iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 9 **Issue:** VII **Month of publication:** July 2021

DOI: https://doi.org/10.22214/ijraset.2021.36876

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

Design and Analysis of the Multi Blade Angle Centrifugal Pump Impeller by Using CFX Tool

M.U. Ravindranath[1], Devu Balaji[2], Angara Vijay Krishna[3], Mohammed Noorulla[4], Bonam Bharath Kumar[5]
[1]Assistant Professor, Dept. of Mechanical Engineering, DMS SVH College of engineering, Machilipatnam, India.
[2, 3, 4, 5]B. tech student, Dept. of Mechanical Engineering, DMS SVH College of engineering, Machilipatnam, India.

Abstract: 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. A design of centrifugal pump is carried out and analyzed to get the best performance point. The design and performance analysis of centrifugal pump are chosen because it is the most useful mechanical rotodynamic machine in fluid works which widely used in domestic, irrigation, industry, large plants and river water pumping system. These specifications have been varied toper form a comparative study of these pump impellers. The impeller was modeled 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 modeled and the flow analysis is carried out using FLUENT software. Thus the valid results regarding the velocity distribution and pressure distributions were predicted for different blade angles.
Keywords: Multiblade, Centrifugal Pump, CFX Tool, Velocity Differences, Blade Angles.

I. INTRODUCTION

Impeller is used in a wide range of industrial and residential applications. Pumping equipment is extremely diverse, varying in type, size, and materials of construction. There have been significant new developments in the area of pumping equipment. They are used to transfer liquids from low-pressure to high pressure in this system, the liquid would move in the opposite direction because of the pressure difference. Centrifugal impeller pumps are widely used for irrigation, water supply plants, stream power plants, sewage, oil refineries, chemical plants, hydraulic power service, food processing factories and mines. Moreover, they are also used extensively.

In the chemical industry because of their suitability in practically any service and are mostly used in many applications such as water pumping project, domestic water raising, industrial waste water removal, raising water from tube wells to the fields. A centrifugal pump delivers useful energy to the fluid on pumpage largely through velocity changes that occur as this fluid flows through the impeller and the associated fixed passage ways of the pump. It is converting of mechanical energy to hydraulic energy of the handling fluid to get it to a required place or height by the centrifugal force of the impeller blade. The input power of centrifugal pump is the mechanical energy and such as electrical motor of the drive shaft driven by the prime mover or small engine. The output energy is hydraulic energy of the fluid being raised or carried. In a centrifugal pump, the liquid is forced by atmospheric or other pressure into a set of rotating vanes. A centrifugal pump consists of a set of rotation vanes enclosed within a housing or casing that is used to impart energy to a fluid through centrifugal force. A pump transfer mechanical energy from some external source to the liquid flowing through it and losses occur in any energy conversion process. The energy transferred is predicted by the Euler Equation. The energy transfer quantities are losses between fluid power and mechanical power of the impeller or runner. Thus, centrifugal pump may be taken losses of energy. The kinds of loss of centrifugal pumps can be differentiated in internal losses and external or mechanical losses. The internal loss is hydraulic losses or blade losses by friction, variations of the effective area or changes of direction losses of quantity at the sealing places between the impeller and housing at the rotary shaft seals.

II. LITERATURE REVIEW

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 behavior and to predict the regions of separation with a high degree of accuracy. Thus CFD is an important tool for pump designers.

Simulate flow pattern through a water pump and compared the difference between among these methods in predicting the pumps performance. Goto 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.Zhou Weidong, Ng and his colleague Zhou and Ng and their 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, Pots, and Sun and Tsukamoto studied pump off-design performance using the commercial software CFX-TASC flow, FLUENT, and STARCD, respectively. 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.

III. MATERIALS

Selecting the right pump type and sizing it correctly are critical components to the success of any pump application. Equally important is selecting construction materials. The initial cost of these materials is normally the first consideration. Operational costs, replacement costs and longevity of service and repair costs will, however, determine the actual cost of the pump during its lifetime. Standard pump part materials (such as cast irons, bronzes and low-carbon steels) are typically the least expensive first cost -- and the most readily available for replacement. However, these materials can become more expensive if they cause premature failure and unexpected service and replacement.

Throughout the years, I have consulted on several wastewater lift station applications where the choice of low-cost cast iron for the pump impeller, even when coated, could not withstand the abrasives in the pumpage and/or cavitation, which were often exacerbated by low-flow suction recirculation. Factors that must be considered in selecting materials for wetted pump parts are, for example, user's experience, expected pump life, intermittent or continuous duty, pumping of hazardous or toxic liquids, condition of the liquid, pump suction energy level, and conditions of service (especially suction conditions).

A. Abrasive Wear

Abrasive wear is the mechanical removal of metal from the cutting or abrading action of solids carried in suspension in the pumped liquid. The rate of wear for any material is dependent upon the following characteristics of the suspended solids:

1) Solid concentration
2) Solids size and mass
3) Solids shape (spherical, angular or sharp fractured surfaces)
4) Solids hardness

Relative velocity between solids and metal surface. The rate of wear is also dependent upon the materials selected for the rotation and stationary components of a centrifugal pump. Although metal hardness is not the sole criterion of resistance to abrasive wear, hardness does provide a convenient index in selecting ductile materials usually available for centrifugal pumps. Such an index is shown in Figure 1, where the abrasive wear-resistance ratio is shown as a function of Brinell hardness for various materials. It should be noted that a brittle material, such as cast iron, exhibits a much lower ratio than either the steels or bronzes of the same hardness. The following tabulation can also be used as a guide in material selection, listed in order of increasing abrasive-wear resistance:

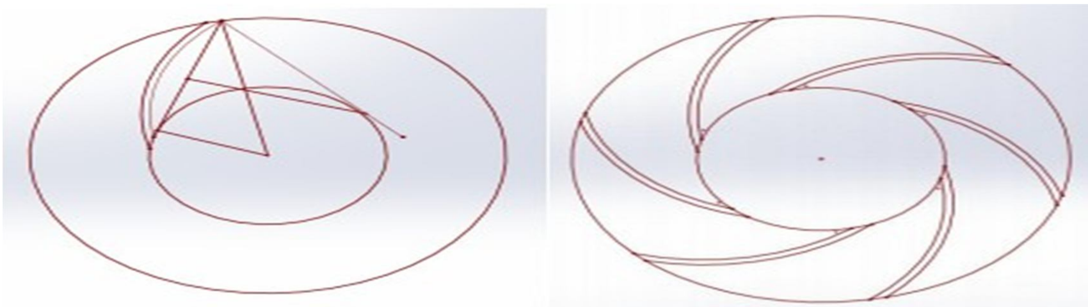
a) Cast iron
b) Bronze
c) Manganese bronze
d) Nickel-aluminum bronze
e) Cast steel
f) 300-series stainless steel
g) 400-series stainless steel

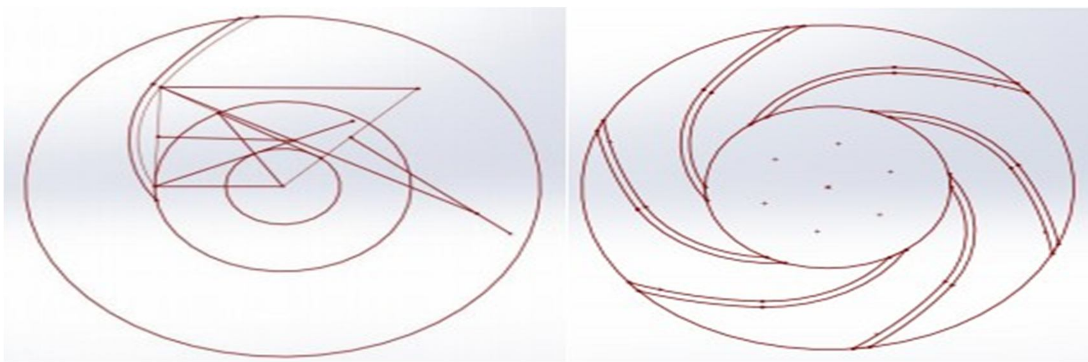
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

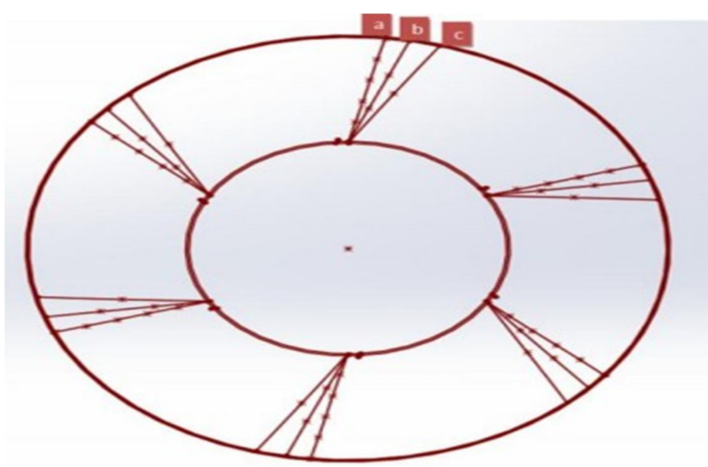
IV. DESIGN

A. The Impeller

Having defined the necessary geometric parameters, the impeller is ready to be drawn. The profiles for leading and trailing edges may be treated after getting the main blade profile, there are three basic ways to draw the plan view of an impeller. In this study, all three methods are used to generate the blade profiles. By doing so, the designer had the chance to decide which one is the most appropriate plan view blade profiles by using three different drawing methods are:-







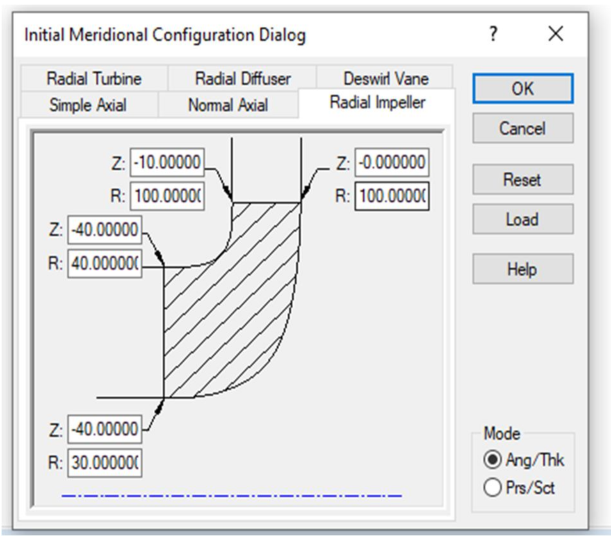
B. Computational Fluid Dynamics

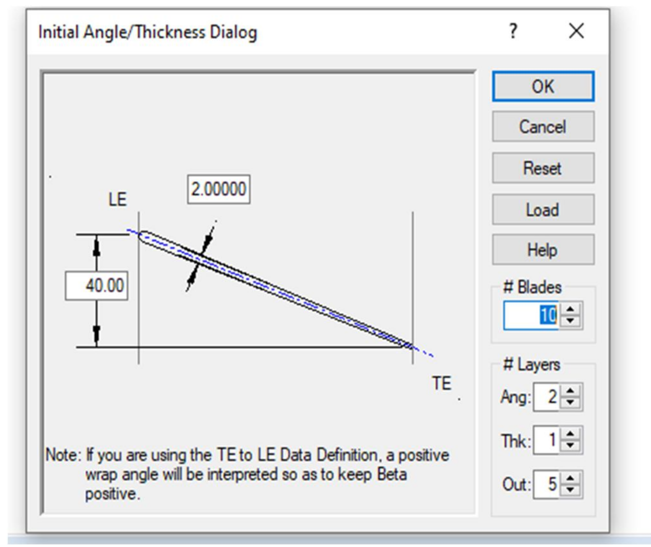
Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is typically performed using experimental apparatus such as wind tunnels. In addition, previously performed analytical or empirical analysis of a particular problem can be used for comparison. A final validation is often performed using full-scale testing, such as flight tests.

CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, weather simulation, natural science and environmental engineering, industrial system design and engine and combustion analysis.

CFD can be seen as a group of computational methodologies (discussed below) used to solve equations governing fluid flow. In the application of CFD, a critical step is to decide which set of physical assumptions and related equations need to be used for the problem at hand [40]. To illustrate this step, the following summarizes the physical assumptions/simplifications taken in equations of a flow that is single-phase (see multiphase flow and two-phase flow), single-species (i.e., it consists of one chemical species), non-reacting, compressible (unless said otherwise). Thermal radiation is neglected, and body forces due to gravity considered (unless said otherwise). In addition, for this type of flow, the next discussion highlights the hierarchy of flow equations solved with CFD.

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements: (i) a pre-processor, (ii) a solver and (iii) a post-processor. We briefly examine the function of each of these elements within the context of a CFD code.





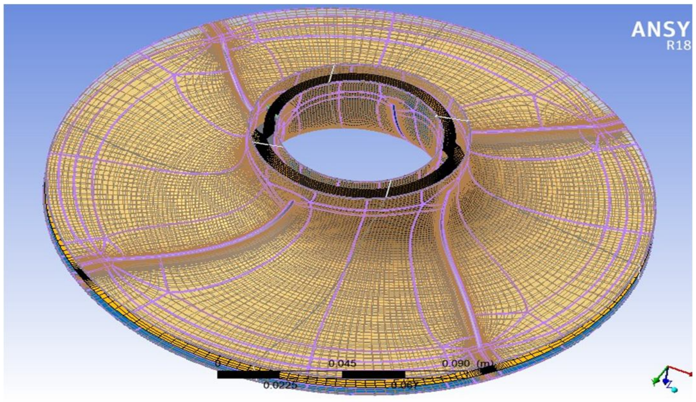
V. BLADE DESIGN

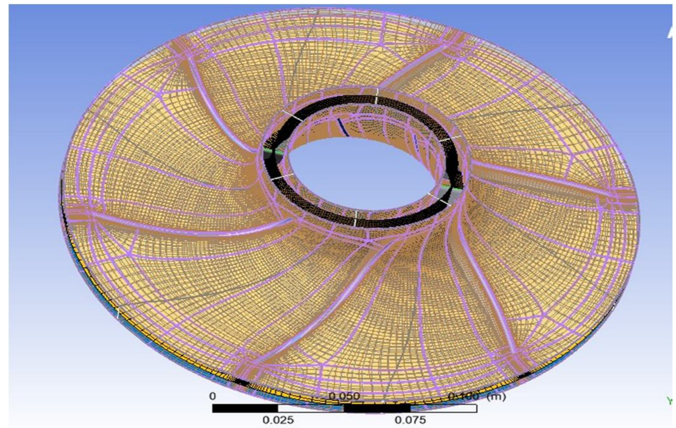
A. Pre-Processor

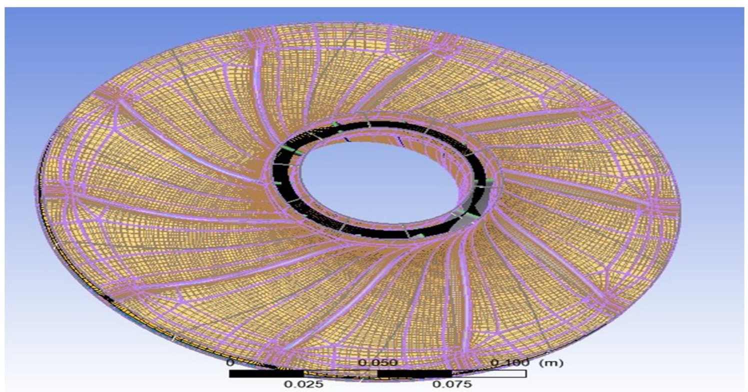
Pre-processing consists of the input of a flow problem to a CFD program by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activities at the pre-processing stage involve:

1) Definition of the geometry of the region of interest: the computational domain
2) Grid generation – the sub-division of the domain into a number of smaller, non- overlapping sub-domains: a grid (or mesh) of cells (or control volumes or elements)
3) Selection of the physical and chemical phenomena that need to be modeled.

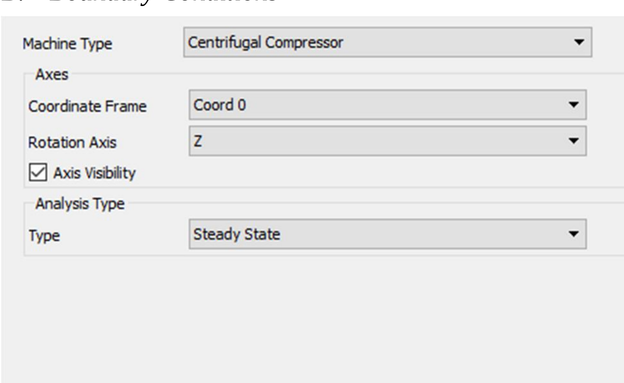
The solution to a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of a CFD solution is governed by the number of cells in the grid. In general, the larger the number of cells, the better the solution accuracy. Both the accuracy of a solution and its cost in terms of necessary computer hardware and calculation time are dependent on the fineness of the grid. Optimal meshes are often non-uniform: finer in areas where large variations occur from point to point and coarser in regions with relatively little change. Efforts are under way to develop CFD codes with a (self-)adaptive meshing capability. Ultimately such programs will automatically refine the grid in areas of rapid variations. A substantial amount of basic development work still needs to be done before these techniques are robust enough to be incorporated into commercial CFD codes. At present it is still up to the skills of the CFD user to design a grid that is a suitable compromise between desired accuracy and solution cost.

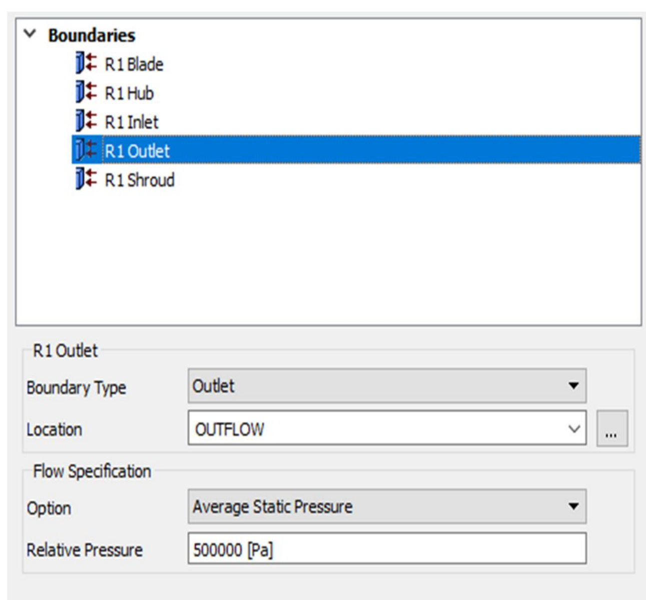


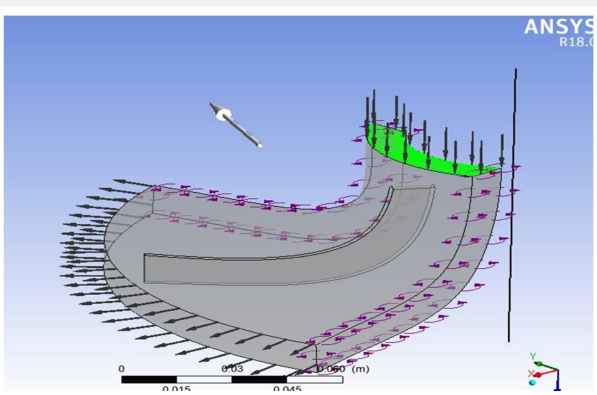




B. Boundary Conditions



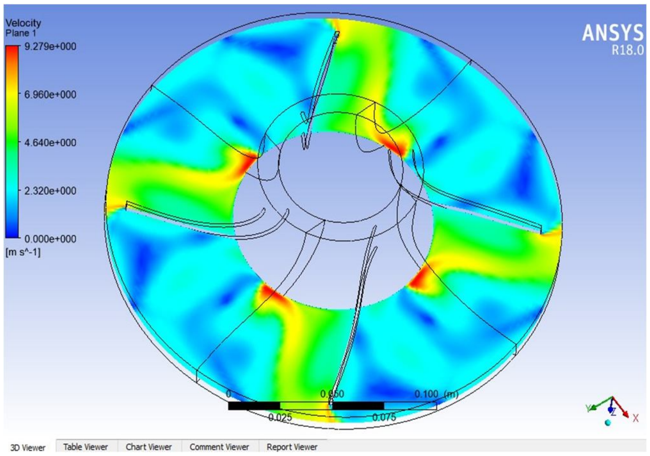


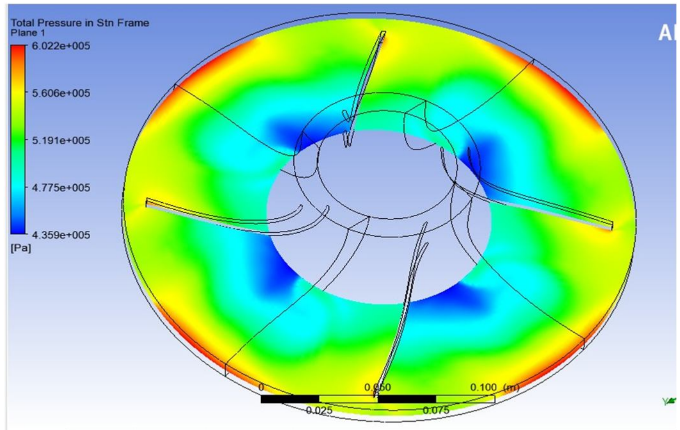


International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

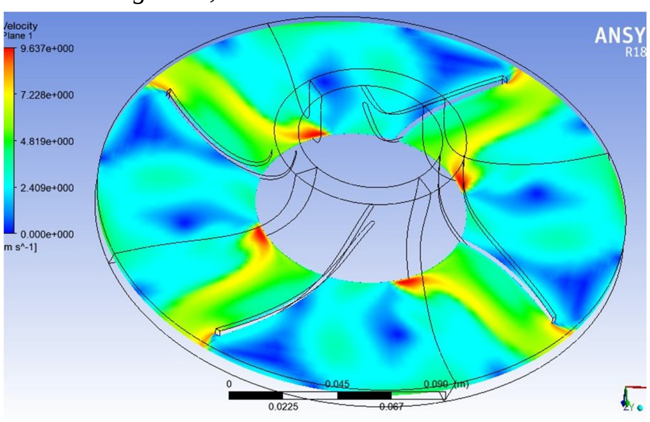
VI. RESULTS

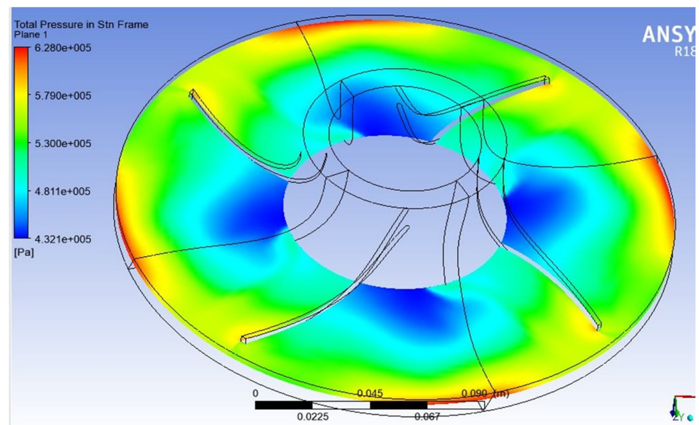
A. *Blades Angle : 23^0 , Blade NO : 4*



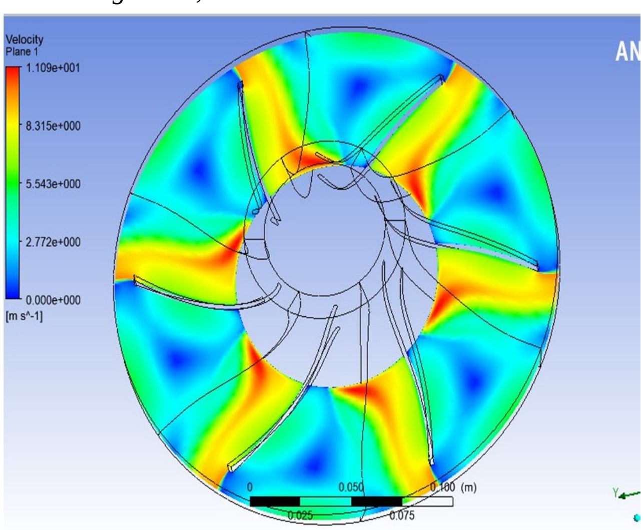


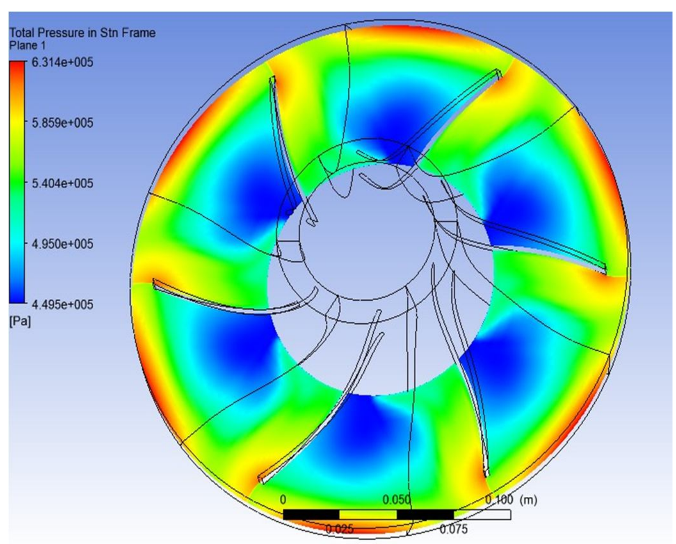
B. *Blades Angle:19^0, Blade NO: 4*



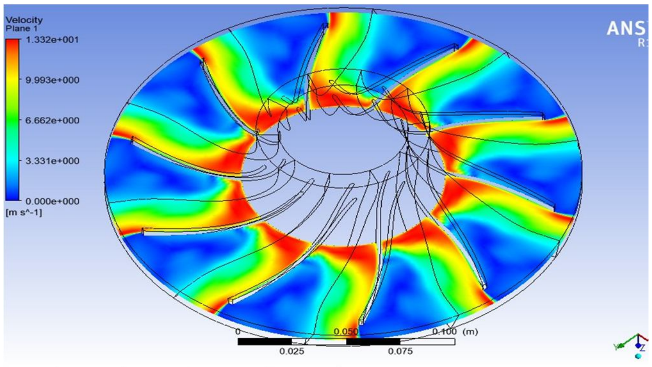


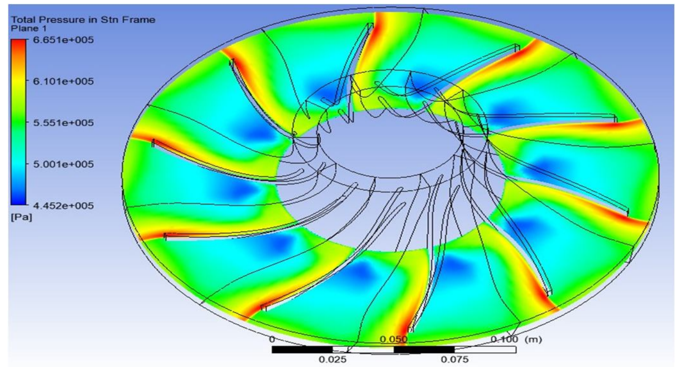
C. *Blades Angle: 19^0, Blade No: 6*



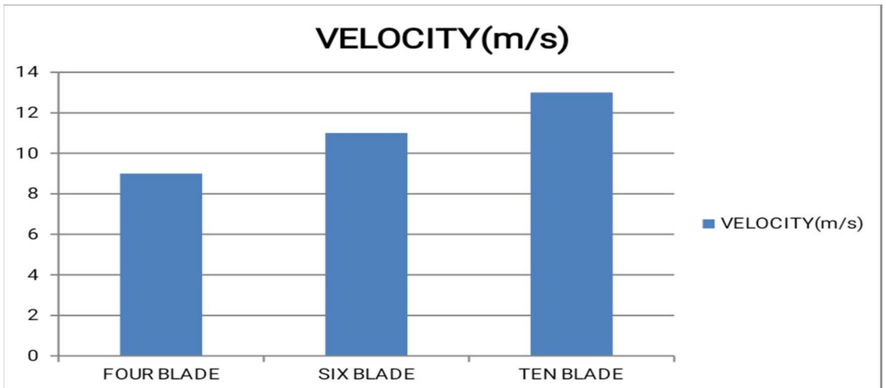


D. Blades Angle: 19^0, Blade No: 10





E. Blade Angle 19^0



VII. CONCLUSION

The impeller was designed and modeled to analyze its performance. The blade-to-blade plane 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 blade to- blade plane was studied. Design of an impeller was carried out by considering the Head, Discharge and the speed of the pump. Here the performance of pumps had been studied by changing their specifications. The results obtained from CFD are used to compare their performance. It was found that the predicted results for 19^0 impeller is better than those of the pump with 23^0 impeller, which suggested that the efficiency of that will be higher than the other pumps.

From this it was found that when the inflow rate is 20% of the design flow rate, no deviation in flow pattern but if the flow rate drops or increase by (30-40%) then the flow pattern deviates slightly and there is a chance for the reverse flow to take place. Here there is no problem related with recirculation and the flows through the pumps are smooth.

REFERENCES

[1] Dynamic characteristics and leakage performance of liquid annular seals in centrifugal pumps Eskild Storteig Department of Marine Engineering Norwegian University of Science and Technology NTNU Submitted 99.

[2] Multiphase Pumping: Achievements and Perspectives J. Falcimaigne1, J. Brac1, Y. Charron1, P. Pagnier1 and R. Vilagines1Oil & Gas Science and Technology – Rev. IFP, Vol. 57 (2002), No. 1, pp. 99-107Copyright © 2002.

[3] Editions Technip Proceedings of the Ninth NRC/ASME Symposium on Valves, Pumps and In service Testing Held at L'Enfant Plaza Hotel Washington, DC July 17-19, 2006.

[4] Optimum values of design variables versus specific speed for centrifugal pumps, H.W Oh and M. K Chung,Taejon, Korea A05098 IMechE 1999.

[5] Jacek wojtusik, 2003, "Application of multimodels to modeling of centrifugal pumps behaviour", AI-Meth, Pp 339-344.

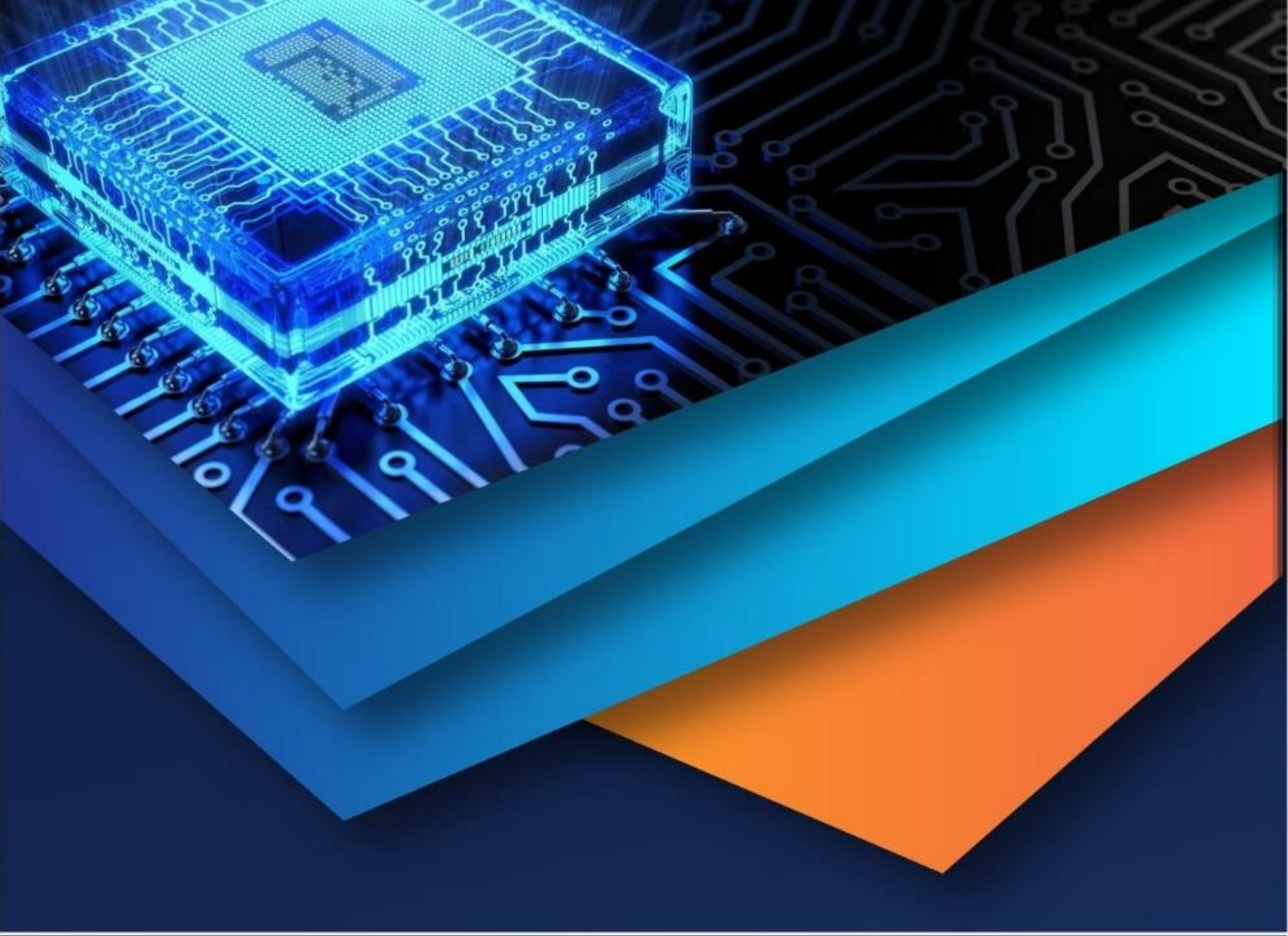
[6] Shukla.S.N and Kshirsagar.J.T. 2003, "Numerical experiments on a centrifugal pump", Indian pumps. Pp 21-30.

[7] Church and Lal, 1973, "Centrifugal pumps and blowers" Metropolitan book Co. Pvt. Limited Lazarkiewiz "Design of Pumps and impellers". Herbert addision, "Centrifugal and other rotadynamic pumps".

[8] 3D flow simulation in an axial flow pump using computational fluid dynamic method,B.C Bhayal , R & D division , INDIAN pumps 2002.

[9] Effect of Reynolds-number and surface roughness on the efficiency of centrifugal pumps J.F. Gülich, Sulzer Pumps, Winterthur and Ecole Polytechnique, Lausanne, Switzerland ASME Journal of Fluids Engineering 125 (2003) 4, 670-679.

[10] Yagnesh Sharma.N, and KaranthK.V., 2003, "CFD analysis of a turbulent steady flow through a volute casing of a centrifugal pump for static pressure distribution", ISME.



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY