INTERNATIONAL JOURNAL OF TECHO-ENGINEERING
ISSN : 2057-5688

DESIGN AND OPTIMIZATION OF CENTRIFUGAL IMPELLER OF VARIOUS DESIGNS USING CFD

GONDU JAGADEESH[1], D. CHANDRASEKHAR[2]

[1]PG Scholar, [2]Assistant professor, department of Mechanical engineering, Sarada institute of science, technology and management (51), Srikakulam

Abstract: Centrifugal pumps are a sub-class of dynamic axisymmetric work-absorbing turbo machinery and used to transport fluid by conversion of rotational kinetic energy to hydrodynamic energy. This project investigates the study of complex internal flows in centrifugal pump impellers with the aid of computational fluid dynamics software thus facilitating the design of pumps. Here three different types of pump impellers had been taken. The pump specifications considered for investigation are discharge and speed. These specifications have been varied to perform a comparative study of these pump impellers. The impeller was modelled in CATIA and the blade-to-blade plane of the impellers was taken for the detailed study purpose because the flow occurs through this passage only. The blade-to-blade plane is modelled in CATIA software and the flow analysis is carried out using CFD software. Thus, the valid results regarding the velocity distribution and pressure distributions were predicated and the performance of those pumps had been compared from the computational results.

Keywords: Centrifugal pump, Impeller analysis, CFD analysis.

I. INTRODUCTION

An impeller is a rotating component of a centrifugal pump which transfers energy from the motor that drives the pump to the fluid being pumped by accelerating the fluid outwards from the centre of rotation. The velocity achieved by the impeller transfers into pressure when the pump casing confines the outward movement of the fluid. Impellers are usually short cylinders with an open inlet (called an eye) to accept incoming fluid, vanes to push the fluid radially, and a splined, keyed, or threaded bore to accept a drive-shaft.

Figure 1: Types of impeller blade

The impeller made out of cast material in many cases may be called rotor, also. It is cheaper to cast the radial impeller right in the support it is fitted on, which is put in motion by the gearbox from an electric motor, combustion engine or by steam driven turbine. The rotor usually names both the spindle and the impeller when they are mounted by bolts. In the case of where flow simply passes through a straight pipe to enter a centrifugal compressor; the flow is straight, uniform and has no vorticity. As illustrated below $\alpha 1 = 0°$. As the flow continues to pass into and through the centrifugal impeller, the impeller forces the flow to spin faster and faster.

In Centrifugal Compressors: The main part of a centrifugal compressor is the impeller. An open impeller has no cover; therefore, it can work at higher speeds. A compressor with a covered impeller can have more stages than one that has an open impeller.

Some impellers are similar to small propellers but without the large blades. Among other uses, they are used in water jets to power high speed boats. Since impellers have no large blades to turn, they can spin at much higher speeds than propellers. The water forced through the impeller is channelled by the housing, creating a water jet that propels the vessel forward. The housing is normally tapered into a nozzle to increase the speed of the water, which also creates a Venturi effect in which low pressure behind the impeller pulls more water towards the blades, tending to increase the speed. To work efficiently, there must be a close fit between the impeller and the housing. The housing is normally fitted with a replaceable wear ring which tends to wear as sand or other particles are thrown against the housing side by the impeller. Vessels using impellers are normally steered by changing the direction of the water jet.

ISSN : 2057-5688

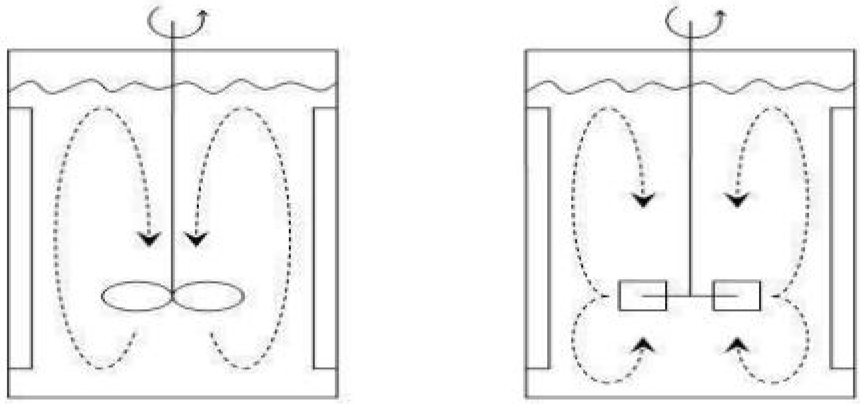

Figure 2: Axial flow and Radial flow impeller

Impellers in agitated tanks are used to mix fluids or slurry in the tank. This can be used to combine materials in the form of solids, liquids and gas. Mixing the fluids in a tank is very important if there are gradients in conditions such as temperature or concentration.

Computational Fluid Dynamics (CFD) is the simulation of fluids engineering systems using modelling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.). The process is as figure 1. Figure 1 Process of Computational Fluid Dynamics Firstly, we have a fluid problem. To solve this problem, we should know the physical properties of fluid by using Fluid Mechanics. Then we can use mathematical equations to describe these physical properties. This is Navier-Stokes Equation and it is the governing equation of CFD. As the Navier-Stokes Equation is analytical, human can understand it and solve them on a piece of paper. The translators are numerical discretization methods, such as Finite Difference, Finite Element, Finite Volume methods. Consequently, we also need to divide our whole problem domain into many small parts because our discretization is based on them. Then, we can write programs to solve them. The typical languages are Fortran and C. Normally the programs are run on workstations or super computers. At the end, we can get our simulation results. We can compare and analyse the simulation results with experiments and the real problem. If the results are not sufficient to solve the problem, we have to repeat the process until find satisfied solution. This is the process of CFD.

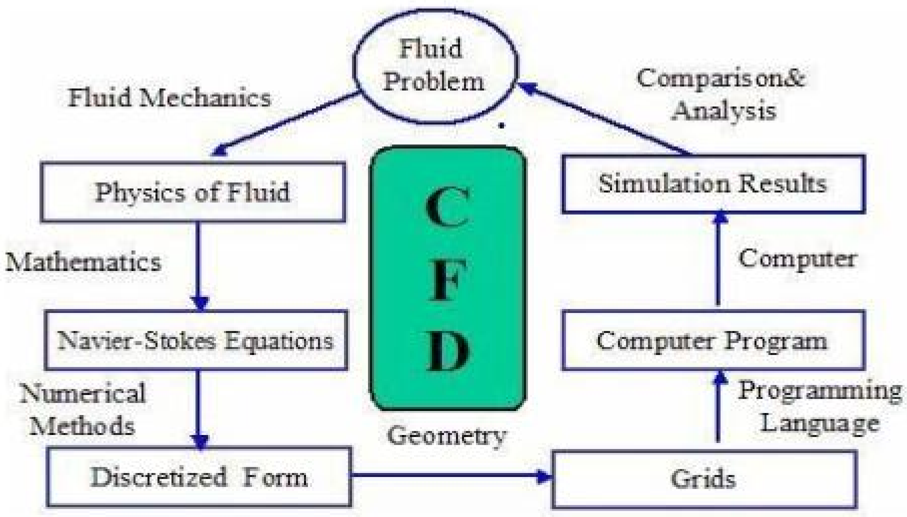

Figure 3: Flow chart of CFD

Computational Fluid Dynamics (CFD) is a powerful tool used to model the real-life behaviour of fluids. It allows the optimisation of design parameters without the need for the costly testing of multiple prototypes. What is more, it is also a powerful graphical tool for visualising flow patterns that can give insight into flow physics that otherwise would be very difficult and costly to discover experimentally, if possible, at all. Governing equations exist to model fluid behaviour, but it is not always possible to apply them to many of the complex flow patterns we see in the real world directly as there would be too many unknown variables. However, CFD involves creating a computational mesh to divide up real world continuous fluids into more manageable discrete sections. The governing equations for fluid flow can then be applied to each section individually, but as the properties of each section are inevitably linked to its neighbouring sections, all the sections can be solved simultaneously until a full solution for the entire flow field can be found. This method obviously requires a huge amount of computational power. As with everything, CFD is not without its limitations. Its accuracy or validity are dependent on a multitude of different factors: the quality and appropriateness of the mesh, the degree to which the chosen equations match the type of flow to be modelled, the interpretation of the results, the accuracy of the boundary conditions entered by the user or the level of convergence of the solution, to name but a few.

CFD: Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of mathematical modelling (partial differential equations) numerical methods (discretization and solution techniques) software tools (solvers, pre and postprocessing utilities) CFD enables scientists and engineers to perform 'numerical experiments (i.e. computer simulations) in a 'virtual flow laboratory 'Often it comes down to the skill of the user, as each flow problem will be slightly different and as a result, will require a slightly different modelling approach. However, experimental data can provide a valuable reference point with which to check the validity of CFD models.

II. Literature Survey

Extensive research work in the area of impeller has been going on over the last few decades in order to improve their performance. The flow process is highly complex in the water pump impellers and it can be predicted well with the aid of CFD and thus facilitating the design of pumps. A CFD approach seems a logical way to have a detailed look at the flow behaviour and to predict the regions of separation with a high degree of accuracy. Thus, CFD is an important tool for pump designers. Oh J.S, RO H.S and Goto. AOh and Ro used a compressible time marching method, a traditional Simple method, and a commercial program of CFX-TASC flow to simulate flow pattern through a water pump and compared the difference between among these methods in predicting the pumps performance. Go to presented a comparison between the measured and computed exit-flow fields of a mixed flow impeller with various tip clearances, including the shrouded and un shrouded impellers, and confirmed the applicability of the incompressible version of the three-dimensional Navier-stokes code developed by Dawes for a mixed flow centrifugal pump. ZhouWeidong, Ng and his colleagues also developed a three – dimensional time marching, incompressible Navier-stokes solver using the pseudo compressibility technique to study the flow field through a mixed flow water-pump impeller. The applicability of the original code was validated by comparing it with many published experimental and computational results. Kaupert, potts, Tsukamoto Kaupert and his colleagues. Although these researchers predicted reverse flow in the impeller shroud region at small flow rates numerically, some contradictions still existed. Kaupert's experiments showed the simultaneous appearance of shroud-side reverse flow at the impeller inlet and outlet but his CFD results failed to predict the numerical outlet – reverse flows. Sun and Tsukamoto validated the predicted results of the head-flow curves, diffuser inlet pressure distribution, and impeller radial forces by revealing the experimental data over the entire flow range, and they predicted back flow at small rates, but they did not show an exact back- flow pattern along the impeller outlet.

E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris (2008) in this study, the performance of impellers with the same outlet diameter having different outlet blade angles is thoroughly evaluated. The One-dimensional approach along with empirical equations is adopted for the design of each impeller. The predicted performance curves result through the calculation of the internal flow field. Headdischarge curve play important role into different outlet angles. The influence of the outlet blade angle on the performance is verified with the CFD. The performance curve becomes smoother and flatter with the increase with the increase outlet blade angle. At nominal capacity, when the outlet blade angle was increased from 20° to 50°, the head was increased by more than 6% but the hydraulic efficiency was reduced by 4.5%. However, at high flow rates, the increase of the outlet blade angle caused a significant improvement of the hydraulic efficiency.

LIU Houlin, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai (2010) Blade number play the important role during designing the pump which affects the characteristics of the pump. The model pump has a design specific speed of 92.7 and an impeller with 5 blades. The blade number is varied to 4, 6, 7 with the casing and other geometric parameters keep constant. The inner flow fields and characteristics of the centrifugal pumps with different blade number are simulated and predicted in non-cavitation and caviation conditions by using commercial code FLUENT. Using rapid prototyping the impeller with different blade numbers is made. With the increase of blade number, the area of low-pressure region at the suction of blade inlet grows continuously, and the uniformity of static pressure distribution at screw section become worse and worse while at diffusion section become better and better.

B.Mohan, B.E.Kumar (2011) the novel axial composite impeller has been developed using commercial tools pro-e. They have chosen the suitable materials for this study, namely Kevlar-49, Carbon and S-Glass with a standard epoxy resin for the composite matrix. Static and dynamic behaviours of the component were analysed using finite element analysis commercial tool ANSYS 14.5. They have analysed the stress distributions and displacements on the composite impeller in static analysis. The stress concentration regions were identified in this analysis. For transient analysis, we have applied dynamic force at various operating speeds of the impeller and analyzed the deflections and stress concentration regions.

S.Rajendran, Dr.K.Purushothaman (2012) described the simulation of the flow in the impeller of a centrifugal pump. The analysis of centrifugal pump impeller design is carried out using ANSYS-CFX. The complex internal flows in Centrifugal pump impellers can be well predicted through ANSYS-CFX. The numerical solution of the discredited three-dimensional, incompressible Navier-Stokes equations over an unstructured grid is accomplished with an ANSYS-CFX.

A Syam Prasad, BVVV Lakshmipathi Rao, A Babji, Dr P Kumar Babu (2013) It described the static and dynamic analysis of a centrifugal pump impeller which is made of three different alloy materials (viz., Inconel alloy 740, Incoloy alloy 803, Warpaloy) to estimate its performance. The investigation has been done by using CATIA and ANSYS13.0 softwares. A structural analysis has been carried out to investigate the stresses, strains and displacements of the impeller and modal analysis has been carried out to investigate the frequency and deflection of the impeller. An attempt is also made to suggest the best alloy for an impeller of a centrifugal pump by comparing the results obtained for three different alloys.

III. Proposed Methodology

The design of the centrifugal impeller blade was designed in CATIA and the types of impeller blade we use to analyse are three blade impeller, four blade impeller, twisted four blade impeller. CATIA (an acronym of computer-aided three-dimensional interactive application) is a multi-platform software suite for computer-aided design (CAD), computer aided manufacturing (CAM), computer aided engineering (CAE), PLM and 3D, developed by the French company Dassault Systems. CATIA offers a solution to shape design, styling, surfacing workflow and visualization to create, modify and validate complex innovative shapes from industrial design to Class-A surfacing with the ICEM surfacing technologies.

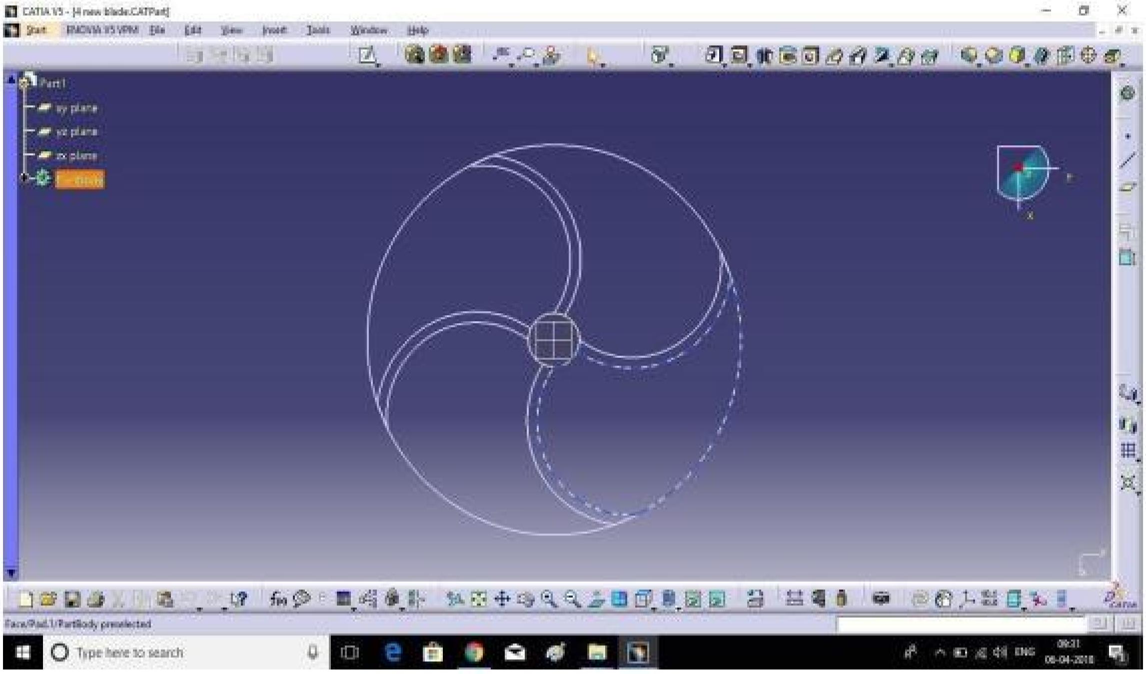

Figure 4: Wireframe views of 4 blades

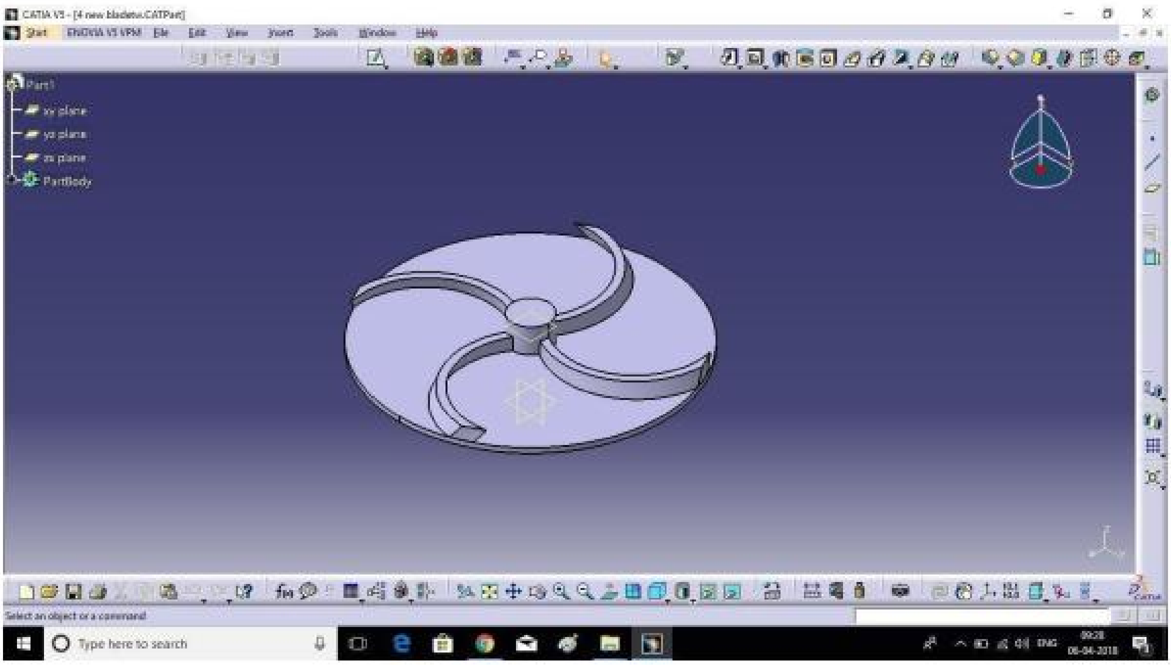

Figure 5: isometric view of twisted 4 blades

CATIA supports multiple stages of product design whether started from scratch or from 2D sketches(blueprints). Systems engineering The CATIA Systems Engineering solution delivers a unique open and extensible systems engineering development platform that fully integrates the cross-discipline modelling, simulation, verification and business process support needed for developing complex 'cyber- physical' products. It enables organizations to evaluate requests for changes or develop new products or system variants utilizing a unified performance-based systems engineering approach. The solution addresses the Model Based Systems Engineering (MBSE) needs of users developing today's smart products and systems and comprises the elements.

IV. Results and Discussions

Figure 7 shows the velocity streamlines for the initial and optimized impeller designs. In the initial design, the velocity vectors indicate that the flow does not perfectly follow the blade profile in the impeller passage. Flow exhibits slight irregular behaviour. In the optimized design, however, the flow is nicely aligned with the blades.

Contents	Velocity (ms-1)	Pressure (pa)
Three blades	8.534e0	3.104e004
Four blades	7.417e0	3.775e004
Twisted four blades	1.363e002	2.253e001

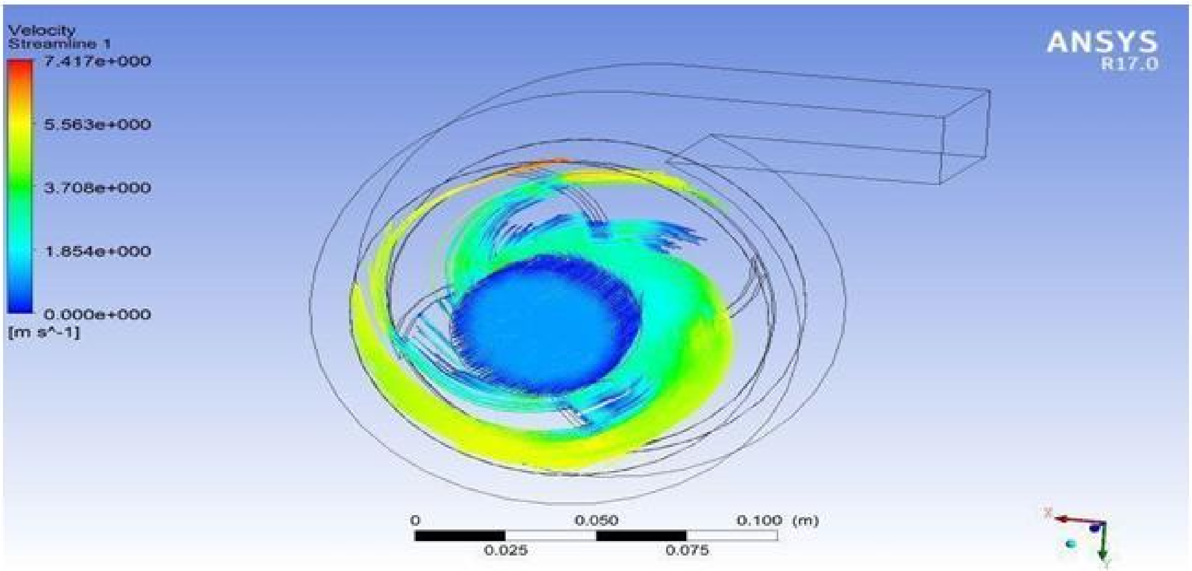

Figure 6: Streamline of 4 Blade Impeller

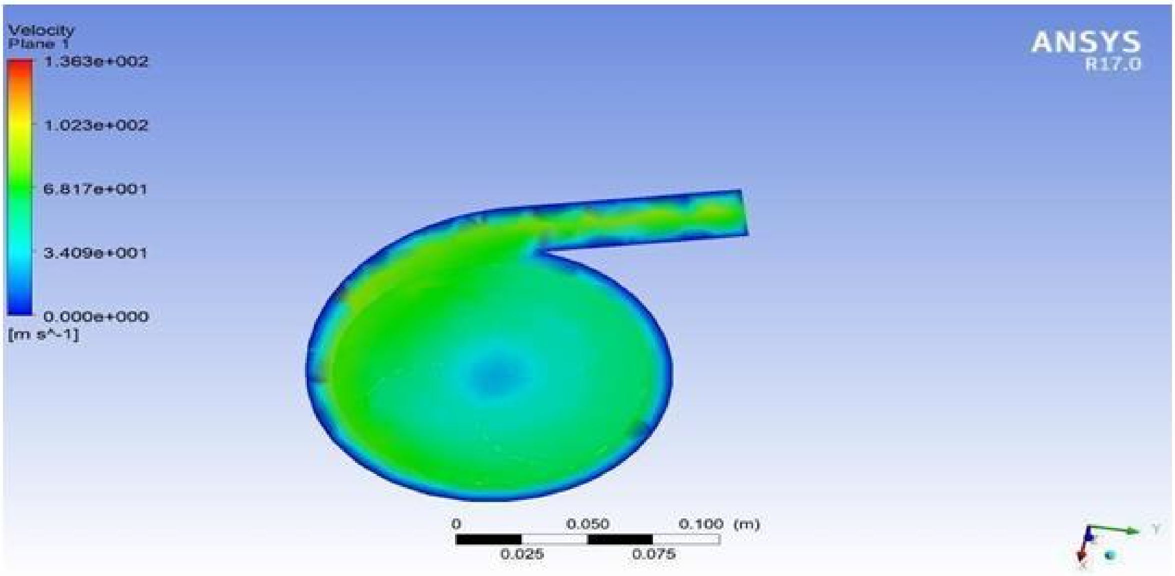

Figure 7: Velocity of Twisted 4 Blade Impeller

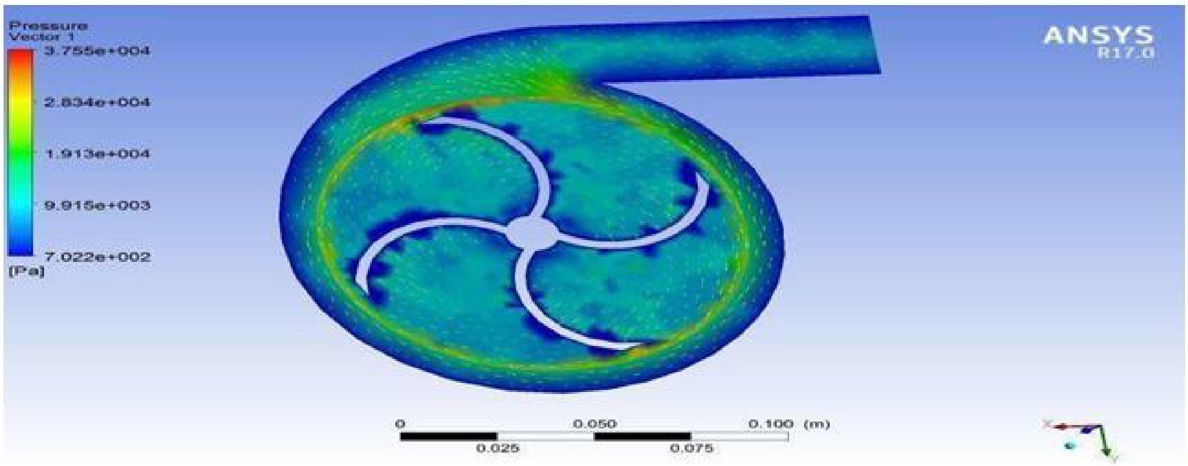

Figure 8: Turbulent Kinetic Energy of 4 Blade Impeller

In Fig 7 shows the pressure flow in the impeller casing to be tested and used to find the pressure ranges in whole body.

As can be seen from the figures, the initial design has seven blades whereas the optimized has nine. It shows the meridional velocity for the initial and optimized impeller designs. In the initial design, the velocity vectors indicate that in some regions the flow deviates from the meridional direction. In the optimized design, however, the flow nicely develops in the meridional direction.

V. Conclusion

The impeller blade was designed and analyse its performance. The various blade geometry of the impeller had been taken for analysis. The analysis was carried out in Fluent (Computational fluid dynamics). The velocity and pressure distribution in the various blade was studied. Design of an impeller was carried out by considering the Head, Discharge and the speed of the pump. Here the performance of three blades geometry had been studied by changing their specifications and design. Twisted four blades had better velocity when compared with other designs.

VI. References

[1]. E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris (2008)," Parametric study of centrifugal pump by varying the outlet blade angle", The open mechanical engineering journal, 2008, 75-83.

[2]. LIU Houlin, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai (2010)," Effects of blade number of characteristics of centrifugal pump", Chinese journal of mechanical engineering,2010.

[3]. B.Mohan, B.E. Kumar, (2014), "Structural Analysis on Impeller of an Axial Flow Compressor using Fem", International Journal of Engineering Research & Technology ISSN: 2278-0181 Vol. 3 Issue 10.

[4]. A Review Paper on Improvement of Impeller Design a Centrifugal Pump using FEM and CFD

[5]. S.Rajendran and Dr.K.Purushothaman, (2012), "Analysis of a centrifugal pump impeller using ANSYS-CFX", International Journal of Engineering Research & Technology, Vol 1 Issue 3.

[6]. A Syam Prasad, BVVV Lakshmipathi Rao, A Babji, Dr.P Kumar Babu, (2013), "Static and Dynamic Analysis of a Centrifugal Pump Impeller", International Journal of Scientific & Engineering Research, Volume 4, Issue 10.

[7]. Neelambika, Virbhadra (2014) "CFD analysis of mixed flow impeller", International journal of research engineering and technology.

[8]. Sambhrant Srivastavaa, Apurba Kumar Roy and Kaushik Kumar, (2014) "Design of a mixed flow pump impeller and its validation using FEM analysis", Science Direct, Procedia Technology 14, PN 181 – 187.

ISSN : 2057-5688

[9]. Santosh Shukla, Apurba Kumar Roy and Kaushik Kumar, (2015) "Material Selection for blades of Mixed Flow Pump Impeller Using ANSYS", Science Direct, Materials Today: Proceedings 2, 2022 – 2029.