International Journal of Engineering Research-Online
*A Peer Reviewed International Journal*
Articles available online http://www.ijoer.in; editorijoer@gmail.com

Vol.5., Issue.2, 2017
March-April

RESEARCH ARTICLE

ISSN: 2321-7758

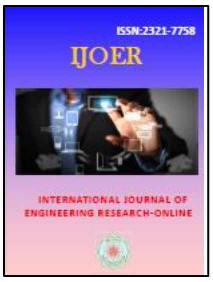
# EFFECT OF FLUID PRESSURE ON IMPELLER

**SYAMBABU NUTALAPATI[1], Dr. D. AZAD[2], Dr. G. SWAMI NAIDU[3]**

[1]Research Scholar, (Ph.D), M.E (Machine Design),JNTUK, India.

[2]Professor, AitamTekali, Srikakulam District, India

[3]Professor and Vice-Principal (Administration), Department of Metallurgical Engineering, JNTUK-UCEV, India.




**ABSTRACT**

An impeller is a rotating component of a centrifugal pump, usually made of iron, steel, bronze, brass, aluminum or plastic, which transfers energy from the motor that drives the pump to the fluid being pumped by accelerating the fluid outwards from the center of rotation. The purpose of this paper is to identify /observe and determine the pattern of pressure distribution by using CFD simulation program after the 3D design and modeling of the pump is made using CATIAV5 basically , this paper revolves around the idea of investigating the effect and distribution of pressure in pump . The standard k-ε turbulence model was chosen for turbulence model.

**Key points**: Impeller, CATIA-V5, CFD, Pressure distribution, K-epsilon (k-ε) turbulence



## 1. INTRODUCTION

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. The velocity achieved by the impeller transfers into pressure when the outward movement of the fluid is confined by the pump casing. 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.

K-epsilon (k-ε) turbulence: K-epsilon (k-ε) turbulence model is the most common model used in Computational Fluid Dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. It is a two equation model which gives a general description of turbulence by means of two transport equations (PDEs). The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows.

MODEL DESIGN: The impeller model has been modeled by using CATIA V5.

**2. CATIA- V5:** CATIA-V5 is the industry's de facto standard 3D mechanical design suit. It is the world's leading CAD/CAM/CAE software, gives a broad range of integrated solutions to cover all aspects of product design and manufacturing. Much of its success can be attributed to its technology which spurs its customer's to more quickly and consistently innovate new robust, parametric, feature based model because that CATIA-V5 is unmatched in this field in all processes, in all countries, in all kind of companies along the supply chains. CATIA–v5 is also the perfect solution for the manufacturing enterprise, with associative applications, robust responsiveness and web connectivity that make it the ideal flexible engineering solution to accelerate innovations.

Vol.5., Issue.2, 2017
March-April

Catia-v5 provides easy to use solution tailored to the needs of small medium size enterprises as well as large industrial corporations in all industries, consumer goods, fabrications and assembly. Electrical and electronics goods, automotive, aerospace, shipbuilding and plant design. It is user friendly solid and surface modelling can be done easily.

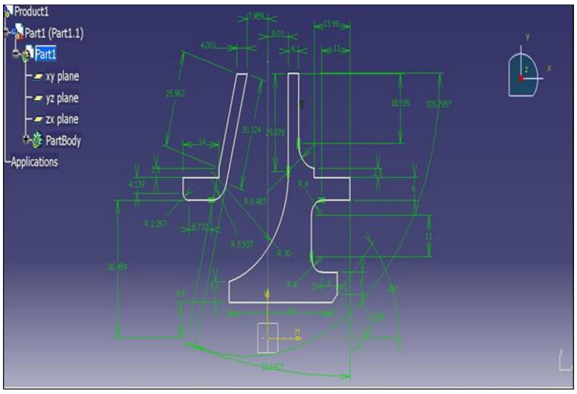



Fig 2.1: CATIA Impeller Plate Design

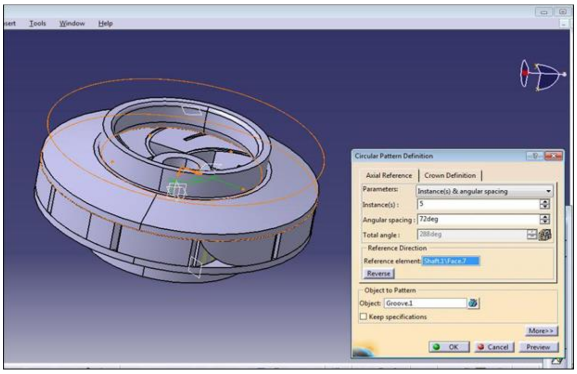


Fig 2.2: CATIA Impeller Design

**3. ANSYS WORKBENCH:**ANSYS workbench is a new generation solution from ANSYS that provides powerful methods for interacting with the ANSYS solver functionality. This environment provides a unique integration with CAD systems, and your design process, enabling the best CAE results.

Fluid flow encountered in everyday life include

- Meteorological phenomena (rain, wind, hurricanes, floods, fires)
- Environmental hazards (air pollution, transport of contaminants)
- Heating, ventilation and air conditioning of buildings, cars etc.
- Combustion in automobile engines and other propulsion system
- Interaction of various objects with the surrounding air/water
- Complex flows in furnaces, heat exchangers, chemical reactors etc.
- Processes in human body (blood flow, breathing, drinking.....)
- Computational fluid dynamics (CFD): It provides a quantitative (and sometimes even quantitative) prediction of fluid flows by means of
- Mathematical modelling (partial differential equations)
- Numerical methods (discretization and solution techniques)
- Software tools (solver, pre- and post-processing utilities)
- CFD enables scientists and engineers to perform „numerical experiments‟ (i.e. computer simulations) in a „virtual flow laboratory‟
- Numerical simulations of fluid flow (will) enable
- Architects to design comfortable and safe living environments
- Designers of vehicles to improve the aerodynamic characteristics
- Chemical engineers to devise optimal oil recovery strategies
- Surgeons to cure arterial diseases (computational hemodynamic)
- Meteorologists to forecast the weather and warm natural disasters
- Safety experts to reduce health risks from radiations and other hazards
- Military organization to develop weapons and estimate the damage
- CFD practitioner to make big bucks by selling colourful pictures.

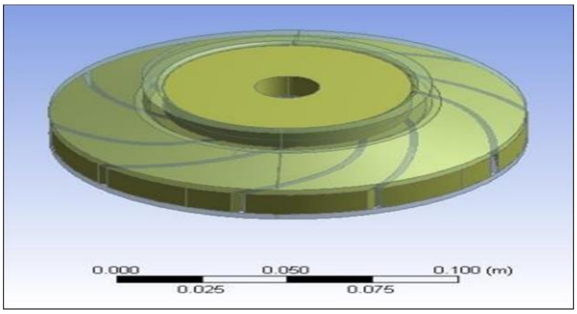



Fig 3.1: Display of Importing Geometry Model

Select the model and mesh. Give rotation option to have a motion in impeller by giving angular velocity.

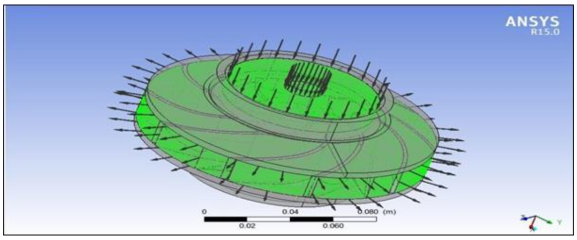



Fig 3.2: Mesh and Its Motion

Select the motion model of fluid in turbulence flow, wall function as k-Epsion.

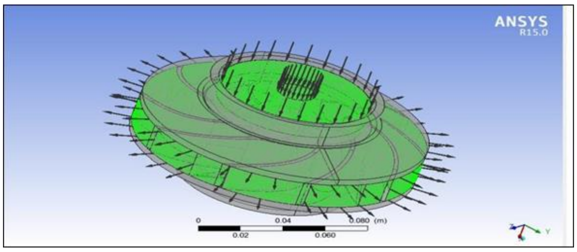



Fig 3.3: Fluid Motion Model

Select the blade and give rotating motion to blades by selecting the rotation axis.

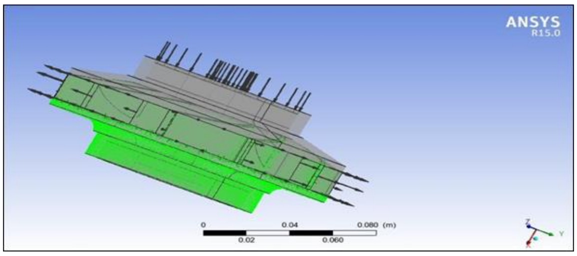



Fig 3.4: Blades Motion

Select flow analysis1 analysis type default insert boundaries

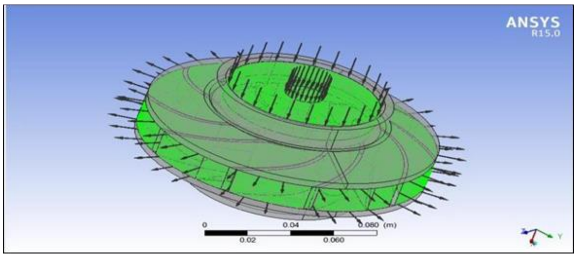



Fig 3.5: Creating Boundary

For giving inlet flow mass & momentum flow rate as 8.78[kg s^-1], flow direction as normal to boundary condition and turbulence as medium is given to model.

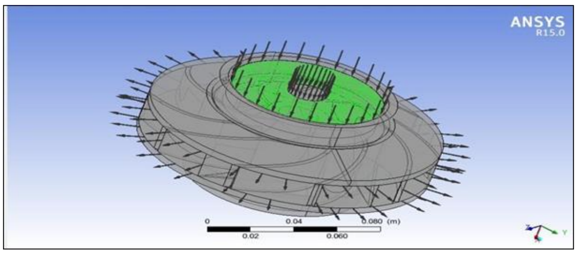



Fig 3.6: Flow Inlet Conditions

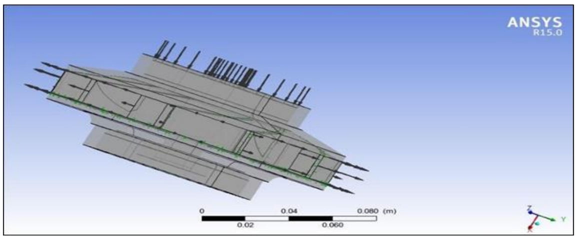



Fig 3.7: Solution Controls

## 4. RESULTS

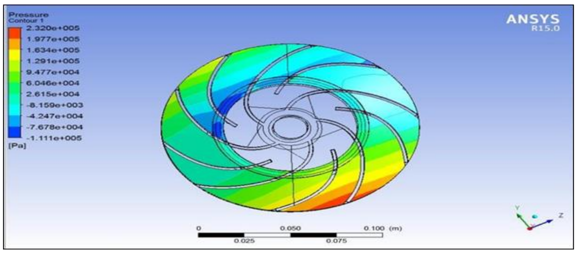



Fig 4.1: Pressure on Stationary Plate

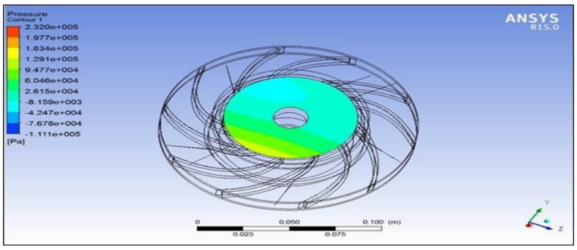



Fig 4.2: Pressure at Inlet

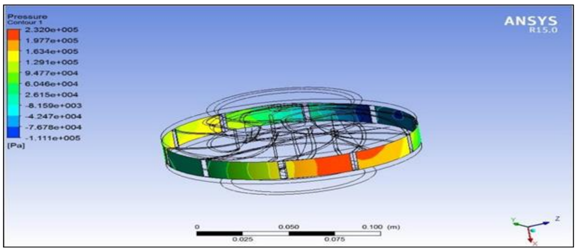



Fig 4.3: Pressure at Outlet

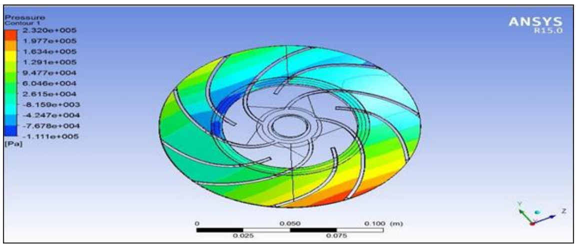
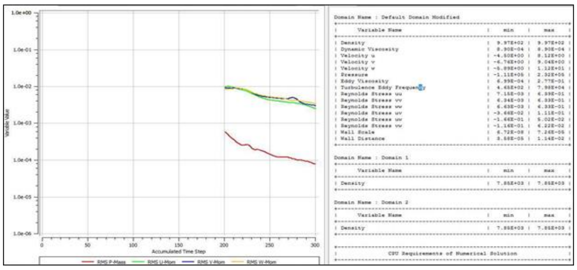


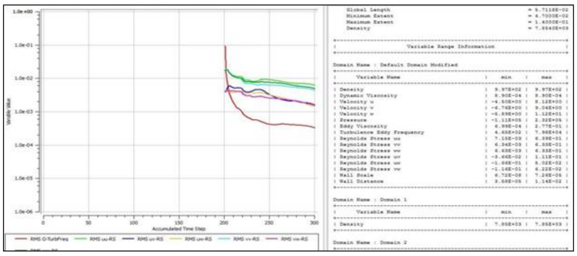

Fig 4.4: Total Fluid Pressure

GRAPH OF MASS AND MOMENTUM:



Graph 4.1: Graph of Mass and Momentum

GRAPH OF TURBULENCE



Graph 4.2: Graph of Turbulence

## 5. CONCLUSION

From the simulation results it is observed that the pressure gradually increases from impeller for inlet to outlet. The static pressure on pressure side is evidently larger than that on suction side at the same impeller radius. From the result the maximum pressure developed in a fluid is 232014 Pa.

## REFERENCE

[1]. P.Gurupranesh, R.C.Radha, N.Karthikeyan, "CFD Analysis of centrifugal pump impeller for performance Enhancement", IOSR Journal of Mechanical and Civil Engineering (IOSR- JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334XPP 33-41

[2]. Raghavendra S Muttalli,Shweta Agrawal, Harshla Warudkar "CFD Simulation of Centrifugal Pump Impeller Using ANSYS-CFX" International Journal of Innovative Research in Science, Engineering and Technology (An ISO 3297: 2007 Certified Organization) Vol.3, Issue8,August2014

[3]. "Design and CFD Analysis of Centrifugal PumpInternational" Journal of Engineering Research and General Science Volume 3, Issue 3, May-June, 2015

[4]. Ragoth Singh, M. Nataraj "Design and analysis of pump impeller using SWFSR". World Journal of Modelling and Simulation Vol. 10 (2014) No. 2, pp. 152-160

[5]. Krishna Kumar Yadav, Karun Mendiratta, V K Gahlot, "Optimization Of The Design Of R Adial Flow Pump Impeller Though Cfd Analysis" Ijret: International Journal Of Research In Engineering And Technology.

[6]. Perez J., Chiva S., Segala W*.,Morales R., Negrao C., Julia E., Hernandez L. "Performance Analysis Of FlOw In A Impeller-Diffuser Centrifugal Pumps Using Cfd": Simulation And Experimental Data Comparisons, European Conference on Computational Fluid Dynamics ECCOMAS CFD 2010.

[7]. S. C. Chaudhari1, C. O. Yadav2 & A. B. Damor, "A Comparative Study Of Mix Flow Pump Impeller Cfd Analysis And Experimental Data Of Submersible Pump" International Journal Of Scientific Engineering And Applied Science– Volume -1, Issue -5, August 2015

[8]. "Analysis And Investigation Of Centrifugal Pump Impellers Using Cfd" Iracst – Engineering Science And Technology: An International Journal (Estij), Issn: 2250-3498 Vol.4, No.4, August 2014