International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 – 5161

Available at http://inpressco.com/category/ijcet

Research Article

Experimental and Numerical Investigation of Regenerative Centrifugal Pump using CFD for Performance Enhancement

Ankita Maity†, Vignesh Chandrashekharan‡ and Muhammad Wasim Afzal†

†Tata Hitachi Construction Machinery Company Private Limited, India
‡Michelin, Canada
†Reliance Industries Limited, India

Accepted 27 Aug 2015, Available online 31 Aug 2015, **Vol.5, No.4 (Aug 2015)**

Abstract

Owing to its unique design, low cost, compact size and low specific speed, regenerative centrifugal pumps are the subject of increased interest in industrial applications. Apart from having these features, they are capable of delivering high heads with stable performance characteristics. They are capable of developing high pressure ratios in a single stage permitting to achieve a head which is equivalent to that of several centrifugal stages obtained from a single rotor with comparable tip speeds. In this paper, an attempt has been made to use computational fluid dynamics (CFD) software to simulate the flow within the regenerative pump and validate the CFD results with experimental data. Several numerical models were analyzed in the study for performance enhancement. It is observed that for the configuration with the outlet chamber having curved walls, the performance of the pump increased due to the decrease in the vortex flow in the impellers, thus minimizing the chances of head loss. Also, the configuration with the semicircular static fluid design on the sides of the impeller showed a significant improvement in the performance compared to the original model. The other design configurations show how the positioning of the impeller blades with respect to each other affects the efficiency of the pump by enhancing the toroidal motion of the fluid.

Keywords: *Regenerative centrifugal pump, Multi staging, Vortex flow, Toroidal motion, Flow field, Rotating stall, Unsteady analysis, Time periodic*

1. Introduction

The regenerative centrifugal pump is a rotor-dynamic machine that is capable of producing high heads in a single stage, but without the problems of wear and lubrication. It has lately excited inventors, engineers and manufacturers because of its unique design, stable characteristics and operational features (John Albert Oerich, 1953).

Although it is also known as the peripheral, turbine, drag, vortex, and side channel pump, the name regenerative centrifugal pump best describes the repeated fluid circulation during the flow process through the radial blades of the pump causing a toroidal motion in the blades of the impeller. The fluid in the pump moves helically in the casing and re-enters the impeller many times in its peripheral path from inlet to discharge. This repetitive action of the impeller blading, in effect, multi staging accounts for the high head per stage characteristics of this class of turbo machines.

*Corresponding author **Ankita Maity** is working as Assistant Manager, Quality Assurance, **Vignesh Chandrashekharan** as Product R&D Industrialization Engineer **and Muhammad Wasim Afzal** as Manager

The regenerative pump like centrifugal pump is a kinetic pump. However the regenerative pump can in many applications offer an effective alternative. The regenerative turbo machines impart both radial and axial component to fluid flow whereas centrifugal turbo machines take the fluid at the center of the impeller and push it radially outwards without imparting any axial component. One of the significant structural advantages of the regenerative type turbo machines is that no complex flow passages or vaning is required.

They are simple and easy to machine and there is no need of scrolls and diffusers. Performance wise, while it is true that centrifugal turbo machines are inherently more efficient than regenerative types, this may not be true under many conditions. The regenerative turbo machine, in its normal range of specific speeds, is efficient and compares favorably with that of centrifugal turbo machines. The head v/s flow rate graph for a regenerative turbo machine is nearly linear and slopes downwards against the more flat curve for a centrifugal pump. This means throttling a valve for a regenerative pump permits more precise changes in flow, without major overshooting or undershooting of the duty point, which frequently occurs with centrifugal turbo machine.

The pump has a construction in which the fluid enters a free rotating impeller having radial vanes machined at each side of its periphery. The vanes guide the fluid from inlet to outlet both separated by a 310° annulus chamber. As the fluid moves through the impeller, it produces a series of helical flows, returning the fluid repeatedly through the vanes for additional energy as it passes through an open annular channel. Thus the rotation of the impeller causes a helical corkscrew flow pattern of the fluid. Between discharge and inlet, the casing clearance is reduced to isolate the high pressure discharge from the low pressure inlet by means of a stripper. It allows only the fluid within the impeller blades to go out through the discharge port, thus also helping in maintaining the regenerative flow pattern. Clearances between the impeller disk and casing is minimum to prevent leakage from high pressure side back to low pressure side. Around greater portion of the periphery of the vanes, there is an annular flow channel with cross sectional area greater than that of impeller vanes. The fluid between the vanes is thrown in and out and across the annular channel repeatedly causing a violent mixing and the angular momentum acquired by the fluid as it circulates through the impeller vanes is transferred to the fluid in open channel.

The complex flow field within the pump represents a significant challenge to detailed analytical modeling (Badami, 1997; Engeda A., 2003). An improved and modified theoretical model was developed to enhance the regenerative pump performance. The concept of having one inlet angle and two exit angles for the twisted impeller blades was used that enhanced the pump head and efficiency (Meakhil and Park, 2005). It was also found that pressure head and efficiency strongly depended on the blade angles as well as blade geometry (Won *et. al.*, 2013). He exhibited that among all blade configurations, chevron angle of 30° yielded best performance. Also by varying the number of impeller blades and changing the geometry of the inlet and outlet passage of the pump, a substantial effect was seen on the pump performance (K. Vasudeva Karanth *et. al.*, 2015).

The above literature indicates that the effect of changing the static fluid design and the effect of making geometrical changes on just the outlet flow passage has not been the focus of study and hence an attempt has been made to numerically establish the effect of the above geometrical modification on pump performance. Also, continuing the work of K. Vasudeva Karanth *et. al.*, 2015, where they have shown, the number of blades having a co-relation with the pump performance, it has been shown how the positioning of the blades on either side of the impeller has an impact when it comes to pump performance.

The impeller of the regenerative pump having 36 radial blades as shown in figure 1. The performance test was carried out and the parameters of the pump were recorded before it was dismantled for geometrical measurements through CMM. Using these details, the numerical model was constructed on GAMBIT and CFD analysis was carried out. The numerical model was then validated with experimental results.

Fig.1 Pictorial view of the impeller having radial blades

2. Materials and Methods

The experimental works involved the setting up the pump and tabulating the readings which was later co-related with the CFD simulation. The entire set-up is shown in figure 2.

The pump used was CRI Self Priming Centrifugal Regenerative Monoblock Pump NR2 (0.5HP). Pipes were connected to the inlet and discharge ports. Vacuum gauge was connected at the inlet to measure the vacuum pressure on impeller. Pressure gauge was connected at the discharge to measure discharge head. Discharge head was fixed by using a gate valve and discharge was measured for a given period of time. This process was repeated for 20 different values of discharge head.

All the readings were tabulated and Discharge v/s Head curve and Efficiency v/s Discharge curve was plotted.

Maximum efficiency obtained is 22.82% at a mass flow rate of 0.519 kg/sec.

Fig.2 Experimental set-up

3. Numerical modeling

3.1. Geometry and grid generation:

The dimensions of the impeller was obtained using the CMM. Using these measured dimensions, the blade was modeled and a profile was created. The 3D singular blade was rotated 35 times at an angle interval of 10° to create the impeller. A 2D model of the inlet, outlet and static fluid was made and extruded by 7 mm along an edge to generate inlet and outlet and the static fluid region was created by rotating the 2D model of the static fluid from -171° to 171°. The face representing cross section of the side channel was used to create stripper. Then the subtract operation was applied to define the exact shape of the stripper starting from the point of intersection between side channel and outlet port to the point of intersection of inlet port with the side channel. Figure 3 shows the meshed configuration of the model.

Following the work of K. Vasudeva Karanth *et. al.*, 2015, the number of mesh elements were restricted to 2.3 million elements, since the variation in pressure values was not more than 0.4% for configurations containing elements greater than 2.3 million and also to save computational time. The inlet and outlet region had 144,000 elements, an impeller with 36x2 impeller vane fluid continuums of 6000 elements each, a 3000 element annular region outside the impeller which is integral with fluid continuum on either sides of the impeller, altogether having 1,540,678 elements. The number of elements in the whole of fluid domain, hence, consisted of 2,300,231 hexahedral elements.

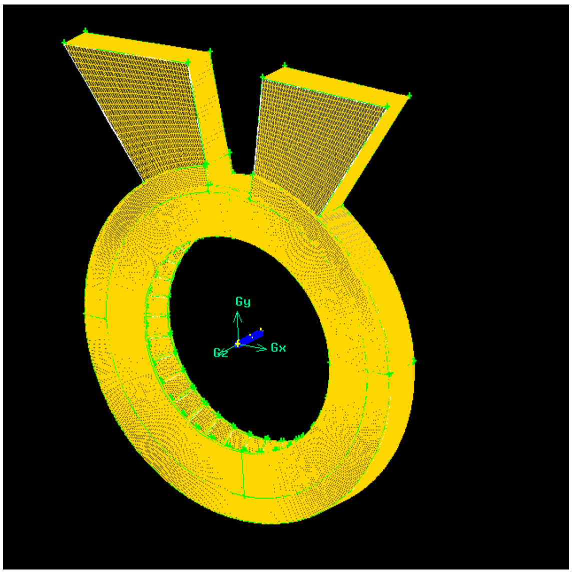

Fig.3 Meshed configuration of the model

3.2 Interfaces, boundary conditions and unsteady formulation

The interfaces were defined as 'inlet-static', 'outlet-static', 'stripper static', 'impeller side-static side' and 'impeller top-static bottom' as shown in figure 4. The boundary conditions used were 'mass flow inlet' at the entry of the pump and 'outflow' at the outlet flange of the pump. A zero slip wall condition was specified along the flow path for casing and vanes. The flow behavior was simulated using a k-ε model. Turbulence intensity was limited to 5% and a turbulent length scale of 0.026 m, which is the cube root of the domain volume, was adopted in the study. The unsteady formulation used was a second order implicit velocity formulation and the solver is pressure based. Discretization was carried out using the second order upwind scheme.

While defining the cell zone condition, the fluid was given a rotational speed of 301.71 rad/sec in anti-clockwise direction.

Rated speed of the pump = 2880 rpm

$$\omega = 2880 * 2\pi/60 = 301.71 \text{rad/sec}$$

Boundary conditions were defined for the impeller wall and it was made moving-rotational. The relative position between the rotor and the stator is updated with each time step of 5.78e-04 seconds, which is the time for 1° rotation of the impeller for a rated speed of 2880 rpm. Maximum iteration per time step was set to 15 and the number of time steps were set to 1080.

Since the nature of flow is unsteady, it was required to carry out the numerical analysis until the transient fluctuations of the flow field became time periodic. This was observed by judging the pressure fluctuations in the surfaces corresponding to, inlet of the pump, impeller exit and exit of the pump. In this analysis this has been achieved after 3 complete rotations of the impeller. The area weighted average values of the static and total pressure and velocity in the computational domain, corresponding to each time step was recorded and a time weighted average value was calculated corresponding to each rotation of the impeller by time step advancement.

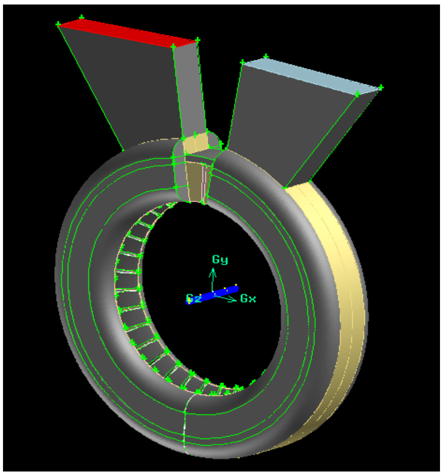

Fig.4 3D model of the pump showing interfaces

4. Validation of the Numerical results

The numerical model for the whole field flow calculations was validated by calibrating the results of the current numerical work with the experimental work. Different mass flow rates were fed and the corresponding pressure heads were obtained.

Figure 5 presents the validation plot for the numerical results with the experimental results. It can be seen from the graph that the head coefficient decreases with the increase in flow coefficient.

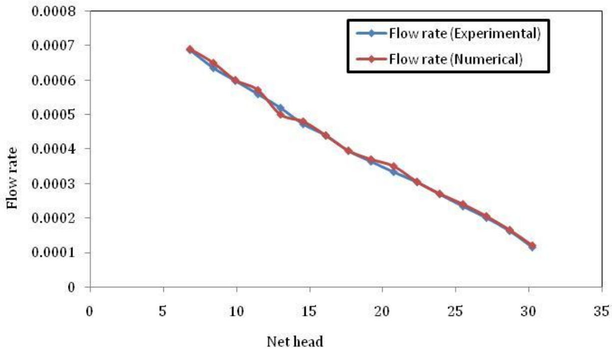

Fig.5 Graph showing the relationship between Head v/s Discharge for CFD data validation

5. Configurations with geometric modifications

Six numerical configurations are adopted in this study

1. Configuration 1 is the regenerative pump with 36 impeller blades which is the validated model.
2. Configuration 2 has a slight curvature in the outlet flow domain as shown in figure 6.
3. Configuration 3 has the static fluid on the sides of the impeller made semi-circular as shown in figure 7
4. Configuration 4 has blades mirrored with an offset of 2° as shown in figure 8.
5. Configuration 5 has different number of blades on either side of the impeller.

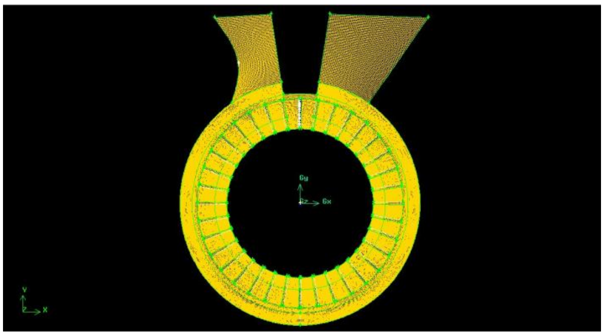

Fig. 6 Pump design showing curvature in the outlet domain

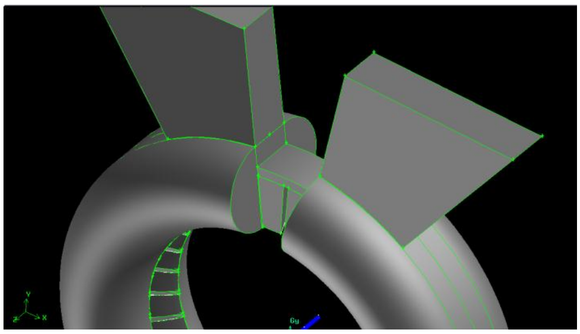

Fig. 7 Pump design having static fluid on the sides of the impeller made semi circular

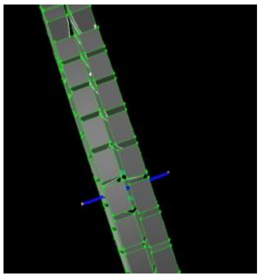

Fig. 8 Pump design having blades mirrored with an offset of 2°

6. Results and Discussions

The numerical analysis was carried out corresponding to the design point mass flow rate which was obtained from the experimental study of the regenerative pump. The static pressure and net pressure for the 5 configurations were recorded and compared after the unsteady analysis had reached a time periodic state, in this case after 3 complete rotations. The comparison of the pressure changes across all the models has been illustrated in Figure 14, Figure 15 and Figure 16.

6.1 Configuration 1

Figure 9(a) shows the contours of velocity magnitude and Figure 9(b) shows the velocity vector for the same configuration where a large rotating stall can be seen in the exit of the chamber of the pump. As quoted by K. Vasudeva Karanth *et. al.*, (2015), this rotating stall is attributed to the fact that fluid takes an abrupt turn towards the exit chamber from the impeller passage leading to a rotary motion before it moves towards the exit.

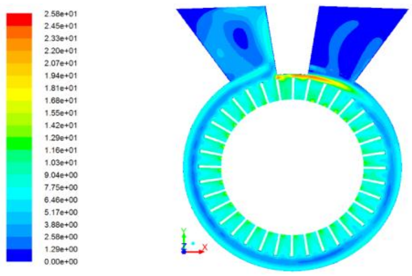

Fig.9 (a) Contours of velocity magnitude for configuration 1

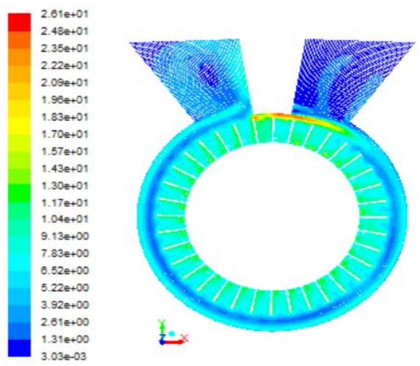

Fig.9 (b) Velocity vector plot for configuration 1

A closer look on the impeller vanes also shows the toroidal motion occurring in the vanes of the impeller as shown in Figure 9(c). The contours of static pressure is shown in Figure 9(d).

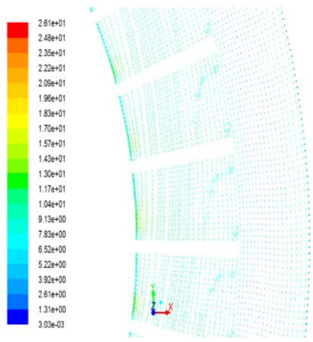

Fig.9(c) Toroidal motion in the impeller vanes for configuration 1

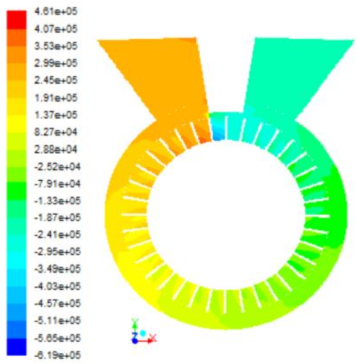

Fig.9(d) Contours of static pressure for configuration 1

The pressure gradually increases in magnitude as the fluid flows from inlet to outlet. The area weighted average of the total pressure was calculated and the same was used as a reference to study the pressure increment or decrement across rest of the models.

6.2 Configuration 2

Figure 10 shows contours of static pressure for configuration 2 which has a curvature in the outlet flow domain. The static pressure rise across the pump is 5.86% higher and total pressure is 5.58% higher than that of configuration 1. This can be attributed to the fact that the vortex flow has minimized as compared to the original resulting into higher heads. Also the large rotating stall at the exit of the pump has reduced.

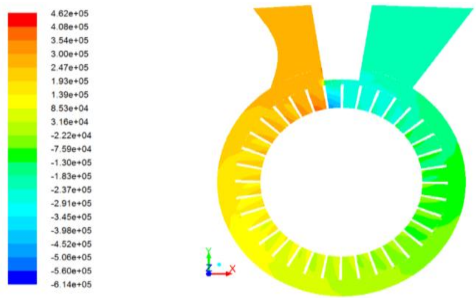

Fig.10 Contours of static pressure for configuration 2

6.3 Configuration 3

Figure 11 shows the contour plot of static pressure for configuration 3 where the static fluid on the sides of the impeller has been made semi-circular. The static pressure rise across the pump is 8.95% higher and total pressure increment is upto 9.41% higher than that of configuration 1. This is attributed to the fact that the semi-circular design enhances the circulatory fluid motion, thereby, increasing the efficiency of the pump.

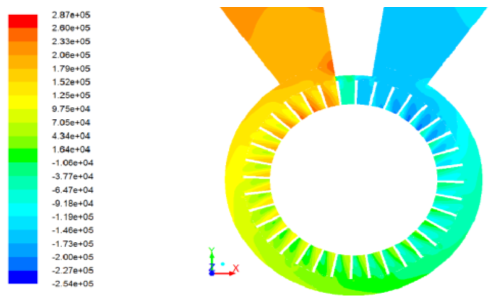

Fig.11 Contours of static pressure for configuration 3

6.4 Configuration 4

Figure 12 shows the contour plot for configuration 4 where the blades on either side of the impeller are given an offset of 2° while mirroring. The static pressure rise across the pump increased by 3.08% and total pressure increased by 3.82% than that of configuration 1.

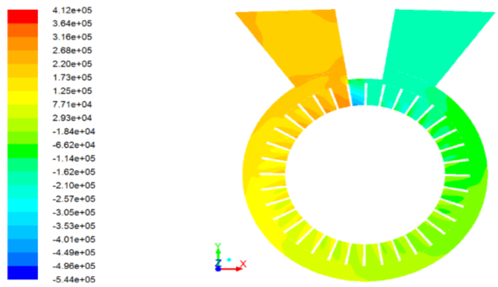

Fig.12 Contours of static pressure for configuration 4

6.5 Configuration 5

Figure 13 shows the contour plot for configuration 5 where the impeller has 36 blades on one side and 35 on the other side. The static pressure rise across the pump increased by 3.40% and total pressure increased by 4.40% than that of configuration 1.

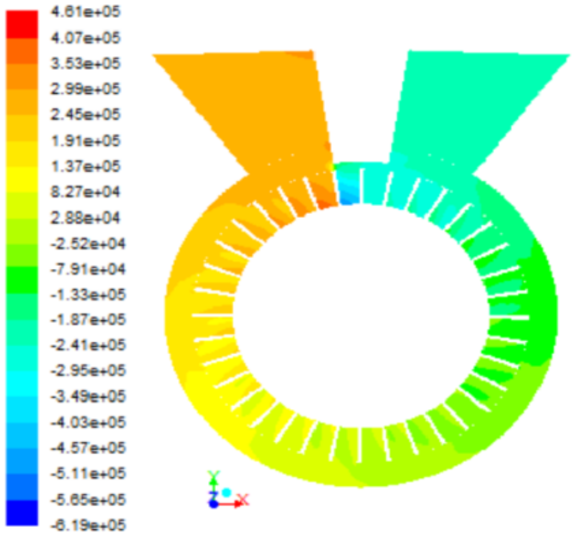

Fig.13 Contours of static pressure for configuration 5

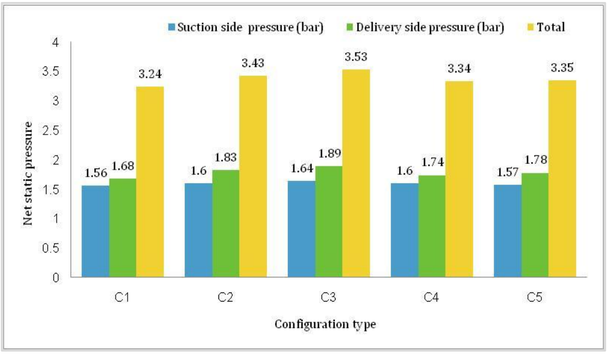

Fig.14 Comparative bar diagrams for net static pressure across all models

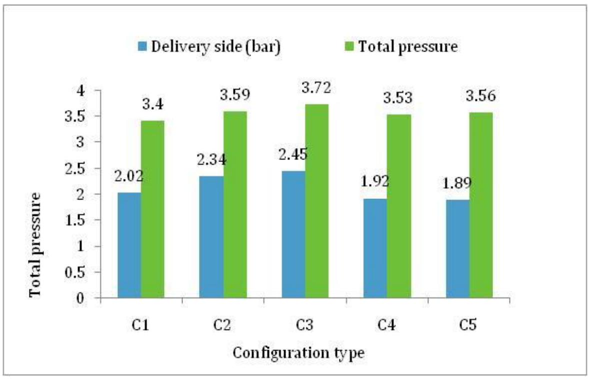

Fig.15 Comparative bar diagrams for total pressure across all models

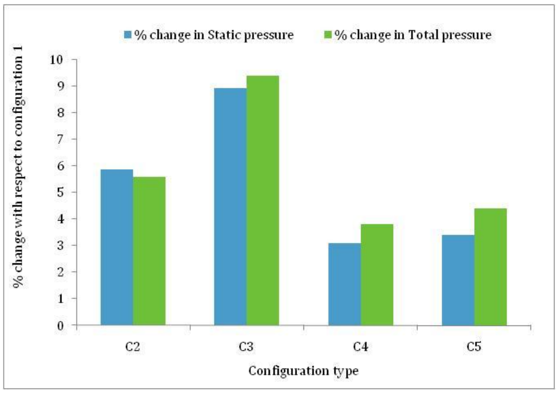

Fig.16 Comparative bar diagrams for percentage change in static and total pressure rise across all models

Conclusions

The numerical analysis using computational fluid dynamics helps in understanding and visualizing the flow phenomena that exists in the regenerative pump. The following changes in the model can contribute to performance enhancement.

1) A curvature in the outlet flow domain increases the net pressure head by minimizing the vortex flow, thus reducing the pressure head loss. As the area of the outlet flow reduces, the large rotating stall at the exit of the pump also reduces, thus leading to improved static and total pressure rise across the pump.
2) The semi-circular static fluid on the sides of the impeller enhances the circulatory fluid motion, thereby increasing the efficiency of the pump.
3) Positioning of the blades on either side of the impeller by offsetting enhances the fluid motion and results into the net increase in static and net pressure head.
4) The different number of impeller vanes on either side of impeller will also contribute to the increase in net pressure head.

Acknowledgement

The computational facilities were extended by the Department of Mechanical and Manufacturing Engineering, Manipal Institute of Technology, Manipal University, India, which is thankfully acknowledged.

References

Badami M, (1997), Theoretical and experimental analysis of traditional and new periphery pumps. *SAE Technical Papers* Series, No 971074, DOI: 10.4271/971074

Engeda A. (2003) Flow analysis and design suggestions for regenerative flow pumps *ASME FEDSM2003*-45681 Fluent 14, (2011), Ansys Inc., Cavendish Court Lebanon, NH, 03766, USA

Meakhail T. and Park S. O., (2005), An improved theory for regenerative pump performance, *Proceedings of the Institution of Mechanical Engineers*, Part A: Journal of Power and Energy, Vol. 219, pp. 213 – 222.

K. Vasudeva Karanth, 2015, Numerical study of a self priming regenerative pump for improved performance using geometric modifications, *International Journal of Current Engineering and Technology*, Vol.5, No.1, pp. 104-109.