with operating speed by taking turbulent modeling. Figure 1 shows the 3D diagram of centrifugal pump for which the analysis is done.

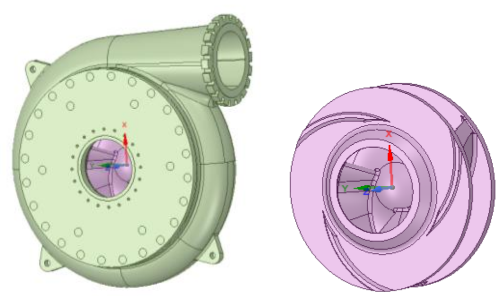


Fig. 1 3D diagram of centrifugal pump

5.1 Assumptions: The simulation of flow inside the centrifugal pump is done on basis of following basic assumptions:

- Steady state condition.
- Constant fluid properties.
- Incompressible fluid flow.
- The walls were assumed to be smooth hence any disturbances in flow due to roughness of the surface wereneglected.

5.2 Boundary conditions: Boundary conditions are the set of conditions specified for the behavior of the solution to a set of differentialequations at the boundary of its domain. Mathematical solutions are determined with the help of boundary conditions to many physical problems. These conditions specify the flow and thermal variables on the boundaries of a physical model.

The pump has various components like inlet, outlet, blades, hub and shroud. The pump inlet was defined as total pressure boundary condition and mass flow rate outlet was given at the pump outlet. The other surfaces were given as wall boundary conditions. Rotating faces of impeller considered as wall and no slip wall condition is applied. At fluid wall interface, there must be no slip.

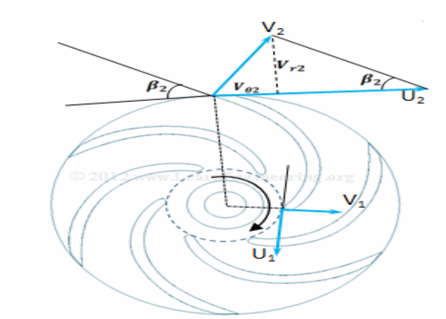
5.3 Solution parameters: Solution parameter is very important in solving any CFD problem. Advection scheme high resolution technique is used to simulate the pump performance. Turbulence numeric is first order. The standard k-ω model is used for turbulence modelling with standard wall function. Convergence criteria for mass,momentum and turbulence parameters were set to10^{-4}. Non- Newtonian fluid is taken as working fluid. Number of iteration used for the simulation of centrifugal pump analysis are1000.

VI. IMPORTANCE OF SPLITTER BLADE

Modelled impellers were having short mid and long blades in addition to original blades which is called splitter blades. It improve the velocity distribution and reduce the back flow in the impeller channel. Back flow in impeller and also flow rate instability of the low-specific-speed centrifugal pump can be reduced by using complex impellers with long, mid and short blades.Splitter blades with 70% length of its main blade length can solve three hydraulic problems of low specific speed centrifugal pumps.

Addition of splitter blades will have some positive effect on the pump cavitation performance It helps to avoid the flow blocking at the impeller inlet and the vortex cavitation inside the blade passages effectively. Pumping head increases as discharge increases. Splitter blades gives smoother pressure and velocity distribution at impeller exit and volute inlet and in turn reduces the pressure fluctuations.

VII. VELOCITY TRIANGLE



- Impeller inlet diameter D_1 =550mm =0.55m
- Impeller outlet diameter D_2=1300mm =1.3m
- Speed =360rpm

Tangential velocity of impeller at inlet $u_1 = \left[\frac{\pi D1N}{60}\right]$

$= \left[\frac{\pi \times 0.55 \times 360}{60}\right]$

$= 10.367 \frac{m}{s}$

Tangential velocity of impeller at outlet $u_2 = \left[\frac{\pi D2N}{60}\right]$

$= \left[\frac{\pi \times 1.3 \times 360}{60}\right]$

$= 24.5 \frac{m}{s}$

Whirling velocity at outlet $Vw_2 = \left[\frac{g\, H_m}{\eta_m u_2}\right]$

$= \left[\frac{9.81 \times 26.06}{0.733 \times 24.5}\right]$

$= 14.23 \frac{m}{s}$

Width of the impeller at outlet B_2 =250mm

=0.250m.

Discharge Q $= 0.898 \frac{m^3}{s}$

Flow velocity at outlet $Vf_2 = \left[\frac{Q}{\pi B_2 D_2}\right]$

$= \left[\frac{0.898}{\pi (1.3)(0.250)}\right]$

$= 0.88 \frac{m}{s}$

From outlet velocity triangle $\tan \phi = \left[\frac{Vf_2}{u_2 - Vw_2}\right]$

$= \left[\frac{0.88}{(24.5 - 14.23)}\right]$

$\tan \phi$ =0.0856

Runner vane angle at outlet ϕ =4.9°

Velocity $Vf_1 = Vf_2$

$Vf_1 = 0.88 \frac{m}{s}$

Runner vane angle at inlet $\tan\theta = \left[\frac{Vf_1}{u_1}\right]$

$= \left[\frac{0.88}{10.367}\right]$

θ =4.85°

Work done by impeller on water/sec $= \frac{W}{g} \times Vw_2 \times u_2$

$$=\frac{\rho \times g \times Q}{g} \times Vw_2 \times u_2$$

$$=\frac{1000 \times 9.81 \times 0.898}{9.81} \times (14.23 \times 24.5)$$

$$=312854.22\frac{Nm}{s}$$

Velocity of water leaving the vane

$$V_2 = \sqrt{{Vf_2}^{\ 2} + {Vw_2}^{\ 2}}$$

$$V_2 = 14.25\ \frac{m}{s}$$

Angle made by the absolute velocity at outlet

$$\tan\beta \quad = \left[\frac{Vf_2}{Vw_2}\right]$$

$$= \left[\frac{0.88}{14.23}\right]$$

$$\beta \quad = 3.54°$$

VIII. RESULTS AND DISCUSSION

Numerical simulations are carried out on the impeller model to predict its performance by giving its working conditions as input. Successive iterations are done by the software to obtain the characteristics such as efficiency, static pressure generated, pressure distribution, direction of flow, turbulence, fluid velocity.

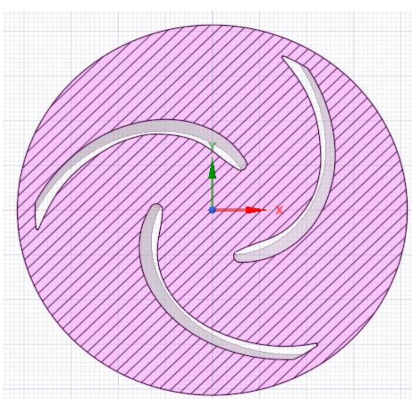


Fig 2 Impeller without splitter blades

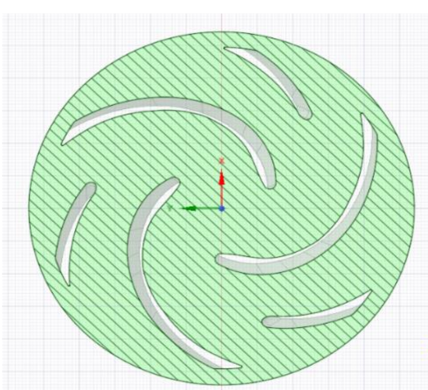


Fig 3 Impeller with splitter blades

8.1 Pressure fields

Figure 3shows the instantaneous static pressure distribution for the original and splittered impellers. The conversion of dynamic pressure produced by the impeller rotation into static pressure by the volute casing can be seen, thus the maximum pressure is obtained in the outlet duct (except at high flow rate and at BEP for the splittered impeller). Whatever the flow rate and the pump are, a nonhomogeneous pressure distribution is observed at the zone around the gap between volute tongue and impeller periphery, characterized by a high gradient of pressure. The volute tongue whose role is to drive the flow towards the fan outlet presents a singularity for the flow.

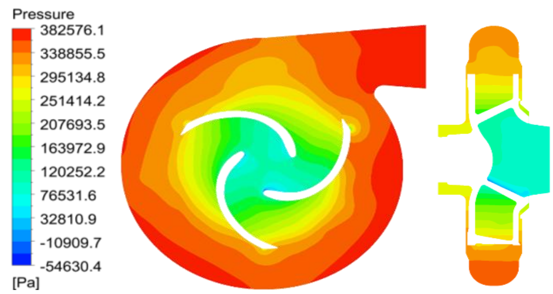


Fig 4 Pressure contours of original pump

8.2 Velocity Fields

The instantaneous velocity vectors in the pump are plotted in Figure 4for the original and splittered impellers. The volute tongue zone presents a strong recirculation of the fluid particles at the gap between the volute tongue and the impeller periphery. The velocity fields show more significant variations when the pump works off the best efficiency point. For the original impeller, a dead volume zone with low velocity magnitude is observed in quarter a periphery from the volute tongue on. The volute tongue is a singularity for the flow that creates a strong recirculation zone at the volute diverging outlet where the fluid particles are slowed down. The impeller with splitter blades drives the fluid better than the original impeller. Velocities are more homogeneous at the impeller periphery for the splittered pump. In fact, what occurs in the blade to- blade space for the original pump is moved to the periphery of the impeller so that all the volute casing space is used.

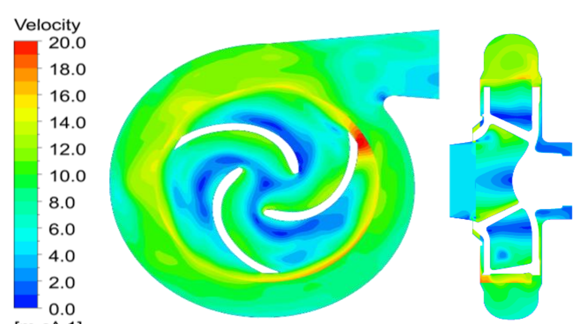


Fig 5 Velocity contours of original pump

The analysis of centrifugal pump by using the 2D streamlines are shown below.

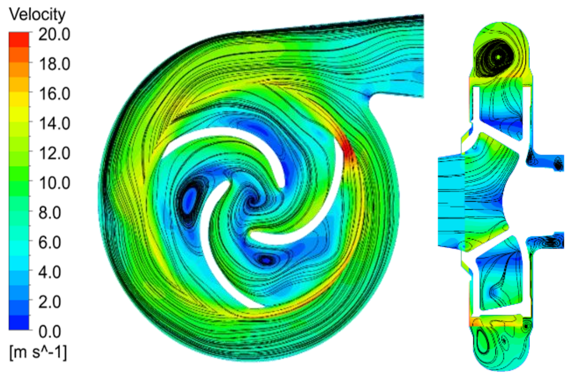


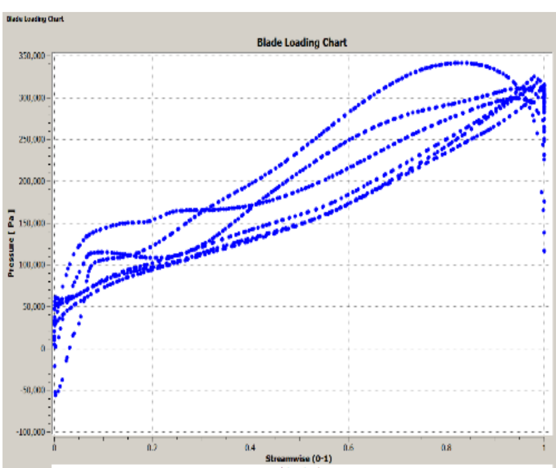
Fig 6 Velocity contours and 2D streamline diagram

The movement of fluid can be seen in Figure 3. The recirculation of the flow can be seen. More recirculation can be seen in the center of the impeller.

8.3 Blade Loads

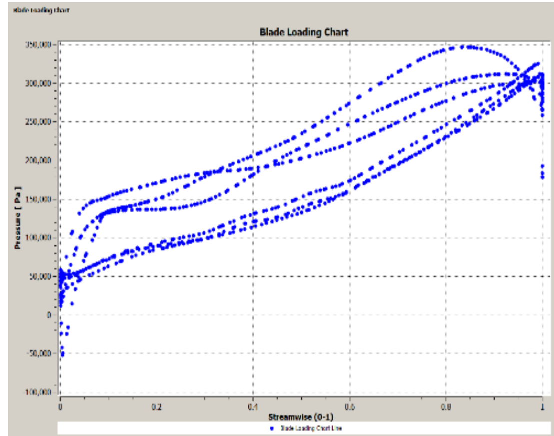
The gross weight of a rotating- wing aircraft divided by the total area of rotor blades. When the plane intersect with the impeller, where the vane touches the plane, pressure will be created. Span length will be always 0 to 1. Span's plane always parallel to hub surface. There are two sides of impeller, suction side and pressure side. Blades are aligned in such a way that suction takes place at the center of the impeller, the water moves away from the impeller without damaging the casing it moves out.

Span 0.2 is the hub, 0.5 is said to be center and 0.8 is the shroud of the impeller. Here X axis is Stream wise and Y axis is Pressure.



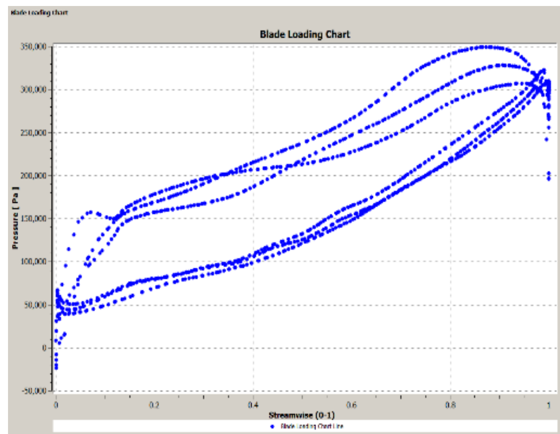
Span 0.2

In hub the pressure side and suction side are interlinked which leads to increase in blade loads and leads to failure in blades.



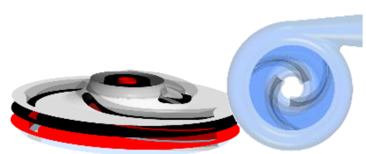
Span 0.5

In centre side, the flow is in correct manner where the pressure is lees in suction side.



Span 0.8

In shroud side, the flow is in correct manner where the pressure is less in suction side and gradually increase in outer side of the impeller.



Hub

Fig 7 Blade loads

8.4 Blade to Blade Velocity Vector

This gives idea about kinetic energy and dynamic pressure acting in different parts. It helps in identifying the direction of fluid particles flowing through the different components. The flow angle must be aligned with impeller angle. If the flow vector is with suction side, then flow is not proper, since the pressure on it increases. Always flow angle must be aligned with impeller angle, if not blade is improperly loaded. Here span 0.5 and span 0.8 have a better flow compared to the other.

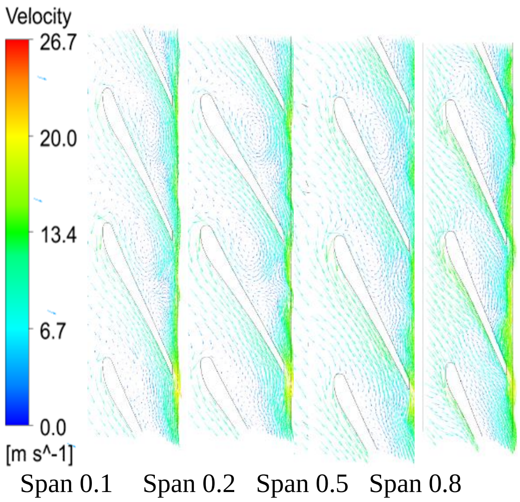


Fig 8 Blade to blade vectors

IX. CONCLUSION

A centrifugal pump impeller is modeled and solved using computational fluid dynamics, the flow patterns through the pump, performance, pressure from hub to shroud line, blade loading, pressure contours at blade leading edge and trailing edge for designing flow rate are presented.

The influence of splitter blades on the velocity and pressure fields in a centrifugal impeller has been analyzed. Adding splitter has negative and positive effects on the pump behavior and cavitation at inlet diameter of splitter blade. Head also increases due to splitter blades addition.

REFERENCES

[1]. Hamsen T.Bubelach and T.Pensler (2008): cavitation in single–vane sewage pumps, Hindawi publishing.
[2]. Jie Jin,Ying Fan Wei Han Jiaxin Hu.(2012): Design and analysis on hydraulic model of the ultra-lowspecific speed centrifugal pump,Elsevier.
[3]. Shah.S.R,Jain.S.V,Patel.R.N,Lakhera.V.J. (2013):CFD for centrifugal pumps:A Review Of the State Of The Art,Elsevier.
[4]. Lomakin.V.O,Chaburko. P.S,Kuleshova M.S.(2016):Multi criteria optimization of the flow of a Centrifugal pump.
[5]. Jafarzadeh.B,Hajari.A,Alishahi.M.(2011): The flow simulation of a low specific speed high speed centrifugal pump,applied mathematical modelling. Elsevier
[6]. Yu Zhang,Sanbao Hu,Yunqing Zhang,Liping Chen.(2014):optimization and analysis of centrifugal pump consideringfluid-structure interaction, Hindawi Publishing Corporation.
[7]. Khin Cho Thin, Mya Khaing, Khin Maung Aye.(2008):Design and performance analysis of centrifugal pump, world academy of science and technology.
[8]. Raghavendra S,Muttalli, Shweta, Agrawal, Harshla Warudkar.(2007):CFD simulation of centrifugal pump impeller using ANSYS-CFX, international journal of innovative research in science, engineering and technology.
[9]. Alex George Muthu P.(2016): CFD analysis of performance characteristics of centrifugal pump impeller to minimizing cavitation, international conference on current research in engineering science and technology.
[10]. Emad H. Imam, Haitham Y. Elnakar.(2014): Design flow factors for sewerage systems in small arid communities, journals of advanced research, Cairo University.