Experimental and CFD Analysis of Regenerative Pump

Rohit S. Kanase[1], Ashok T. Pise[2], Pravin C. Garje[3]
[1, 2, 3] Department of Mechanical Engineering
[1, 2, 3] GCE, Karad

Abstract- *Regenerative pump is rotodynamic turbomachine capable of developing high head at low flow rates. In this paper, an experimental and CFD analysis is carried out in order to investigate the effect of varying flow rate on the performance of pump like head generation, power input and overall efficiency. For this purpose, experiment is carried out by operating pump at five different flow rate. The result showed that head generated by pump and power input decreases with increase in flow rate. As the flow rate increases the overall efficiency increases up to 31 LPM and then it decreases. The maximum efficiency obtained from experimentation is 19.61 at 31 LPM. CFD analysis is used to investigating the complex flow field within pump, visualize the recirculating flow zones and other flow losses .CFD result shows that vortices are formed at outlet region, Straight radial impeller vanes causes flow direction changes abruptly hence pressure losses occurs and from pressure contour observed the water pressure increases continuously as it passes from inlet port to outlet port because water moves helically in the casing chamber and re-enters in the impeller vane passage many times in its peripheral path.*

Keywords- kinetic pump, helical corkscrew flow pattern, high head, low flow rates

Nomenclature

P	Pressure	N/m^2
P	Power	W
η	Overall efficiency	%
H	Head generated by pump	m
V	voltage	V
I	Current	A
ρ	Density	kg/m^3
Q	Volume flow rate	m3/s
g	Gravitation acceleration	m/s^2
$\cos \phi$	Power factor	

Subscripts

1	Suction side
2	Delivery side
i	input
o	Outlet

Abbreviation

RFP Regenerative Flow Pump

I. INTRODUCTION

Pumps are one of the largest user of electricity in industry, agriculture as well as daily water usage applications and out of those pumps, centrifugal pumps usage is nearly 73 %. The RFP is kinetic pump like centrifugal pump; however, it can offer a more effective alternative in many applications. The main characteristic of RFP is to produce high head at low flow rate and also they have self-priming characteristics. The specific speed of regenerative pumps is very low and they share some characteristics of positive displacement pumps without any problems of wear and lubrication, therefore RFP has found many applications in industry and day to day water usage in place of positive displacement machines. They are simple and easy to machine and they have no need of scrolls and diffusers.

The regenerative pump consists an impeller with 30 to 50 radial vanes at each side of its periphery. These vanes rotate in the nearly 330° annular channel. The fluid enters the channel through the inlet port and recirculates through the impeller vanes due to the centrifugal force action on fluid. The name RFP best describes the repeated fluid circulation during the flow process through the vanes. The water pressure increases continuously as it passes from inlet port to outlet port because water moves helically in the casing chamber and re-enters in the impeller vane passage many times in its peripheral path as shown in Fig. 1. Each passage between impeller vanes may be act as a conventional stage of pressurization. These repetitive treatment of pressurization causes regenerative pumps with single impeller have ability to generate a head identical to that of certain centrifugal stages with equal tip speeds. The cavitation in these pumps are very less, because of its smaller pressure gradient, therefore, regenerative pumps require lower NPSH than centrifugal pumps.

CFD tool ANSYS Fluent used to study the effect of geometry modifications like curvature in the outlet flow domain, offsetting impeller blades on either side of impeller, semicircular static fluid on the side of impeller and different number of blade on each side of impeller on regenerative pump performance [1]. CFD software ANSYS Fluent used in order to investigate the effect of inlet and outlet angle of curved blade on the performance of a pump [2]. CFD software ANSYS Fluent used to analyse outcome of geometrical modification on pump performance. These modifications are radial inlet and exit chamber with constant width, splitter vanes used near the outlet flow domain and varying number of impeller blades [3]. Numerical and analytical technique Compares for finding the performance for a new RFP design. The performance characteristics figure out using CFD software ANSYS Fluent and a new one-dimensional model is validated to experimental results [5]. The numerical simulations using commercial CFD software ANSYS Fluent carried out to improve the head of the regenerative pump. Many alternatives were made in the geometry of pump these are providing additional splitter in the outlet passage, increasing number of vanes and inclining the straight radial vanes [6].

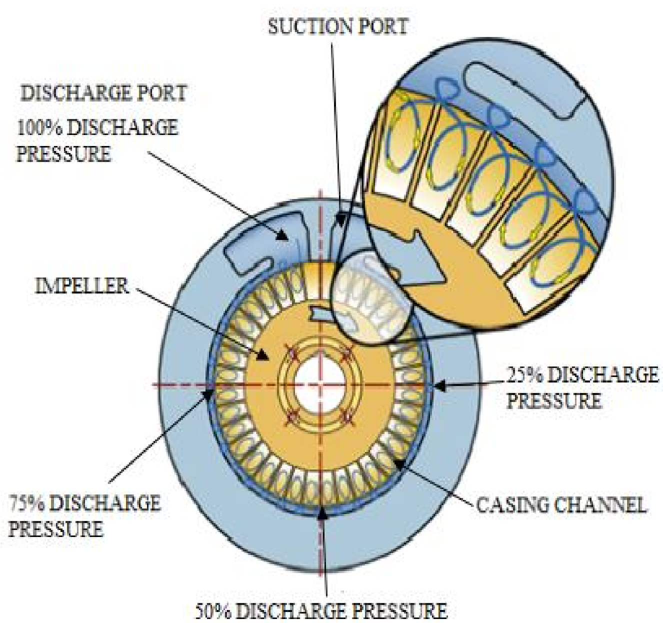

Figure 1. Regenerative pump helical flow path

Reviewed the status of the RFP and proposed a design guideline, with the aim to improve the performance and efficiency of the RFP. All previous work focused on the fully developed flow region in the regenerative pump and this work expanded the attention to the developing region [7]. An improved and modified theoretical model proposed that can explain the change in the circulatory velocity caused by variation in channel area. This work extend to the developing region. Furthermore, in order to make the suggested model, several loss models were assumed and the results of predictions were compared with experimental and CFD data [8]. An experimental study carried out to investigate the effect of straight blades with inclined blade angles and chevron angles on head and efficiency of pump. Detailed numerical investigation of airfoil blading regenerative compressor carried out for optimizing performance [14].

After the extensive literature revealed, it is need to study effect of different geometrical modification suggested by different researchers. And is reviewed in order to study its effect on performance enhancement of regenerative pump. The main purpose of CFD analysis is to visualize the complex flow field in regenerative pump like flow field of liquid at both side of impeller and flow of liquid from inlet port to outlet port.

II. EXPERIMENTAL METHODS

a. Specifics of pump and motor

Present regenerative pump consists of an impeller with 36 radial vanes on each side at its periphery, inlet port, discharge port, stripper to isolate the high-pressure discharge from the low-pressure inlet, flow passage, and a casing. The main geometric parameters of the tested regenerative pump are presented in Figure 2

Table 1. Specification of motor

Motor Details	
KW/Hp	:0.37/0.5
Speed (Rpm)	:2900
Current (Amps)	:3
Freq. (Hz)	:50
Voltage (Volts)	:220

b. Experiments apparatus and procedure.

The schematic view of the test rig arrangement for regenerative pump is shown in Fig 2. A reservoir tank was installed in order to store and receive water that is used as a working fluid. The fluid was drawn to the pump from the tank and the fluid flow was adjusted via a gate type flow control valve in the return line to the tank. The suction and delivery pipe size is 25mm.The fluid flow rate was measured using magnetic flow meter, situated upstream of the flow control valve and upstream of the pump. The pump itself was driven by an induction motor operating at a constant speed 2900 rpm. The suction pressure and the delivery pressure were determined by means of vacuum gauge PG1 and pressure gauge PG2, respectively

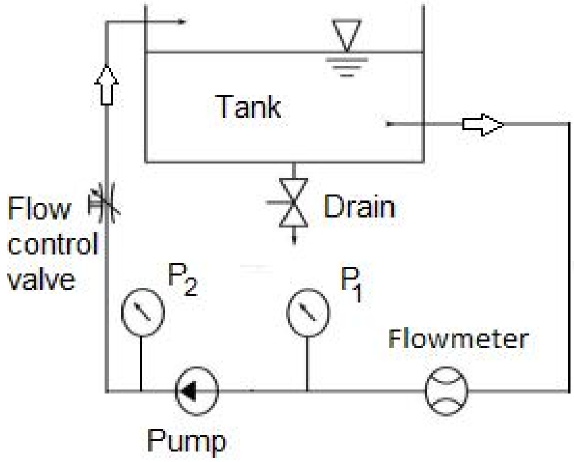

Figure 2. Schematic of regenerative pump test rig

The experimentation is carried out by varying the flow rate and its effect on the other parameter like head, power input and overall efficiency are observed

The uncertainty in measurement of flow rate was about ±5% and the pressures were measured with the uncertainty of ±6%

III. DATA REDUCTION

i. Head generated by pump (H),
   a. $H = (P_2\text{-}P_1)/\rho \times g$

ii. Power input /Motor power (P_i),
   a. $P_i = V\times I\times \cos\varphi$

iii. Water power at outlet (P_O)
   a. $P_O = \rho\times g\times Q\times H$

iv. Overall efficiency (η)
   a. $\eta = (P_O / P_i) \times 100$

III. NUMERICAL SIMULATION AND MODELLING

In this numerical study CFD software ANSYS CFX-17.2 was used to solve the three-dimensional Reynolds Navier-Stokes equation. Governing equations are:

Continuity equation:

$$\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0$$

Momentum equation:

$$\rho\left(\frac{\partial v_x}{\partial T} + v_x\frac{\partial v_x}{\partial x} + v_y\frac{\partial v_x}{\partial y} + v_z\frac{\partial v_x}{\partial z}\right) = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left[(\mu + \mu_t)\frac{\partial v_y}{\partial x}\right] + \frac{\partial}{\partial y}\left[(\mu + \mu_t)\frac{\partial v_T}{\partial y} + \frac{\partial}{\partial z}\left[(\mu + \mu_t)\frac{\partial v_y}{\partial z}\right] + S_{Tx}$$

In this study, multiple reference frame (MRF) modelling approach is used. MRF modelling is also referred as “frozen rotor” technique. The inner region near to the impeller is rotating reference frame and chamber region is treated as a stationary reference frame

a. Geometry

Each part of pump is modeled and assembled in the solidworks 2013 shown in Fig 3. Fluid model is extracted from pump assembly and imported in CFD software ANSYS CFX for further numerical investigation of flow field and pump performance. The fluid model is divided into two parts chamber domain and impeller domain for the purpose of giving rotational boundary condition to impeller. In Ansys design modeler form new part which contains chamber body and impeller body, this help in meshing to generate conformal mesh at the common surfaces (interfaces) of two bodies.

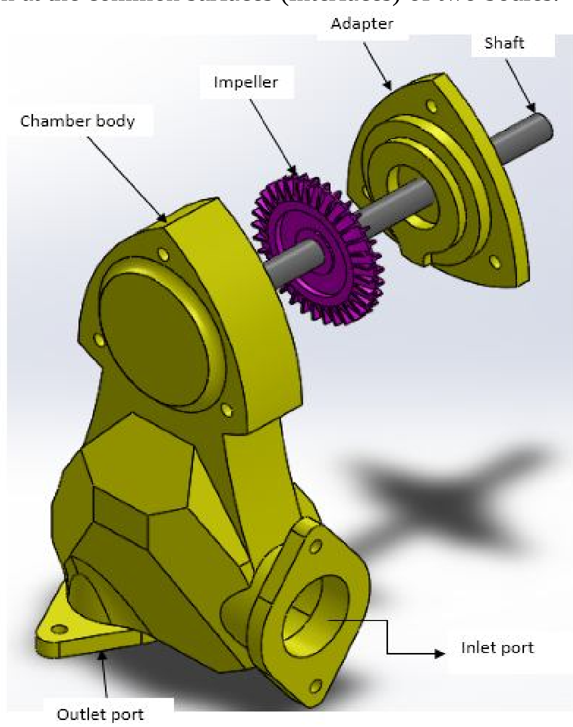

Figure 3. Part arrangement in assembly

b. Grid generation

ANSYS Workbench software is used for grid generation. Both domain are meshed separately by automatic mesh method shown in Fig 4. Advance size functions curvature and proximity used to adapt curved profile and thin gap in geometry during meshing. Local mesh sizing and refinement tool are used to improve quality of mesh. Inflation prism layer are provided at impeller and chamber wall to precisely capture the effect of wall friction on pressure losses and other flow parameter. During mesh generation, orthogonal quality, skewness and aspect ratio are assured to be in desirable range. Skewness was held below 0.92 and aspect ratio applied below 5:1.

Grid independence test is carried out at six different number of nodes 0.9 million, 1.5 million, 1.8 million, 2.3 million, 3.2 million, 3.8 million, from the observation it is found that the variation in head generated by pump was not more than 0.5% for configurations containing element greater than 1.8 million, so grid independent solution obtain at 1.8 million nodes.

c. CFX solver setup

In CFX solver setup chamber domain is define as stationary and impeller domain as rotating at constant speed 2900 rpm. Water at 25o C was selected as working fluid. Reference pressure is kept 0 Pascal. K-ε and SST model are used for analysis, results of both model are nearly same hence further work is continued with SST model. Constant total pressure boundary condition was applied at inlet region in absolute pressure scale and constant mass flow rate boundary condition was applied at outlet. For this quasi-steady simulation, the grid between chamber domain and impeller domain are connected by using frozen rotor interface.

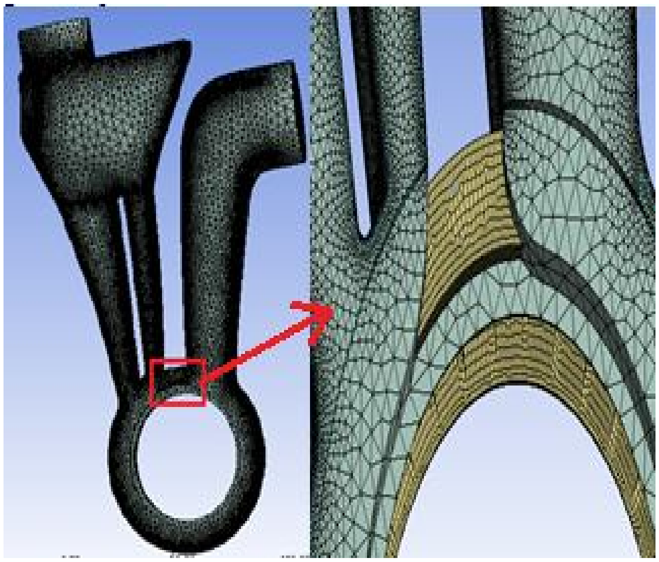

Figure 4. Meshed Pump Model

For this simulation, convergence criteria set to maximum residual of 1×10-5.turbulance intensity was limited to 5%.high resolution Advection scheme and turbulence scheme is applied for discretization of turbulent dissipation rate, momentum equation, pressure, turbulence kinetic energy, to achieve high accuracy. Physical timescale is set to 5.747×10-5seconds, which is time required for 10 rotation of the impeller at rated speed. Expression for head generated by pump, torque on impeller, power input and hydraulic efficiency added and monitored to observe the values on graph.

V. VALIDATION OF THE CFD RESULT

CFD Results are validated by comparing with experimental results. The average percentage deviation of CFD results from experimental results are 15% which is within acceptable limit. Various parametric graph from CFD and experimental results are shown in Fig 5, 8, 9.

VI. RESULTS AND DISCUSSIONS

a. Effect of varying flow rate on the following parameter

i. Head generated by pump

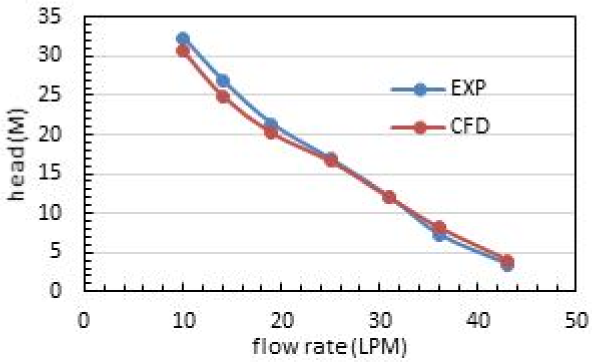

Figure 5. Head vs. flow rate graph

As the discharge increases total head decreases continuously as shown in Fig 5. From CFD results, Fig 6 and Fig 7, it is observed that as flow rate increases the fluid comes in contact with impeller is decreases hence no of stages of pressurization decreases because of that total head generated by pump decreases.

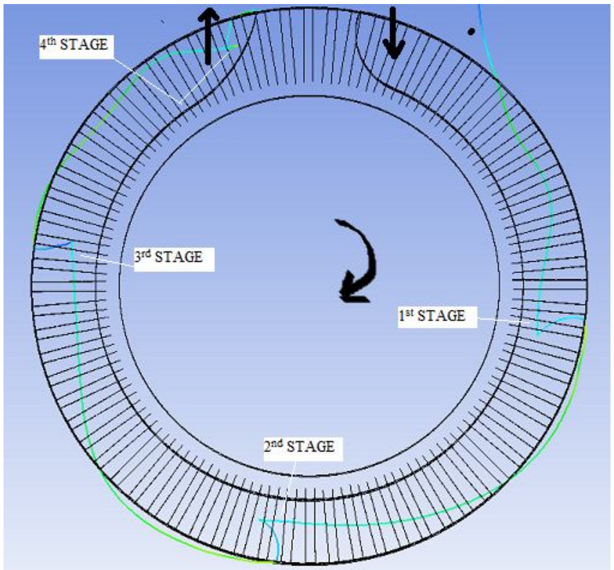

Figure 6. Flow pattern at 31 LPM flow rate from CFD result

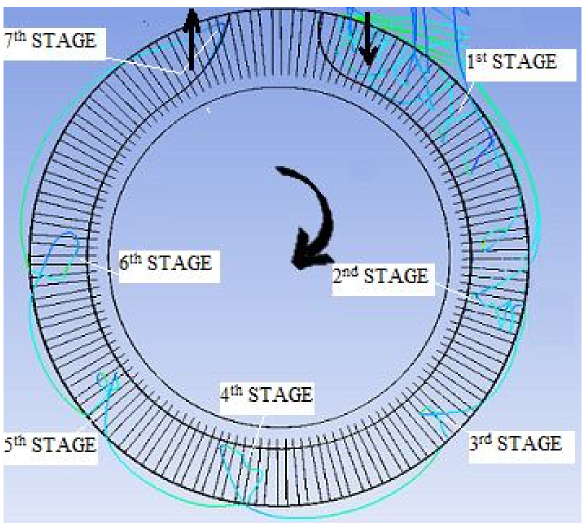

Figure 7. Flow pattern at 10 LPM flow rate from CFD result

ii. Power input to pump (motor power)

As flow rate increases the fluid comes in contact with impeller is decreases hence number of stages of momentum exchange between impeller and water decreases hence power consumption of pump decreases continuously with increase in flow rate as shown in Fig 8. This phenomenon is completely reverse of centrifugal pump that is in centrifugal pump power input increases with increase in flow rate.

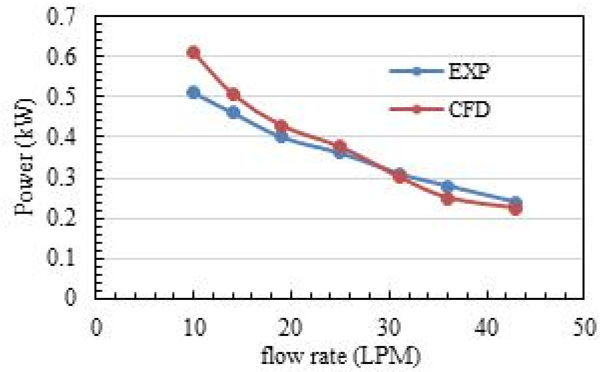

Figure 8. Power input vs. flow rate graph

iii. Overall efficiency of pump

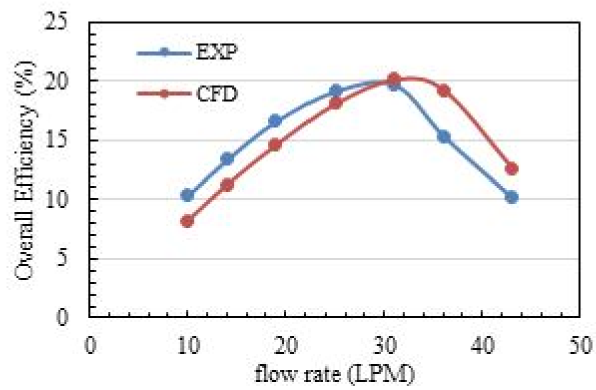

Figure 9. Overall efficiency vs. flow rate graph

As the discharge increases the overall efficiency increases up to 31 LPM and then it decreases as shown in Fig 9.

b. Observations from CFD Analysis

i. Total pressure contour plot

From the pressure contour at the midplane of fluid model shown in Fig 10, it is observed that the water pressure increases continuously as it passes from inlet port to outlet port, because of water come in contact with impeller vanes repeatedly during flow.

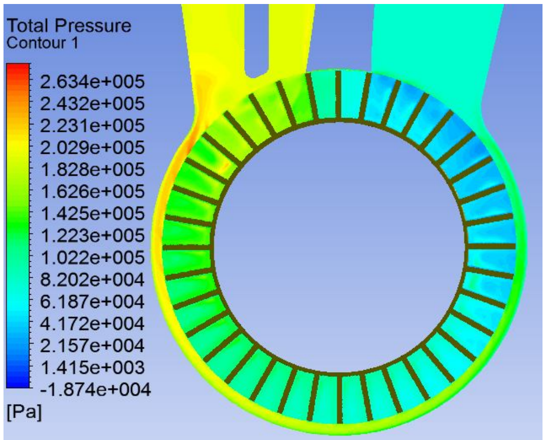

Figure 10. Total pressure contour plot from CFD result.

ii. Velocity vector plot at outlet port

From the velocity vector plot at outlet port of RFP as shown in Fig 11, it is observed that flow vertices are formed on both side of stripper which causes some pressure loss of water. It will be avoided by using curved splitter as mentioned by Vasudeva Karanth et al. [3]

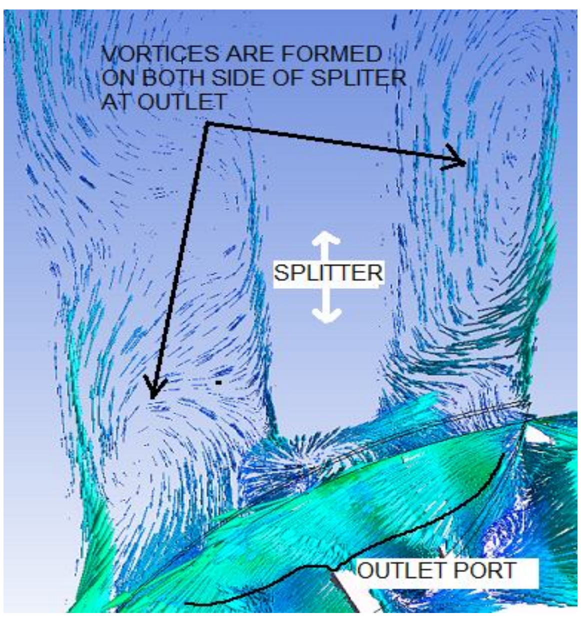

Figure 11. Velocity vector plot at outlet port from CFD result

iii. Velocity vector plot at in-between impeller vane

Impeller with Straight radial vanes causes flow direction changes abruptly hence velocity and pressure losses occurs shown in Fig 12.This losses will be overcome by using curved vane profile as mentioned by . J. Nejadrajabali et al. [5]

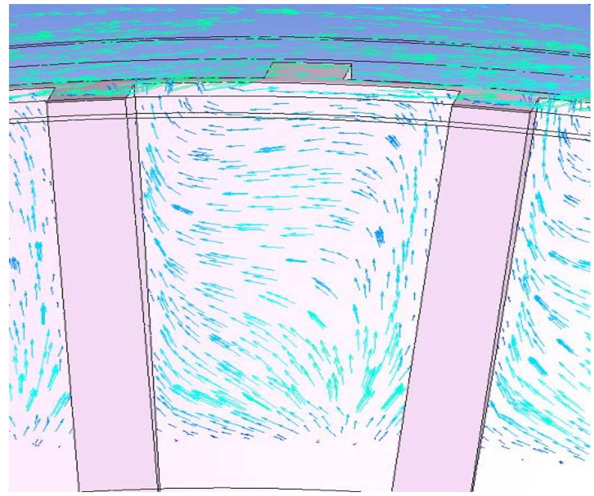

Figure 12. Velocity vector plot at in-between impeller vanes from CFD result

VII. CONCLUSION

Experimental and CFD analysis of regenerative pump is carried out to investigate the effect of varying flow rate on the performance (Head, Power input, Overall efficiency) of pump. Especially CFD analysis is used to investigating the complex flow field within pump, it will helps to reduce the recirculating flow zone and other losses and improve the overall efficiency of pump.

From Experimental and CFD results it is concluded that:

1. As the flow rate increases power input and head generated by pump decreases continuously.
2. As the discharge increases the overall efficiency increases up to 31 LPM and then it decreases, the maximum efficiency obtained from experimentation is 19.61% at 31 LPM.

From CFD results it is concluded that:

1) The water pressure increases continuously because water moves helically in the casing and re-enters the impeller many times in its peripheral path from inlet to outlet.
2) Flow vertices are formed at outlet on both side of stripper which causes some pressure losses.
3) Straight radial vanes causes flow direction changes abruptly hence velocity and pressure losses occurs

VIII. ACKNOWLEDGMENT

Author wishes to thanks GCE, Karad for their lab facility available at advanced CFD laboratory. Also thanks to Crompton Greaves Consumer Electrical Limited, Ahmednagar for giving proper guidance and CFD software facilities.

REFERENCES

[1] Ankita Maity, V. Chandrashekharan, and M. W. Afzal. "Experimental and numerical investigation of regenerative centrifugal pump using CFD for performance enhancement." International Journal of Current Engineering and Technology 5, no. 4 (2015): 2898-2903.

[2] J. Nejadrajabali, A. Riasi, and S. A. Nourbakhsh. "Flow Pattern Analysis and Performance Improvement of Regenerative Flow Pump Using Blade Geometry Modification." International Journal of Rotating Machinery 2016 (2016).

[3] K Vasudeva Karanth, M. S. Manjunath, and Yagnesh N. Sharma. "Numerical study of a self-priming regenerative pump for improved performance using geometric modifications." International Journal of Current Engineering and Technology 5, no. 1 (2015): 105-109.

[4] Quail, Francis, Thomas Scanlon, and Matthew Stickland. "Design optimization of a regenerative pump using numerical and experimental techniques." International Journal of Numerical Methods for Heat & Fluid Flow21, no. 1 (2011): 95-111.

[5] Quail, Francis J., Thomas Scanlon, and Armin Baumgartner. "Design study of a regenerative pump using one-dimensional and three-dimensional numerical techniques." European Journal of Mechanics-B/Fluids 31 (2012): 181-187.

[6] Satish M Rajmane, and S. P. Kallurkar. "CFD Analysis of Domestic Centrifugal Pump for Performance Enhancement." International Research Journal of Engineering and Technology (IRJET) 2, no. 02 (2015).

[7] Engeda, Abraham. "Flow analysis and design suggestions for regenerative flow pumps (Keynote)." In ASME/JSME 2003 4th Joint Fluids Summer Engineering Conference, pp. 1847-1857. American Society of Mechanical Engineers, 2003.

[8] J. W.Song, Abraham Engeda, and M. K. Chung. "A modified theory for the flow mechanism in a regenerative flow pump." Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 217, no. 3 (2003): 311-321.

[9] T. Meakhail, and Seung O. Park. "An improved theory for regenerative pump performance." Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 219, no. 3 (2005): 213-222.

[10] Choi, Won Chul, Il Su Yoo, Mu Ryong Park, and Myung Kyoon Chung. "Experimental study on the effect of blade angle on regenerative pump performance." Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 227, no. 5 (2013): 585-592.

[11] H Iversen, W. "Performance of the periphery pump." Trans. ASME 77, no. 1 (1955): 19-28.

[12] A.Shirinov, and S. Oberbeck. "High vacuum side channel pump working against atmosphere." Vacuum 85, no. 12 (2011): 1174-1177.

[13] H. Altmann, "A regenerative compressor." Journal of Engineering for Industry (1977): 637.

[14] D.Griffini, S. Salvadori, M. Carnevale, A. Cappelletti, L. Ottanelli, and F. Martelli. "On the Development of an Efficient Regenerative Compressor. “Energy Procedia 82 (2015): 252-257.

[15] H. Sixsmith, "The theory and design of a regenerative compressor." Research paper for Institute of Refrigeration, Institute of Marine Engineers (1981): 69-77.

[16] Engeda, Abraham, and Muhammad M. Raheel. "Systematic design approach for radial blade regenerative turbomachines." Journal of propulsion and power21, no. 5 (2005): 884-892.

[17] Badami, Marco. Theoretical and experimental analysis of traditional and new periphery pumps. No. 971074. SAE Technical Paper, 1997.

[18] Quail, Francis J., Thomas Scanlon, and Matthew Strickland. "Development of a regenerative pump impeller using rapid manufacturing techniques." Rapid Prototyping Journal 16, no. 5 (2010): 337-344.