International Journal of Engineering and Manufacturing Science.
ISSN 2249-3115 Volume 7, Number 2 (2017), pp. 309-318

http://www.ripublication.com

# Investigation Into Flow Field of Centrifugal Pump Impeller

**B.Subbarao [1], Dr. E.Ramjee [2], Dr. M. Devaiah[3], Dr. T. Siva Prasad[4]**

[1, 3, 4] *Department of Mechanical Engineering, Geethanjali College of Engineering and Technology, Telangana State, India*

[2]*Professor, Department of Mechanical Engineering, JNTU Hyderabad, Telangana, India.*

**Abstract**

This study deals with the design and performance analysis of centrifugal pump impeller. In this thesis, centrifugal pump is analyzed by using a single-stage end suction centrifugal pump. Two main components of a centrifugal pump are the impeller and the casing. The impeller is a rotating component and the casing is a stationary component. In centrifugal pump, water exits radially, while water enters axially through the impeller eyes. The pump casing is to guide the liquid to the impeller, converts the high velocity kinetic energy of the flow from the impeller discharge into pressure. A mean of centrifugal pump impeller is passed out and analyzed to get the best performance point. The design and performance analysis of centrifugal pump impeller are chosen because pump is the most useful mechanical Rotodynamic machine in fluid works which is widely used in domestic, irrigation, industry, large plants and river water pumping system. In this study, 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. Corresponding pressure contours and velocity contours are plotted at design flow rate (20 LPS), part flow rate (16 LPS) and excess flow rate (25 LPS). The simulation values are compared with analytical solution

***Key Words*:** CFD and ANSYS fluent 14.5.

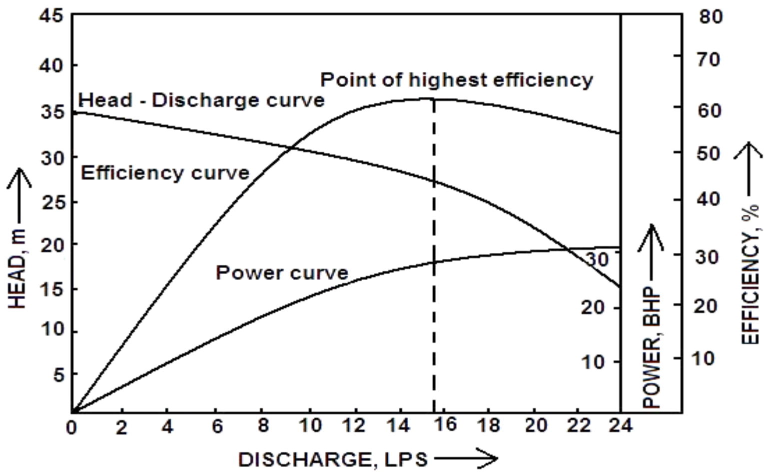
## 1. INTRODUCTION

### 1.1 Geometry Modeling & Simulation:

In design of a pump 3 variables are to be considered. They're:

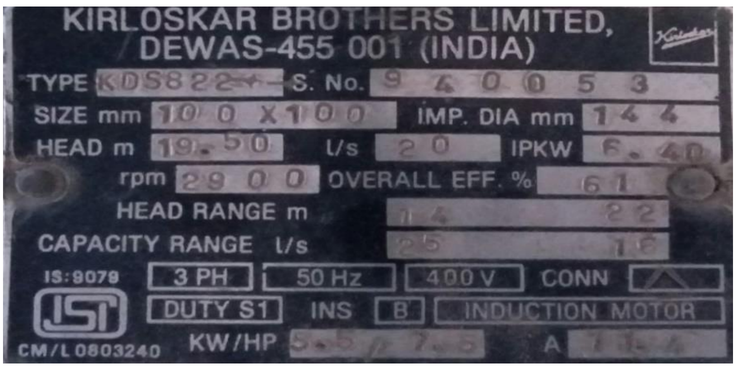
1. Head developed by the pump
2. Discharge/ Flow at the particular head
3. Speed of the Pump

Considering the above three variables a pump will be designed by plotting a performance curve and choosing the best efficiency point (BEP) as shown in the figure, In the figure dotted line indicates the BEP of a centrifugal pump which is considered for plotting the performance curves.




**Performance chart of centrifugal pump**

### 1.2 Specifications



### 1.3 Pressure Calculations

**Case – 1**

Mass flow rate 20 LPS = 20 kg/s = 72m3/hr

Head developed = 19.50 m

Delivery head (hd) = 19.50m = 1.9118 bar

Suction Head (hs) = 15.257m = 1.495 bar

Static Head (Hs) = hs + hd

= 3.410 bar

= 34,000 Pascal

Manometric head (Hm) = hs + hd + hf + $Vd22g$

hf = 0.8m = 0.0784 bar

Vd = 2.5465 m/s

∴Hm = 3.410 + 0.0784 + 0.3305

= 3.889 bar

= 381,890 Pascal

**Case - 2**

Mass flow rate 25LPS = 25 kg/s = 90 m3/hr

Head developed = 14 m

Delivery head (hd) = 14 m = 1.3725 bar

Suction head (hs) = 15.257 m = 1.4958 bar

Static Head (Hs) = hs + hd

= 1.4958 + 1.3725

= 2.8683 bar

= 286,830 Pascal

Manometric Head (Hm) = Hs + hf + $Vd22g$

hf = 1.1m = 0.1078 bar

Vd = 3.1831 m/s

Hm = 2.8683 + 0.1078 + 0.5164

= 3.4925 bar = 349,250 Pascal

**Case - 3**

Mass flow rate 16 LPS = 16 kg/s = 57.6m3/hr

Head developed = 33 m

Delivery head (h0) = 33m = 3.2353 bar

Suction head (hs) = 15.257 m = 1.4958 bar

Static head (Hs) = hs + hd

= 1.4958 + 3.2353

= 4.7311 bar

= 473,109.99 Pascal

Manometric head (Hm) = Hs + hf + $Vd22g$

hf = 0.4487 m = 0.0439 bar

Vd = 2.0371 m/s

∴ Hm = 4.7311 + 0.0439 + 0.2115

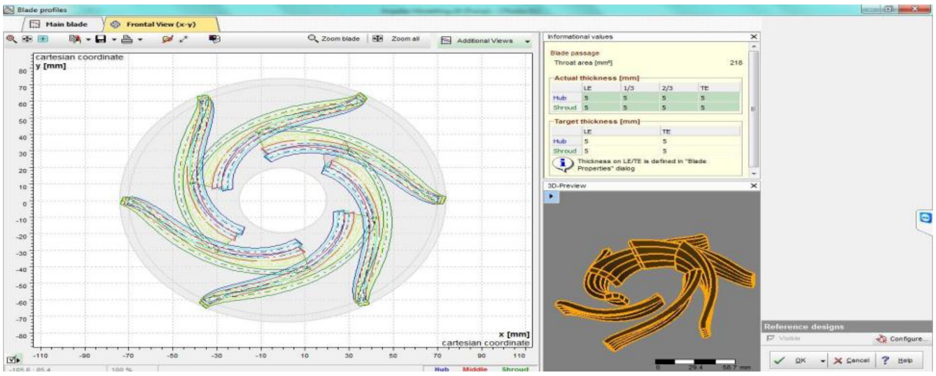
= 4.9865 bar

= 498,650 Pascal

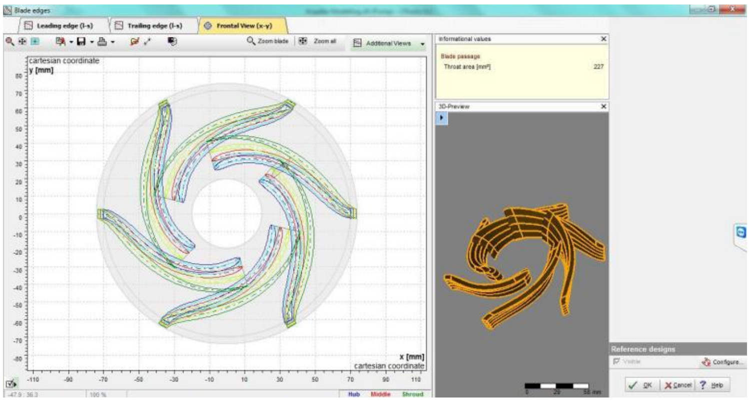
### 1.3 Analytical Results

| **S. No** | **Dimension** | **Variable Notation** | **Value** |
|---|---|---|---|
| 1 | Mass flow rate | Q | 20 kg/s |
| 2 | Head | H | 19.50 m |
| 3 | Speed | N | 2900 RPM |
| 4 | Impeller inlet diameter | D1 | 77.26 mm |
| 5 | Outlet diameter | D2 | 144 mm |
| 6 | No. of blades | Z | 6 |
| 7 | Vane/blade thickness | t | 5 mm |
| 8 | Inlet blade angle | β1 | 30.15° |
| 9 | Outlet blade angle | β2 | 43.77° |

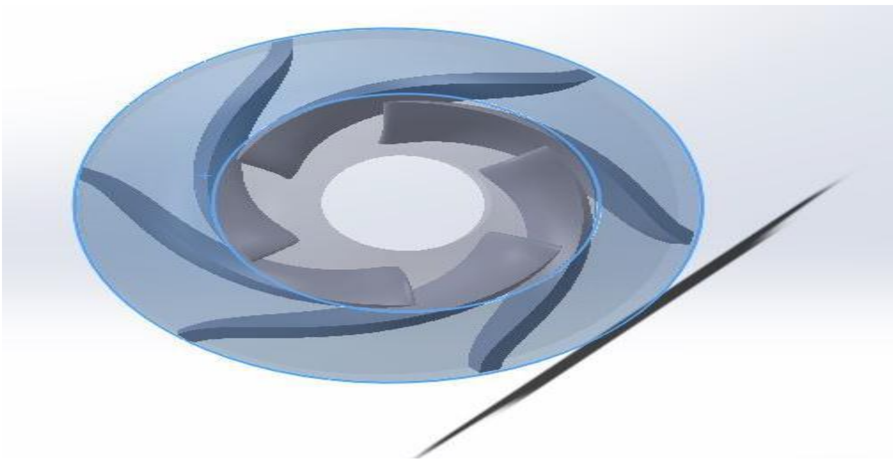
## 2. Modeling Procedure



**Blade profiles**



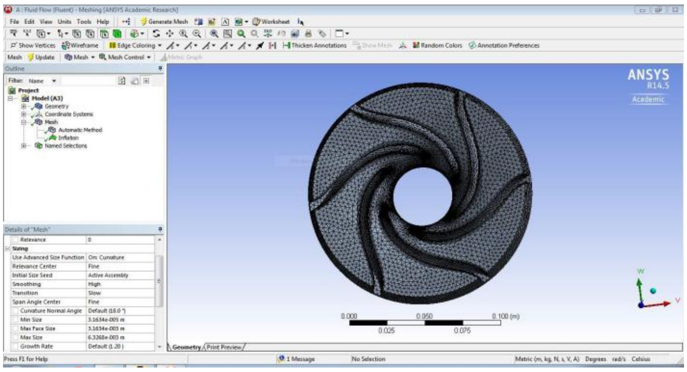
**Blade edges-Over view**



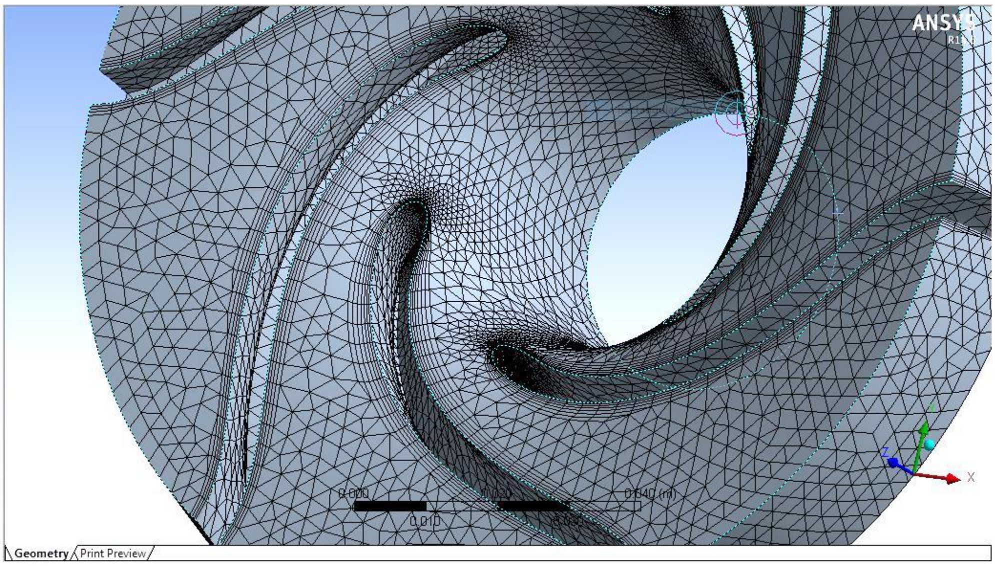
**Transparent model of impeller**

## 2.1 Geometry Analysis

## CFD Analysis



**Meshing of Impeller**



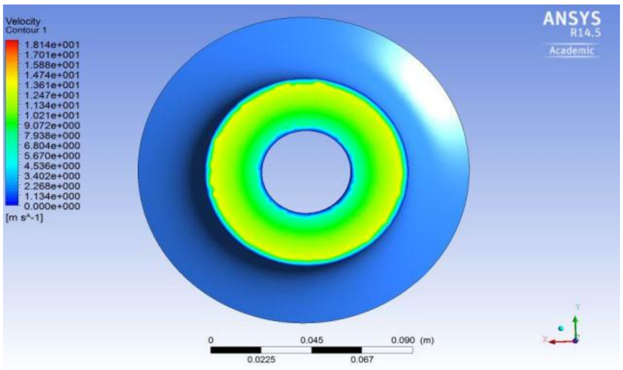
**Inflation at blade walls**

## 2.2 Physical properties of fluids used in CFD

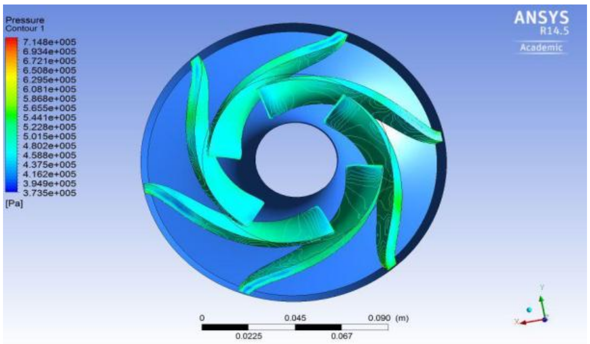
| Materials | Water | Aluminum |
|---|---|---|
| Density kg/m3 | 998.3 | 2719 |
| Specific Heat J/kg K | 4.18 | 871 |
| Thermal conductivity W/m k | - | 202.4 |
| Viscosity Kg/ms | 0.001003 | - |

## 3. RESULTS and Error Analysis

### 3.1Flowrate of 16 LPS

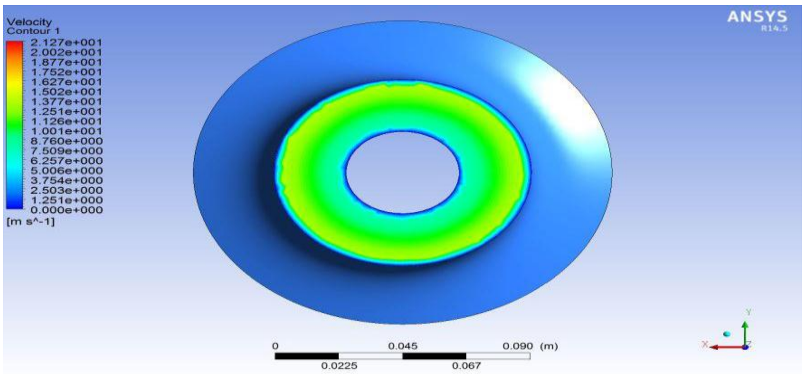



**Velocity distribution at Inlet for 16 LPS flow rate**

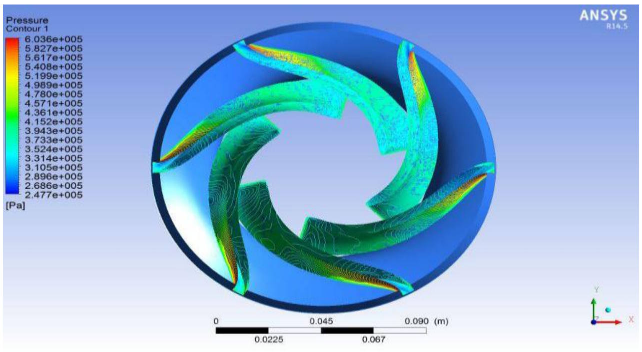



**Pressure distribution on blades for 16 LPS flow rate**

### 3.2 Flow rate of 20 LPS

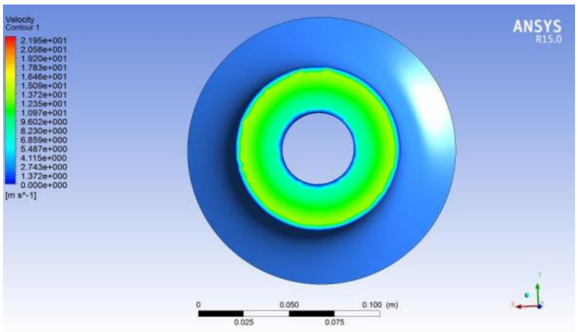



**Velocity distribution at Inlet for 20 LPS flow rate**

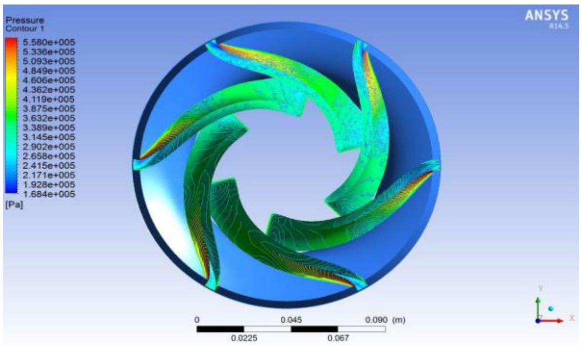



**Pressure distribution on blade for 20 LPS flow rate**

## 3.3Flowrate of 25 LPS




**Velocity distribution at Inlet for 25 LPS flow rate**




**Pressure distribution on blades for 25 LPS flow rate**

## 3.4 Error Analysis

### Analytical and Simulation results

For 16 LPS flow rate the Velocity value of simulation result is less when compared to analytical result and the error is 15.285%.

The pressure value of simulation result is more when compared to the analytical result and the error is 30.92%.

- For 20 LPS flow rate the Velocity value of simulation result is less when compared to analytical result and the error is 2.69%.

The pressure value of simulation result is more when compared to the analytical result and the error is 36.73%.

- For 25 LPS flow rate the Velocity value of simulation result is more when compared to analytical result and the error is 2.40%.

| **Flow Rate** | | **Velocity (m/s)** | | | **Pressure (Pascal)** | |
|---|---|---|---|---|---|---|
| **Analytical Result** | **Simulation Result** | **Error (%)** | | **Analytical Result** | **Simulation Result** | **Error (%)** |
| 16 LPS | 21.865 | 18.522 | 15.285 | 498650 | 721872 | 30.92 |
| 20 LPS | 21.865 | 21.270 | 2.69 | 381890 | 603600 | 36.73 |
| 25 LPS | 21.865 | 22.404 | 2.40 | 349250 | 566065 | 38.30 |

The pressure value of simulation result is more when compared to the analytical result and the error is 38.30%.

## 4. CONCLUSIONS

From analytical solution it is observed that the efficiency increases till the BEP and decreases as the flowrate increases. The head curve decreases with increase in flowrate. The output power or hydraulic power also decreases as the flowrate increases. From ANSYS simulation, the flow field distribution throughout the impeller is also observed. At the inlet the velocity and pressure distribution is uniform at observed flowrates. Similarly for the outlet velocity and pressure distribution drastic variations are observed. On blades cavitation is observed at the trailing edge. Drastic changes in pressure and velocity is observed at the outlet which is caused due to the cavitation at the tip of the blade trailing edge. The same pattern of variation in pressure and velocity is observed at designed flowrate (20 LPS), part flowrate (16LPS) and excess flowrate (25 LPS).

From the ANSYS simulation, it is observed that the simulation value of pressure rise is more when compared to the analytical value (30.9% error) and the value of velocity in the ANSYS simulation is lesser than the value obtained in analytical solution (2.4% error). This is because of the flow reversal zone which is caused due to the vortices or wakes at the tip of the blade. As there is a vortices formation, it causes drastic increase in pressure at the localized region and due to the turbulence or flow reversal the magnitude of velocity after the turbulent region decreases due to loss of energy in vortices

## REFERENCES

[1] Eric Dick, Jan Vierendeels, Sven Serbruyns and John Vande Voorde, (2001) "Performance prediction of centrifugal pumps with CFD tools". Task quarterly 5 No 4 (2001), 579–594, tq0405e7/580 26 I 2002 BOP s.c., Retrieved from http://www.bop.com.pl.

[2] Jose´ Gonza´lez, Joaquı´n Ferna´ndez, Eduardo Blanco, Carlos Santolaria(2002) "Numerical Simulation of the Dynamic Effects Due to Impeller-Volute Interaction in a Centrifugal Pump". Vol. 124, JUNE 2002 Copyright © 2002 by ASME Transactions of the ASME.

[3] Weidong Zhou, Zhimei Zhao, T. S. Lee, and S. H.Winoto (2003) "Investigation of Flow through Centrifugal Pump Impellers Using Computational Fluid Dynamics". International Journal of Rotating Machinery, 9(1): 49–61, 2003 Copyright °c 2003 Taylor & Francis 1023-621X/03 $12.00 + .00 DOI: 10.1080/10236210390147380.

[4] K M Guleren and A Pinarbasi (2004) "Numerical simulation of the stalled flow within a vaned centrifugal pump". Proc. Instn Mech. Engrs Vol. 218 Part C: J. Mechanical Engineering Science.

[5] Miguel Asuajea, Farid Bakira, SmaÏne Kouidri‡a, Robert Reya (2004) "Inverse Design Method for Centrifugal Impellers and Comparison with Numerical Simulation Tools". International Journal of Computational Fluid Dynamics, 18: 2, 101 — 110.

[6] John S. Anagnostopoulos (2006) "CFD Analysis and Design Effects in a Radial Pump Impeller".Wseas Transactions on fluid mechanics. Issue 7, Vol. 1, July 2006 ISSN: 1790-5087.56

[7] José González, Carlos Santolaria (2006) "Unsteady Flow Structure and Global Variables in a Centrifugal Pump".Journal of Fluids Engineering Copyright © 2006 by ASME September 2006, Vol. 128 / 937

[8] Si Huanga, Mohammed F. Islamb, Pengfei Liu (2006) "Numerical simulation of 3D turbulent flow through an entire stage in a multistage centrifugal pump". International Journal of Computational Fluid Dynamics, 20: 5,309 — 314.

[9] Adnan Ozturk, Kadir Aydin, Besir Sahin and Ali Pinarbasi (2009) "Effect of impeller-diffuser radial gap ratio in a centrifugal pump". Journal of Scientific & Industrial Research Vol. 68, March 2009, pp.203-213.