Volume 6, Issue 1, January 2019 ISSN: 2348-8565 (Online)
IJMERT
International Journal of Modern Engineering and Research Technology
Website: http://www.ijmert.org Email: editor.ijmert@gmail.com

CFD Analysis on Impeller Passage of Centrifugal Pump

G. Narendar
Associate Professor
Department of Mechanical Engineering,
University College of Engineering,
Osmania University,
Hyderabad, (T.S.) [India]
Email: ouprofessor2000@gmail.com

G. Venod
PG Student
Department of Mechanical Engineering,
University College of Engineering,
Osmania University,
Hyderabad, (T.S.) [India]
Email: collectppt@gmail.com

S. Charvani
Associate Professor,
Institute of Aeronautical Engineering,
Hyderabad, (T.S.) [India]
Email: charvani2000@gmail.com

ABSTRACT

The flow patterns in centrifugal pump impellers with five, six, seven and eight blades, respectively, were measured. The effect of parameters on the velocity and pressure distribution of four impellers were employed to check the effects of the number of the impeller blades. Velocity measurements were performed at the different radial sections, where a yaw probe was traversed along the lines parallel to the impeller axis in the regions of the suction side (S), center (C), and pressure side (P), respectively. Additional measurements along the lines S' and P' were made for the impeller of Z = 6. The pressures on the passage wall were measured at the same radial positions as the velocity measurement on both the shroud and hub as well as on the blade surfaces. The distributions of the total pressure loss at each section of the impeller passage were obtained from the data of the velocity and static pressure measurements.

The standard k-ε turbulence model was chosen for turbulence model. From the simulation results it was observed that the pressure increases gradually from impeller inlet to outlet. The static pressure on pressure side is evidently larger than that on suction side at the same impeller radius. In addition to this, it was observed that, the velocity increases from impeller inlet until it enters the volute casing. It then drops to a minimum value at outlet region.

Keywords:— *Centrifugal pump, cfd analysis, velocity distribution.*

I. INTRODUCTION

A pump is a device that moves fluids (liquids or gases), or sometimes slurries, by mechanical action. Pumps can be classified into three major groups according to the method they use to move the fluid: direct lift, displacement, and gravity pumps.

Pumps operate by some mechanism (typically reciprocating or rotary), and consume energy to perform mechanical work by moving the fluid. Pumps operate via many energy sources, including manual operation, electricity, engines, or wind power, come in many sizes, from microscopic for use in medical applications to large industrial pumps.

II. Literature Review

Issa Chalghoum et al.[1] analyzed the centrifugal pump NS32 using numerical model and predicting the performance of a fluid flow in complex geometry, they conducted several numerical simulation with different turbulence models. To evaluate the effect of turbulence model on the flow characteristics, the k-ε, SST and SST-CC turbulence models were tested. The simulations have been made using the multiple reference frames (MRF) technique to take into account the impeller– volute interaction. To analyze the internal flow, simulation was carried out for several relative positions between the impeller blades and the volute tongue. In fact, the pressure fluctuations were numerically measured at five locations along the impeller and compared to experimental measurements. In addition, the unsteady pressure head evolution versus time was followed up. Then, an experimental validation of global and local characteristics of the pump was carried out.

Munish Gupta et al.[2] generated numerical model of an impeller and the complex internal flow fields are investigated by using the Ansys-CFX computational code. The internal flow is not quite smooth in the suction and pressure side of the blade due to non tangential inflow conditions which results in the flow separation at the leading edge. Pressure and velocity distribution inside impeller of the centrifugal pump has direct influence due to change of flow rate. Similar computational simulation models can also be used for analyzing the pressure and velocity of the turbines, compressor, fan and blower.

Miguel Asuaje. [3] performed a 3D-CFD simulation of the impeller and volute of a centrifugal pump using CFX codes. The pump has a specific speed of 32 (metric units) and an outside impeller diameter of 400mm. First, a 3D flow simulation for the impeller with a structured grid is presented. A sensitivity analysis regarding grid quality and turbulence models were also performed. The final impeller model obtained was used for a 3D quasi-unsteady flow simulation of the impeller-volute stage. A procedure for designing the volute, the non structured grid generation in the volute, and the interface flow passage between the impeller and volute are discussed. This flow simulation was carried out for several impeller blades and volute tongue relative positions. As a result, velocity and pressure field were calculated for different flow rates, allowing to obtain the radial thrust on the pump shaft.

Tilahun Nigussie [4] observed and determined the pattern of velocity profile and pressure distribution by using CFD simulation program after the 3D design and modeling of the pump is made using Vista CPD. Basically, this paper revolves around the idea of investigating the effect and distribution of velocity profile and pressure within a pump. 3D Navier Stokes equations were solved using ANSYS CFX.

The standard k turbulence model was chosen for turbulence model. From the simulation results it was observed that the pressure increases gradually from impeller inlet to outlet. The static pressure on pressure side is evidently larger than that on suction side at the same impeller radius. In addition to this, it was observed that, the velocity increases from impeller inlet until it enters the volute casing. It then drops to a minimum value at outlet region.

Ajith M S et al. [5] investigated the flow in the centrifugal pump impeller using the Ansys Fluent. Impeller is designed for the head (H) 70 m; discharge (Q) 80 L/sec; and speed (N) 1400 rpm. Impeller vane profile was generated by circular arc method and point by point method and CFD analysis was performed for the impeller vane profile. Further the impeller was analyzed for both forward and backward curved vane. The simulation on vane profile was solved by Nevier-Stokes equations with modified k - ω turbulence model. The impeller-Velocity and pressure distribution were analyzed for these Impellers.

B. Subbarao et al. [6] studied that, the pump is driven by 5.5 KW electric motor and the design is done in CFturbo 9 modeling package. The head and flow rate of this pump are 19.50 m and 20 LPS respectively and the motor speed is 2900 rpm. The number of impeller blade is 6 blades. The performance study of centrifugal pump is carried out after designing the dimensions of centrifugal pump. Simulation of present work is carried out in a commercial CFD software ANSYS fluent 14.5. The effect of parameters on the velocity and pressure distribution of four impellers with five, six, seven and eight blades, respectively, were employed to check the effects of the number of the impeller blades. Velocity measurements were performed at the different radial sections, where a yaw probe was traversed along the lines parallel to the impeller axis in the regions of the suction side (S), centre (C), and pressure side (P), respectively. Additional measurements along the lines S' and P' were also made for the impeller of Z = 6. The pressures on the passage wall were measured at the same radial positions as the velocity measurement on both the shroud and hub as well as on the blade surfaces. The distributions of the total pressure loss at each section of the impeller passage were obtained from the data of the velocity and static pressure measurements.

III. Solution Methodology

Fluid mechanics is the study of fluids either in motion (fluid dynamics) or at rest (fluid statics) and the subsequent effects of the fluid upon the boundaries, which may be either solid surfaces or interfaces with other fluids. Both gases and liquids are classified as fluids, and the number of fluids engineering applications is enormous: breathing, blood flow, swimming, pumps, fans, turbines, airplanes, ships, rivers, windmills, pipes, missiles, icebergs, engines, filters, jets, and sprinklers, to name a few. When you think about it, almost everything on this planet either is a fluid or moves within or near a fluid. In the beginning, the use of these techniques was customary only in the areas of aerospace and nuclear technology. Subsequently, the use has spread to a variety of products, physical situations, and manufacturing processes. Some examples of interesting applications of computational modeling are cooling of electronics systems, rotating and reciprocating machinery, furnaces, and combustion chambers.

IV. Modelling and Methodology

Boundary conditions are the set of conditions specified for the behavior of the solution to a set of differential equations at the boundary of its domain. Boundary conditions are important in determining the mathematical solutions to many physical problems. These conditions specify the flow and thermal variables on the boundaries of a physical model. They are, therefore, a critical component of simulation and it is important that they are specified appropriately. The boundary conditions are defined on cell faces and they do not have a finite thickness and they provide a means of introducing a step change in flow properties.

The design of centrifugal pump is divided in two categories: Impeller Design and Volute Design. The detailed procedure of single volute casing and impeller design can be found in different literature; in this paper vista CPD for the design of centrifugal pump is used. The duty parameters required by the pump are assumed to be:

1. Head = 20 m,
2. Flow rate = 280 m3/hr,
3. RPM = 1500,
4. Density = 1000 Kg/ m3,

Impeller Employed. The dimensions and specifications of the impellers tested are given in Figure 1, Four impellers with five, six, seven and eight blades, respectively, were employed to check the effects of the number of the impeller blades.

Velocity measurements were performed at the different radial sections, where a yaw probe was traversed along the lines parallel to the impeller axis in the regions of the suction side (S), center (C), and pressure side (P), respectively. Additional measurements along the lines S' and P' were also made for the impeller of Z = 6.

The pressures on the passage wall were measured at the same radial positions as the velocity measurement on both the shroud and hub as well as on the blade surfaces. The distributions of the total pressure loss at each section of the impeller passage were obtained from the data of the velocity and static pressure measurements.

The investigation was carried out at a rotational speed of n = 1500 rpm, at which the similarity law in the pump performance was ascertained to be established. The flow rate corresponded to the impeller of Z = 6, and for the impeller of Z = 5,7,8 tested.

Operating Conditions and Pump Geometry

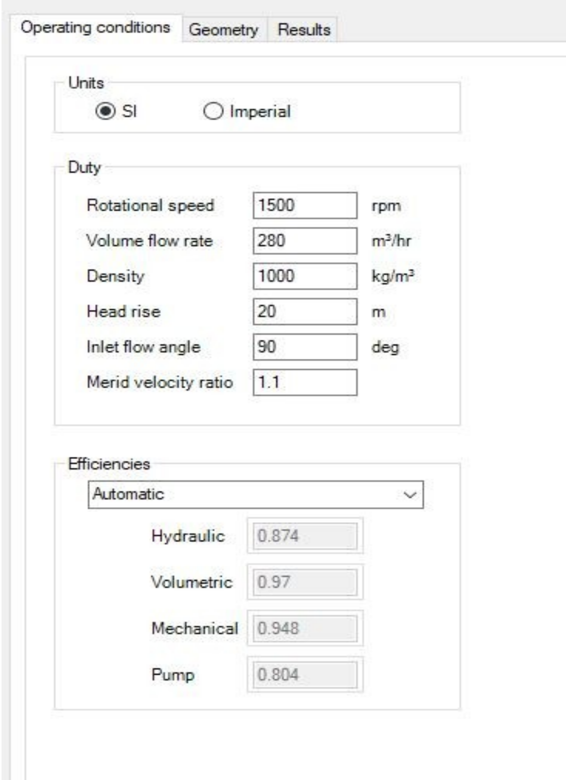

Figure 1: Operating Conditions

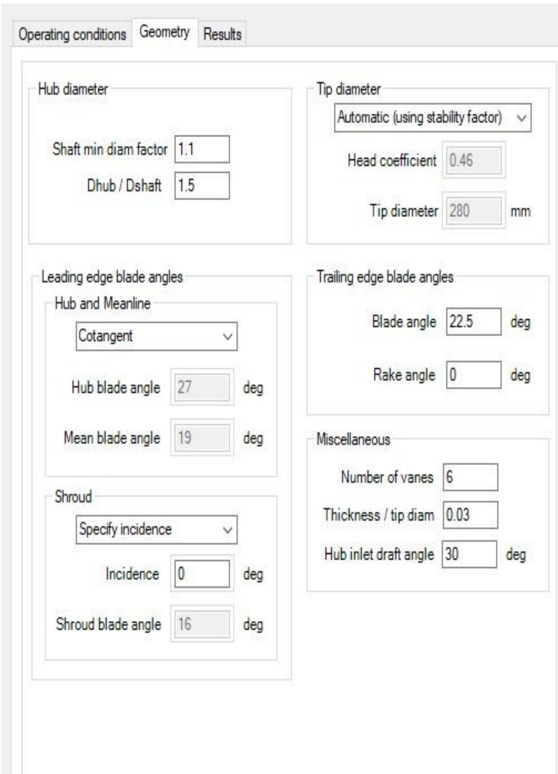

Figure 2: Pump Geometry

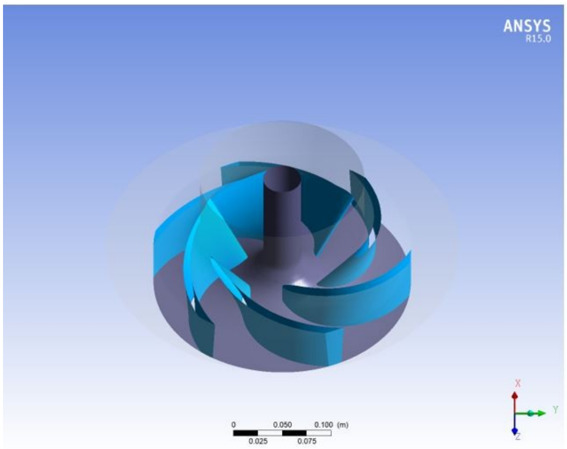

Figure 3: Model of Blade, Hub and Shroud

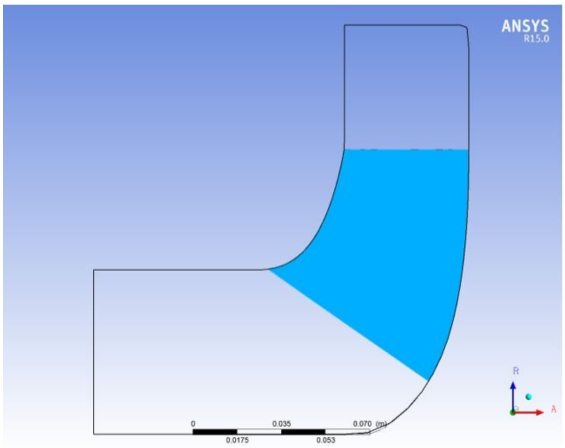

Figure 4: Blade, Hub and Shroud

Mesh Generation

The next step after modelling the pump components is to descretize it into smaller mesh elements. This process of discretization is called meshing. Meshing can be done by using different types of elements like tetrahedral, hexahedral, wedge, etc. In the present work, coarse tetrahedral and hexahedral mesh with different interval is used pump domain. The quality of mesh is checked by calculating the equisize skewness, aspect ratio & equiangle skewness.

Once the pump geometry has been specified and a mesh has been created, where the flow equations need to be solved. Mesh Information for CFX, number of nodes and elements are 413100 and 387585.

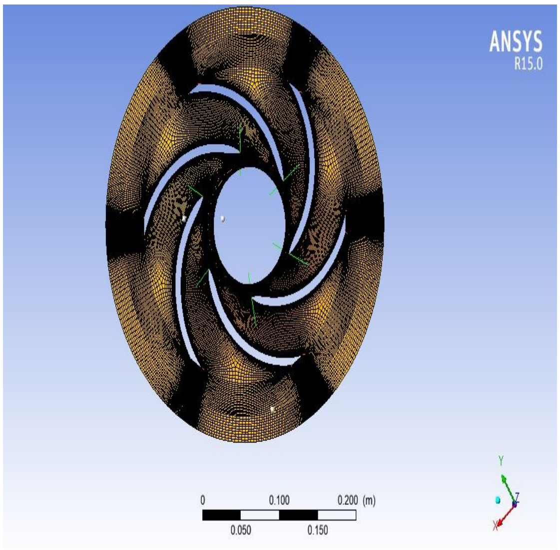

Figure 5: Meshing of total impeller

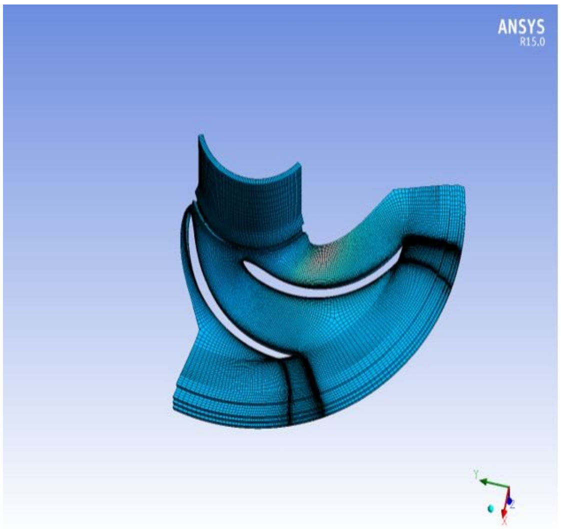

Figure 6: Mesh Elements at 50% Span

Flow Analysis

Design points for a parametric study can be specified using the required duty of the pump in the setup steps:

Input Material: Material is also assigned to the parts of the pump as: Casing and Impeller: Aluminium alloy, Hydraulic Region: Water, Rotating part: Rotating region. Boundary Conditions: Boundary conditions are applied to the inlet and outlet of the pump i.e. 0 Pa at inlet, 280 m^3/hr at outlet, and 1500 rpm.

Boundary Conditions

The boundary conditions are specified as follows:

- Mass flow is given at suction pipe entering suction.
- Inlet passage faces, rotating faces of impeller are considered as wall.
- At outlet face of delivery pipe section pressure outlet is applied.
- The suction pipe, impeller passages, inlet passage and delivery pipe considered as fluid zone.

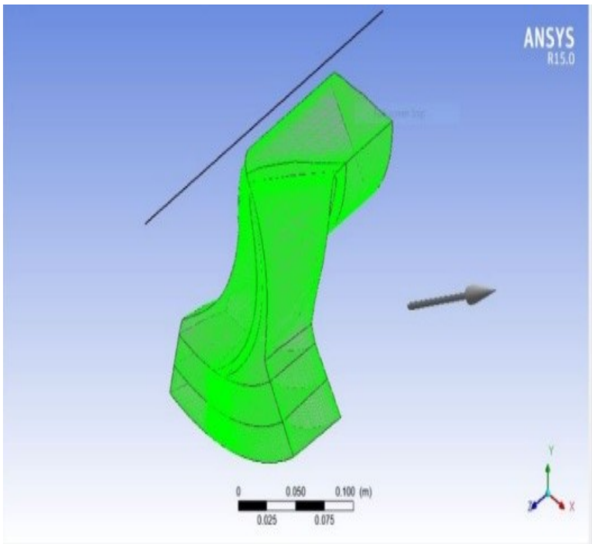

Figure 7: The RPM is given to impeller about z-axis

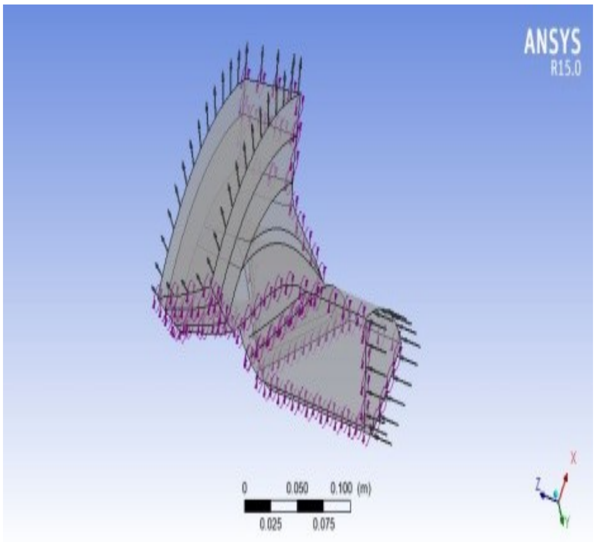

Figure 8: Flow of water in impeller blade

V. Results and Discussion

Experimental methods and past experience are undoubtedly important, but the most effective way to study pump performance is through Computational Fluid Dynamics (CFD).

The CFD-code (ANSYS Turbo system – R 15 (Ansys CFX), version 15), has been used in this project for the flow analysis of pump with end-suction volute type: The impeller and volute geometry was designed by Vista TF CPD V15 software by the required duty parameters by the pump to be design as a case study are: Head = 20 m, Flow rate = 280 m3/hr, RPM = 1500, Density = 1000 Kg/ m3, and the model prepared has been analyzed in CFD tool CFX and its performance is analyzed. It is found that the design and analysis methods lead to completely very good flow field predictions.

The distribution of pressure along stream wise direction at 20, 50 and 80 spans. Pressure increases gradually along stream wise direction within impeller passage and has higher pressure in pressure side than suction side of the impeller blade. Figure shows the distribution of velocity along stream wise direction at 20, 50 and 80 span.

The distribution of total pressure between the blades of the impeller is shown in the figure .The lowest total pressure appears at the inlet of the impeller suction side. This is the position where cavitation often appears in the centrifugal pump. The highest total pressure occurs at the outlet of impeller, where the kinetic energy of flow reaches maximum.

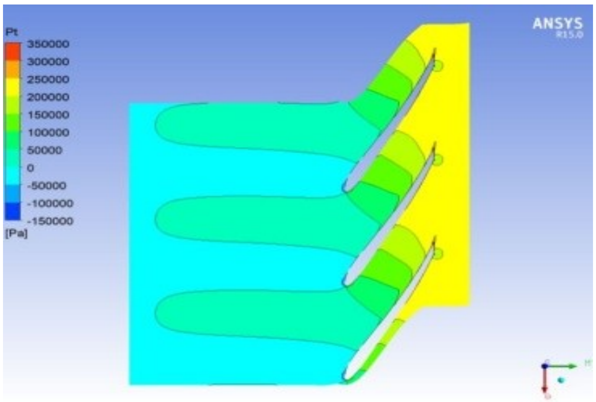

Figure 9: Contour of Pt at 50% Span

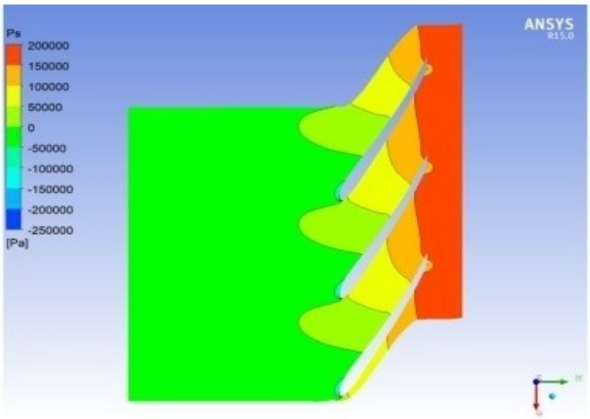

Figure 10: Contour of Ps at 50% Span

The figure shows the static pressure distribution at the span of 50 between the blades of the impeller. It is observed that the static pressure inside the impeller blades is asymmetry distributed. The minimum static pressure area appears at the back of the impeller blade at suction side, at the inlet and maximum static pressure at the exit.

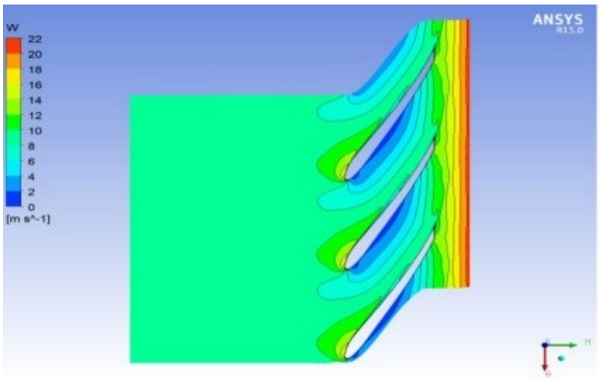

Figure 11: Contour of W at 50% Span

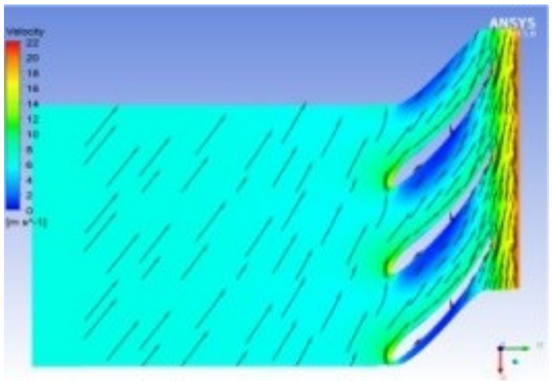

Figure 12: Velocity Vectors at 20% Span

Figure shows the distribution of velocity along stream wise direction at 20, 50 and 80 span. The separation of flow can be seen at the blade leading edge. Since, the flow at the inlet of impeller is not tangential to the blade, the flow along the blade is not uniform and hence the separation of flow takes place along the surface of blade.

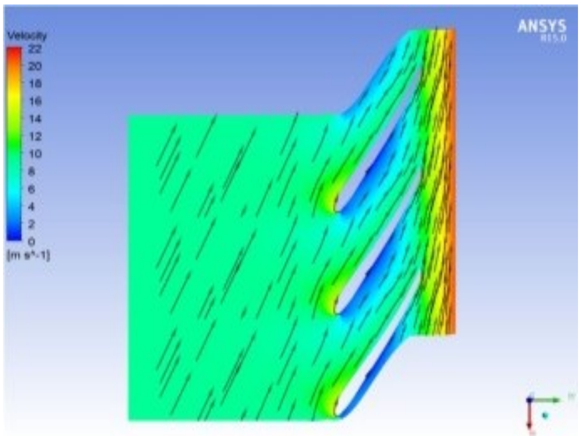

Figure 13: Velocity Vectors at 50% Span

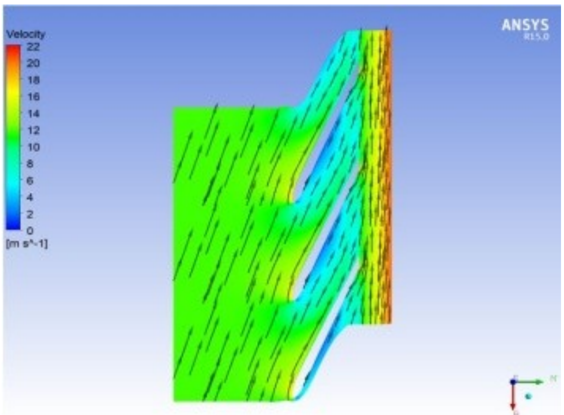

Figure 14: Velocity Vectors at 80% Span

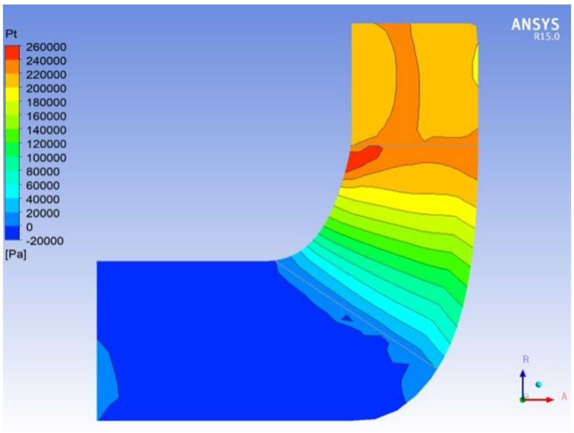

Figure 15: Contour of Mass Averaged Pt

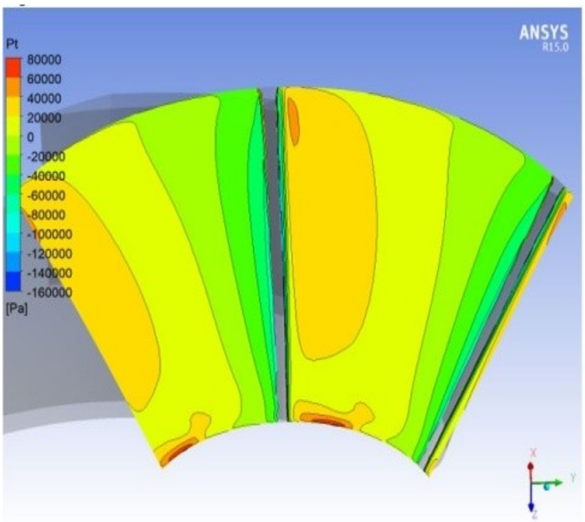

Figure 16: Contour of Pt at Blade LE

Circumferential Plots

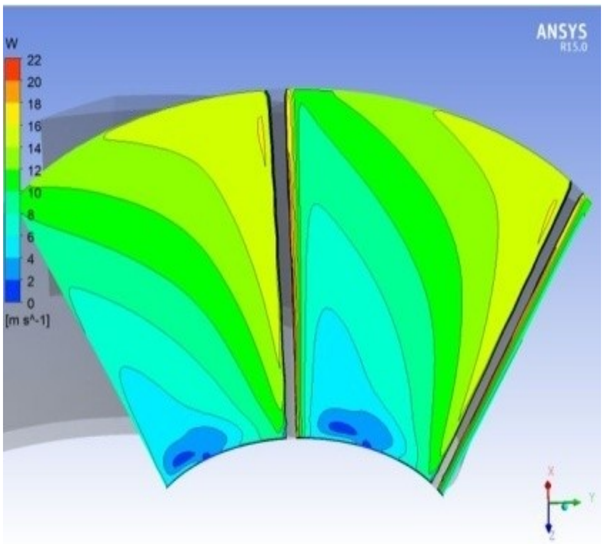

Figure 17: Contour of W at Blade LE

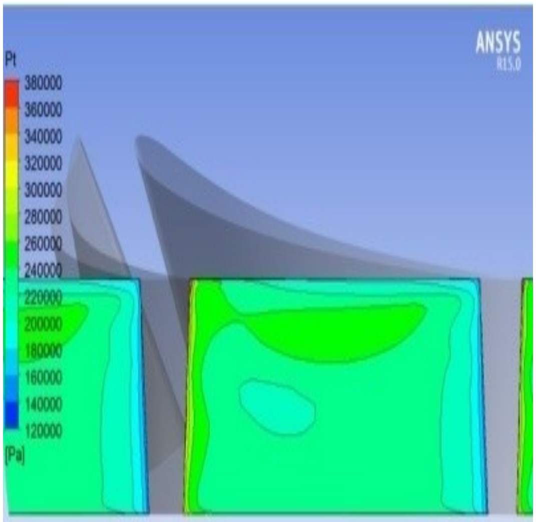

Figure 18: Contour of Pt at Blade TE

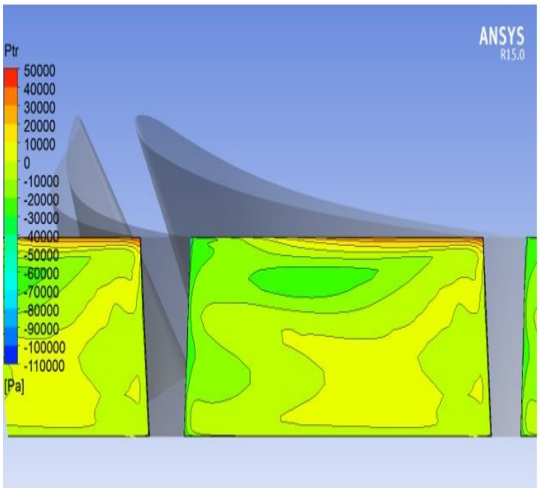

Figure 19: Contour of Ptr at Blade TE

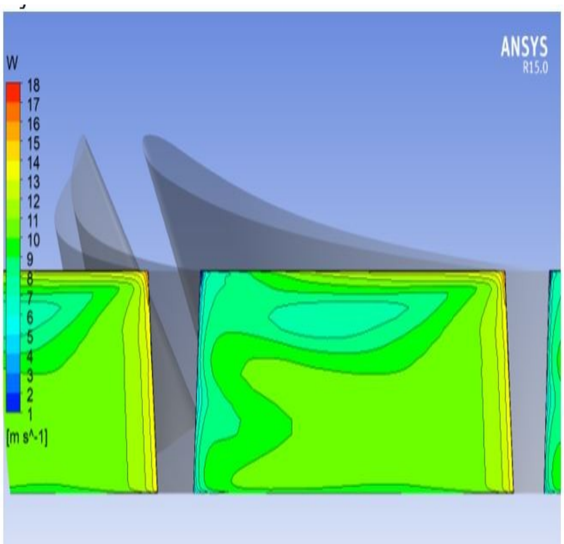

Figure 20: Contour of W at Blade TE

Streamline Plot

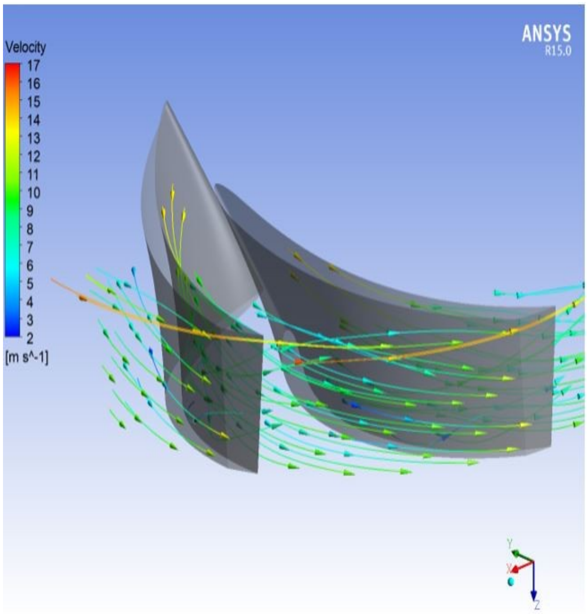

Figure 21: Velocity Streamlines at Blade TE

The table given below gives a summary of the performance results for the pump impeller.

Table 1: Performance Results for the Pump Impeller

Rotation Speed	157.08 radian/ s
Diameter	0.263 m
Volume Flow Rate	0.078 m^3/ s
Head (LE-TE)	22.91 m
Head (IN-OUT)	22.34 m
Flow Coefficient	0.027
Head Coefficient (IN-OUT)	0.128
Shaft Power	17.8 kW
Power Coefficient	0.0036
Total Efficiency (IN-OUT) %	95.7
Static Efficiency (IN-OUT) %	81.4

Blade Loading Charts

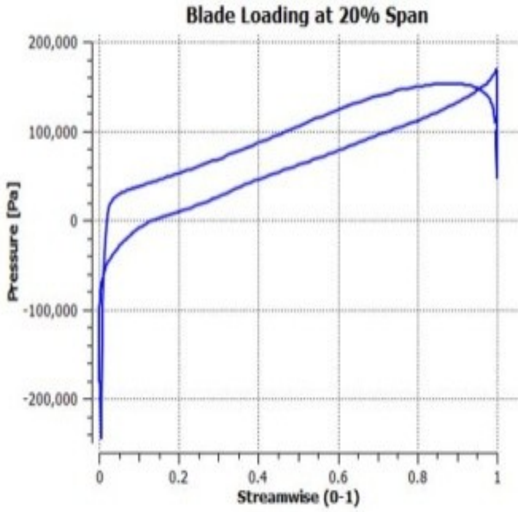

Figure 22: Blade Loading at 20% Span

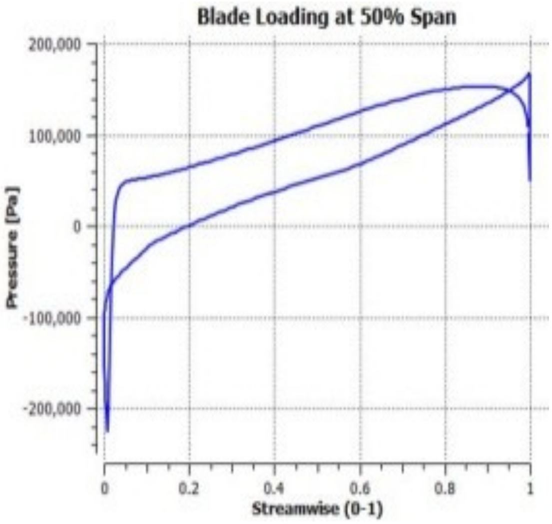

Figure 23: Blade Loading at 50% Span

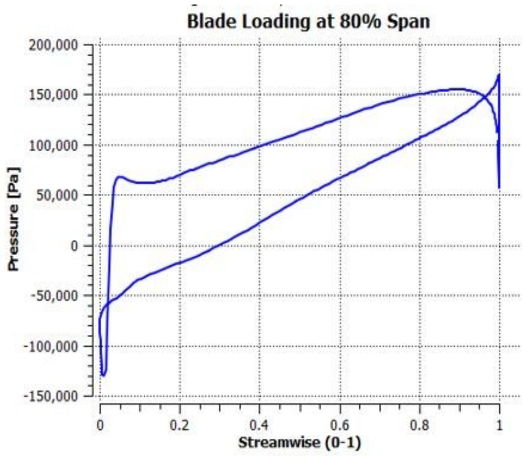

Figure 24: Blade Loading at 80% Span

Streamwise Charts

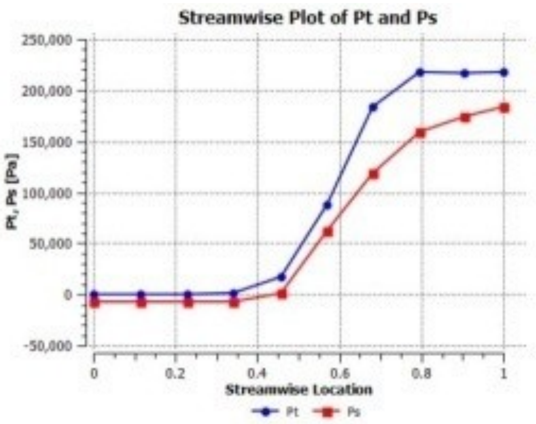

Figure 25: Streamwise Plot of Pt and Ps

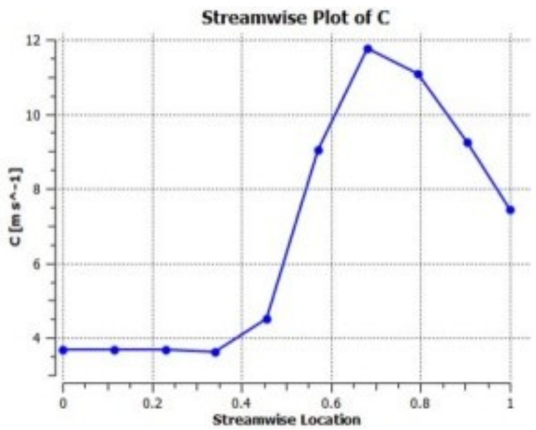

Figure 26: Streamwise Plot of C

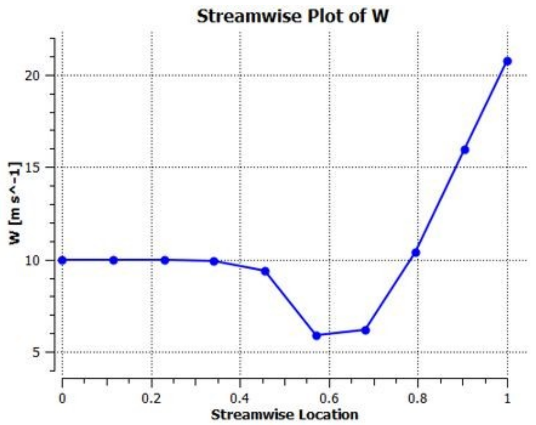

Figure 27: Streamwise Plot of W

Pressure distribution in different blade numbers:

The pressure distribution in centrifugal pump impellers with five, six, seven and eight blades, respectively, were measured. The effect of parameters on the pressure distribution of four impellers with five, six, seven and eight blades, respectively, were employed to check the effects of the number of the impeller blades.

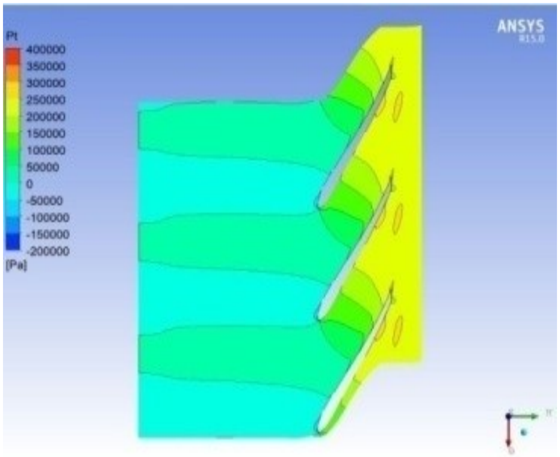

Figure 28: Contour of Pt at 50% Span for 5 blade

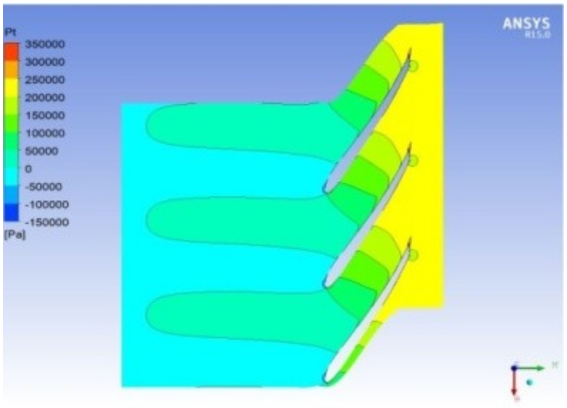

Figure 29: Contour of Pt at 50% Span for 6 blades

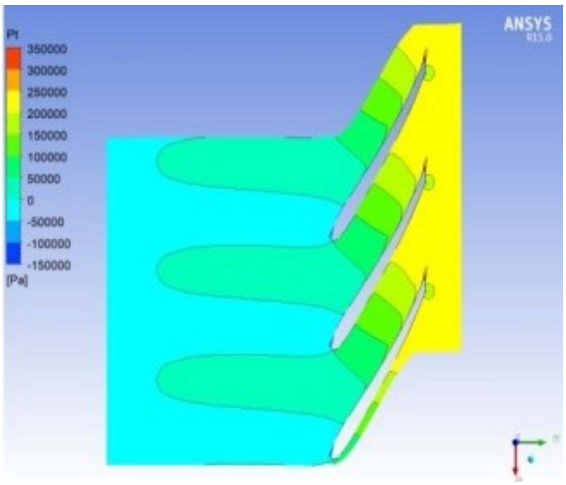

Figure 30: Contour of Pt at 50% Span for 7 blades

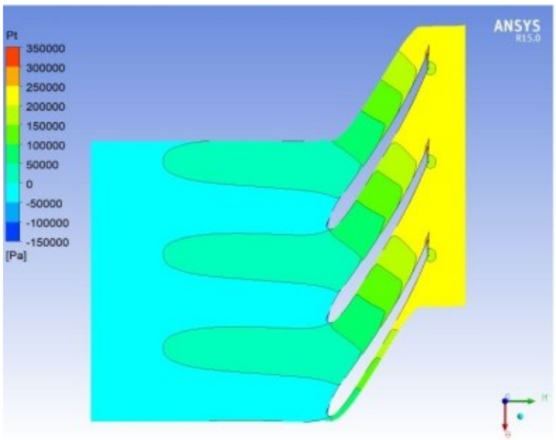

Figure 31: Contour of Pt at 50% Span for 8 blade

Pressure increases gradually along stream wise direction within impeller passage and has higher pressure in pressure side than suction side of the impeller blade. Figure shows the distribution of pressure along stream wise direction for different blade numbers.

The effect of parameters on the velocity and pressure distribution . Four impellers with five, six, seven and eight blades, respectively, were employed to check the effects of the number of the impeller blades. Velocity measurements were performed at the different radial sections, where a yaw probe was traversed along the lines parallel to the impeller axis in the regions of the suction side (S), center (C), and pressure side (P), respectively.

The pressures on the passage wall were measured at the same radial positions as the velocity measurement on both the shroud and hub as well as on the blade surfaces. The distributions of the total pressure loss at each section of the impeller passage were obtained from the data of the velocity and static pressure measurements.

The distribution of total pressure between the blades of the impeller is shown in the figure. The lowest total pressure appears at the inlet of the impeller suction side. This is the position where cavitation often appears in the centrifugal pump. The highest total pressure occurs at the outlet of impeller, where the kinetic energy of flow reaches maximum. As the number of blade increases pressure inside the impeller increases gradually along stream wise direction with in impeller passage.

VI. CONCLUSIONS

A numerical model of an impeller has been generated and the complex internal flow fields are investigated by using the Ansys-CFX computational code. The internal flow is not quite smooth in the suction and pressure side of the blade due to non tangential inflow conditions which results in the flow separation at the leading edge. Pressure and velocity distribution inside impeller of the centrifugal pump has direct influence due to change of blade number. Similar computational simulation models can also be used for analyzing the pressure and velocity of the turbines, compressor, fan and blower.

Future work aims to investigate the cavitation flow in the pump impeller and analyze its effect on pump performance and efficiency, as well as to estimate erosion zone. It is necessary to study the occurrence of the cavitation in order to avoid the fall of performance due to this phenomenon. The most important aspect of this study is to put a new method of design, which minimizes the occurrence of cavitation, and increases the efficiency of the pump.

REFERENCES:

[1] Munish Gupta, Satish Kumar, Ayush Kumar: "Numerical Study of Pressure and Velocity Distribution Analysis of Centrifugal Pump" 1 Dec. 2011

[2] MiguelAsuaje, FaridBakir, FrankKenyery: "Numerical Modelization of the Flow in Centrifugal Pump: Volute Influence in

Velocity and Pressure Fields" 5 January 2004

[3] Tilahun Nigussie, Edessa Dribssa: "Design and CFD Analysis of Centrifugal Pump" Volume 3, Issue 3, May-June, 2015

[4] Ajith M S, Dr. Jeoju M Issac: "Design and analysis of centrifugal pump impeller using Ansys fluent. Volume 4, Issue 10, October 2015

[5] B Subbarao, Dr. E. Ramjee, Dr. M. Devaiah, Dr. T. Siva Prasad: "Investigation of into flow field of centrifugal pump impeller". Volume 7, Number 2 (2017)

[6] Issa Chalghoum, Hatem Kanfoudi, "Numerical Modeling of the Flow Inside a Centrifugal Pump: Influence of Impeller–Volute Interaction on Velocity and Pressure Fields", Arab J Sci Eng (2016) 41: 4463

* * * * *