IOSR Journal of Engineering (IOSR JEN) **www.iosrjen.org**
ISSN (e): 2250-3021, ISSN (p): 2278-8719
PP 20-27

Experimental Analysis of Performance of Centrifugal Pump

Aashay Desai[1], Umair Siddiqui[2], Sher Afghan Khan[3], Emaad Ansari[4]
[1]*(Mechanical Engineering Department, NMIMS, India)*
[2]*(Mechanical Engineering Department, NMIMS, India)*
[3]*(Mechanical Engineering Department, International Islamic University of Malaysia)*
[4]*(Mechanical Engineering Department, M.H. SabooSiddik College of Engineering)*

Abstract: *In the world of hydraulics, the most preferred pumping devices are centrifugal pumps. This is due to their advantages like small in size, easier for maintenance, high efficiency, etc. [1]. The hydraulic machines which convert the mechanical energy into hydraulic energy are called hydraulic pumps. The hydraulic energy is in the form of pressure energy. A pump is a device that transfers fluids from one place to another by doing mechanical action. Thus according to the standard definition, Centrifugal pump is a device that converts Rotational form of energy (via motor) to energy in a moving fluid. The most essential parts that are included in centrifugal pump are Impeller, Casing, Suction pipe with a foot valve and strainer, and delivery pipe. We will discuss about this in depth afterwards.*

Keyword: *Discharge,Efficiency,Head,Power,Pump,Rotations per minute(R.P.M)*

I. Introduction

The hydraulic machines which convert the mechanical energy into hydraulic energy are called hydraulic pumps. The hydraulic energy is in the form of pressure energy. If the mechanical energy is converted into pressure energy by means of centrifugal force acting on the fluid, then the hydraulic machine is termed as centrifugal pump. It acts as the reverse of an inward flow reaction turbine. Thus this explains that the flow in centrifugal pumps is in radial outward directions. The centrifugal pump flows on the principle of vortex flow, which means that when a certain mass of a liquid is rotated by an external torque, there exists an increase in the pressure head of rotating fluids. This increase in pressure head at any point in the rotating fluid is proportional to the square of the tangential velocity of the liquid at that point (i.e., the increase in pressure head = $\frac{V^2}{2g}$). Thus where radius is more, generally at the outlet of the impeller, the rise in pressure head will be more and the liquid will be discharged at the outlet with a high pressure head. Hence, due to this high pressure head, the liquid can be lifted to a high level. Some applications of Centrifugal pump are –

1. Oil & Energy - pumping crude oil, slurry, mud; used by refineries, power generation plants
2. Industrial & Fire Protection Industry - Heating and ventilation, boiler feed applications, air conditioning, pressure boosting, fire protection sprinkler systems.
3. Waste Management, Agriculture & Manufacturing - Wastewater processing plants, municipal industry, drainage, gas processing, irrigation, and flood protection
4. Pharmaceutical, Chemical & Food Industries - paints, hydrocarbons, petrochemical, cellulose, sugar refining, food and beverage production
5. Various industries (Manufacturing, Industrial, Chemicals, Pharmaceutical, Food Production, Aerospace etc.) - for the purposes of cryogenics and refrigerants.

1.1 LITERATURE OVERVIEW

Pump designers are continually being challenged to provide machines that operate more efficiently, quietly and reliably at lower cost. Many investigators have applied CFD as a numerical simulation tool to carry out different investigations on centrifugal pumps. This section describes the research work carried out by the researchers in the centrifugal pumps by CFD approach. [5]

1.1.1 Performance prediction at different operating conditions:

Centrifugal pumps are widely used in many applications, so the pump system may be required to operate over a wide flow range in different applications. The most previous numerical studies were focused on the design or near-design state of pumps. Few efforts were made to study the off-design performance of pumps, where the performance of pump deteriorates [6]. With the aid of the CFD approach, the complex internal flows through the different components of pump can be studied at different operating conditions which help in improvement in the performance at off-design conditions. Mentzoset al. [7] carried out a numerical simulation of the internal flow in a backward curve vaned centrifugal pump. The MRF approach used to take into account the impeller-volute interaction was completely failed, due to its fixed coupling formulation. However, its use was recommended for basic understanding of the flow at various operating points. The transient analysis was suggested as a real tool for understanding of the interaction between impeller and spiral casing. Three-dimensional computational model of centrifugal pump is shown in Fig. 1.

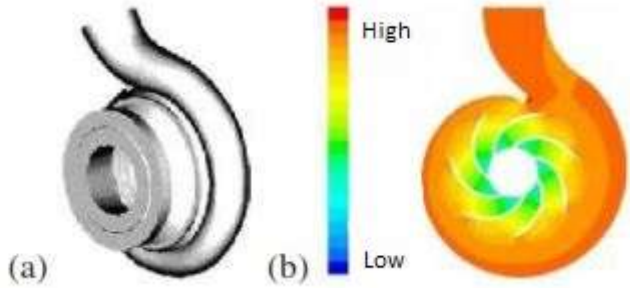

Fig. 1. Three-dimensional computational model of centrifugal pump and (b) static pressure contours in the pump.

Mentzos et al. [8] simulated the flow through the impeller of centrifugal pump using finite-volume method along with a structured grid system for the solution of the discretized governing equations. The CFD technique was applied to predict the flow patterns, pressure distribution and head-capacity curve. It was reported that, although the grid size was not adequate to investigate the local boundary layer variables, global ones were well captured. The proposed approach was advocated for the basic understanding of the flow at various operating points. Shah et al. [9] carried out steady state simulation of 200 m^3 /hr. capacity centrifugal pump using RANS equations. The non-uniformities were observed in different parts of the pump at off-design conditions which resulted in the decrease in efficiency. The k-ω SST turbulence model provided better results compared to RNG k-ε model. The operating characteristic curves predicted by the numerical simulation were compared with the results of model testing and were found in good agreement. The static pressure contours in the pump at rated discharge are shown in Fig. 1(b).

1.1.2 Parametric study:

CFD helps in prediction of flow behavior in different parts of the hydraulic machines before actually manufacturing them. In case of modification of existing systems, the modifications can be incorporated in numerical model and their effects can be predicted before implementing them. CFD analysis helps in studying the effects of various parameters, independently as well as by forming the non-dimensional groups, on pump performance. Bacharoudis et al. [10] analyzed the performance of pump by varying the outlet blade angles by keeping the same outlet diameter. The numerical simulation of 3-D, incompressible Navier-Stokes equations was carried out with a commercial CFD finite-volume code. 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. Anagnostopoulos [11] developed numerical model for the simulation of the 3-D turbulent flow in centrifugal pump impeller to solve the RANS equations. The impeller geometry was represented by a number of controllable design variables, providing the capability of modifying the impeller shape and testing different configurations. The results of such parametric studies showed that, a remarkable gain in hydraulic efficiency may be achieved by optimizing the impeller geometry.

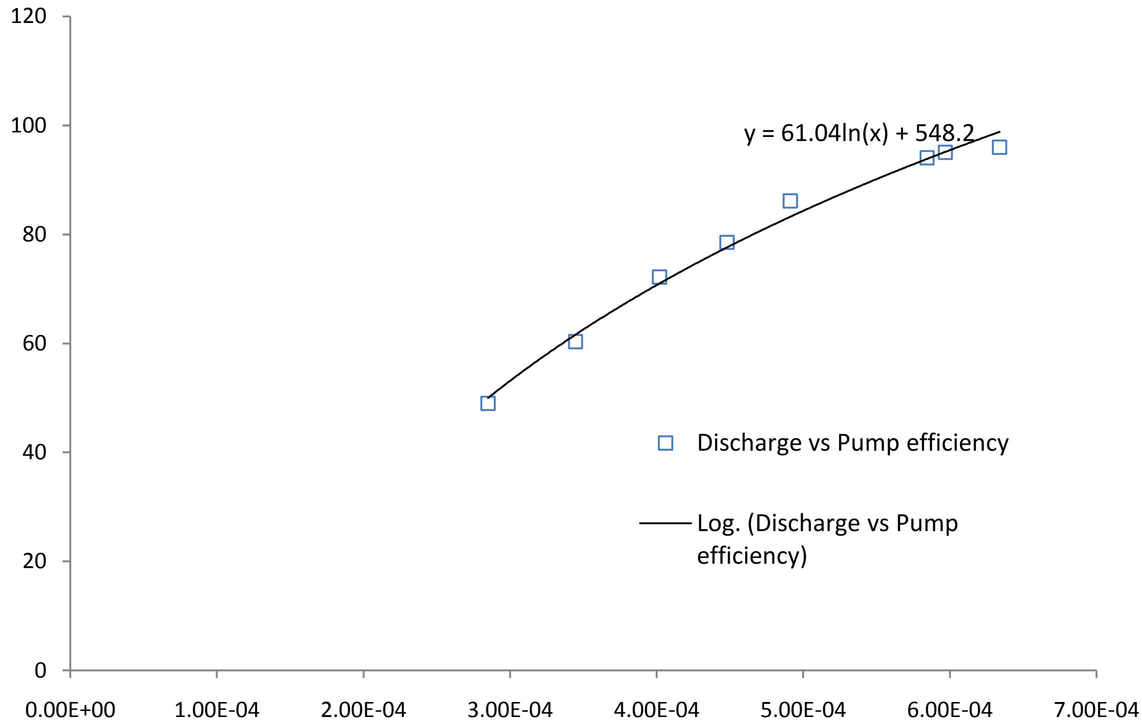

Fig. 2. Efficiency versus capacity curve.

Patel and Ramakrishnan [12] numerically studied the effects of changing hub curve profile and stator angle in mixed flow pump at duty point and at part load. The analysis concluded that: (i) the nature of head & power versus capacity curves obtained was similar to that of standard mixed flow pump (ii) pump efficiency was predicted within + 5% range at duty point. However, more variation was observed at off-design conditions and (iii) efficiency was improved by 1% after matching stator angle and changing hub curve profile. The efficiency versus capacity curves, actual and predicted by CFD analysis, are shown in Fig. 2.

1.1.3 Cavitation analysis:

Cavitation may occur in different regions of the pump when local pressure goes below the vapor pressure correspond to fluid temperature. The mechanism of cavitation erosion has been studied for more than a hundred years, but until now there has been no general theory of cavitation erosion damage to analytically calculate cavitation erosion rate in impellers of centrifugal pumps or to evaluate erosion intensity at the pump design stage. Medvitz et al. [13] used multi-phase CFD method to analyze centrifugal pump performance under cavitation conditions. The homogeneous two phase RANS equations were used wherein mixture momentum and volume continuity equations were solved along with vapor volume fraction. Performance trends of partial discharge and blade cavitation, including breakdown, were observed and compared qualitatively with experimental measurements. Nohmiet al. [14] studied the cavitation flow in a low specific speed centrifugal pump with compressible air-vapor-liquid two-phase medium (TE model) and constant enthalpy vaporization (CEV) model. The study revealed that, at the high flow rate cavitation bubbles appear at the leading edge on pressure side and the head drops gradually. The TE model was able to predict the gradual head drop but the computations were found to be unstable; whereas, CEV model was unable to predict the gradual head drop. In both the codes, further modification was recommended to achieve stable and accurate results. Caridad et al. [15] carried out numerical analysis in a centrifugal pump impeller of submersible pump conveying an air– water mixture, which was similar to cavitating flow. A sensibility analysis with regard to the gas-void fraction and the bubble diameter was performed. The variations in impeller head and relative flow angle at the outlet were presented as a function of liquid flow rate and phase distribution within the impeller. It was found that, larger bubble diameter lead to larger head experimented by the impeller. The numerical results and diffuser losses showed excellent agreement with the experimental results.

1.1.4 Investigations on interacting components:

The relative movement between impeller and volute generates an unsteady interaction which affects not only the overall pump performance but is also responsible for pressure fluctuations. Pressure fluctuations interact with the volute casing and give rise to dynamic effects (mainly unsteady forces) over the mechanical parts, which are one of the most important sources of vibration and hydraulic noise. Gonzalez et al. [16] showed the capability of a numerical simulation in capturing the dynamic and unsteady flow effects inside a centrifugal pump due to impeller-volute interaction. A viscous Navier-Stokes equation along with sliding mesh technique was applied to consider the impeller-volute interaction. The amplitude of the fluctuating pressure field at the blade passing frequency was successfully captured by the model for a wide range of operating flow rates. Both experiments and numerical prediction showed the presence of a spatial fluctuation pattern at the blade passing frequency as a function of the flow rate. Pressure fluctuations in the volute wall are shown in Fig. 5(a). Wang and Tsukamoto [17] developed numerical method for more realistic prediction of pressure fluctuations due to rotor-stator interaction in a diffuser pump by considering the change in operating point of the pump. Velocity vectors around channel 1 and 2 of impeller blades under rotating stall. The pressure fluctuations were predicted in 2-D unsteady incompressible flow using a vortex method, in which vortices shed from solid boundary were determined based on the momentum equations. It was reported that the pressure in the diffuser passage fluctuates with the basic frequency of the impeller blade passing frequency.

1.2 OBJECTIVE:

The primary objective of this experiment is to measure the performance of a centrifugal pump and compare the results to the manufacturer's specifications. Secondary objectives are to familiarize the student with the characteristics of a centrifugal pump and to introduce the student to the homologous scaling relationships. In addition, this experiment gives the student further exposure to the use of computerized data acquisition systems. Additional objectives may be specified by the instructor. [4]

II. Experimental Setup

The apparatus consists of a pump, motor, flow control valve, shaft torque meter, pressure transducers at the inlet and the outlet and two inline paddle wheel flowmeters. This is a 2" unit which measures all the flow that goes through the pump. Unfortunately, this flowmeter is unreliable at flowrates of less than about 15 gpm. Fluid velocity in the pipe is insufficient to overcome friction in the paddle wheel. However, there is a smaller, 3/4" flowmeter in the system which can be used at lower flows.[4] It only measures the flowrate through the three smallest pipes used for that experiment. To make use of it, the valves in the test rig must be set so that all of the pump flow goes through these smaller pipes. This small flowmeter can then be used at low pump flowrates. This flowmeter is not valid at flows greater than 20 gpm. The pump is a Price Pump Co. Model B-15, size 1-1/2x2x6. The size information indicates a 1-1/2 inch outlet pipe and a 2 inch inlet pipe with a nominal maximum impeller diameter of 6 inches. Our pump has a 5.88 inch impeller. Manufacturer's data for this pump is contained in Figures 4 and 5.

The motor is a two speed type that can operate at either a nominal 1800 or 3600 rpm depending on the setting of the connected control. The pressure transducers and the two flowmeters are connected to an IBM PC controlled data acquisition system. Each of the devices which are connected to the DAS has a voltage output which is proportional to the magnitude of the measured variable. See the above information on the validity of the flowmeter data. The shaft torque meter is a shaft which twists in torsion in linear proportion to the applied torque. This twist is measured by a scale on the rotating device.

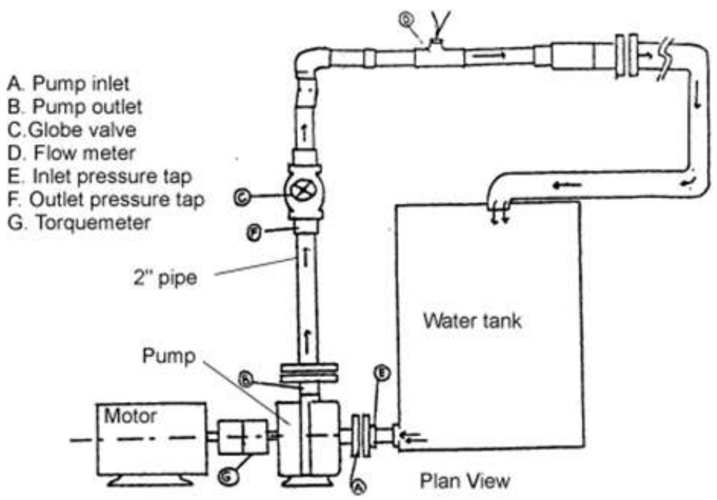

Fig. 3. Sketch For centrifugal pump apparatus

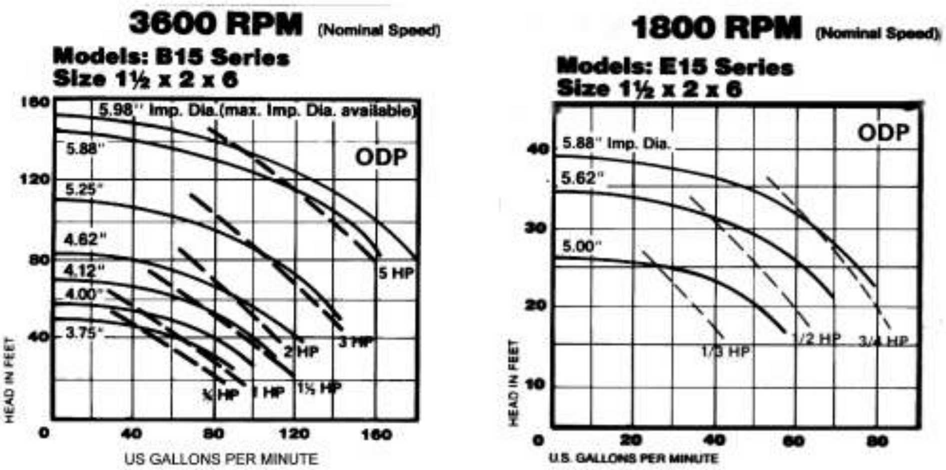

Fig. 4. Head vs. flow for Price Pump Model B15, 1 1/2x2x6, at 3600 and 1800 rpm Manufacturer's data sheet.

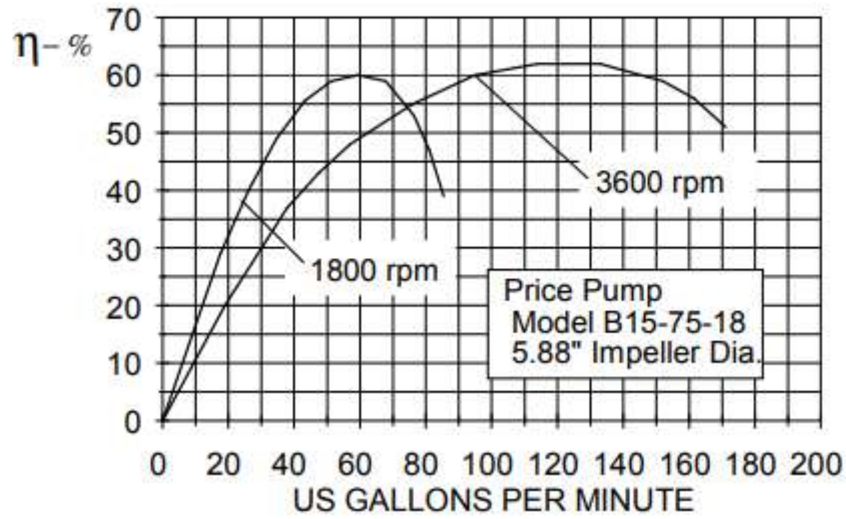

Fig. 5. Efficiency vs. flow for Price Pump model B15, 1 1/2x2x6, with 5.88" dia. impeller for 1800 and 3600 rpm. Based on mfg. data.

The torque meter can be read using a strobotachometer flashing at the shaft rotational speed. The strobotach can also be used to measure the shaft rotational speed. Both the inlet and outlet pressure taps are located on two inch schedule 40 plastic pipe which has an inside diameter of 2.067 inches. The actual outlet diameter of the pump is 1.610 inches and there is a sudden expansion to 2.067 inches and then short section of 2.067 pipe before the outlet pressure tap.One may want to evaluate these sources of head loss when using the mechanical energy equation to evaluate the pump head. A bourdon type pressure gage is also connected to the outlet pressure tap. This gage can be used to check the accuracy of the outlet pressure transducer. The accuracy of the inlet pressure transducer can be checked by applying the Bernoulli equation between the water surface in the supply tank and the location of the inlet pressure tap. Since the pressure and velocity at the water surface are both zero, the Bernoulli equation takes the form;

$$z_1 = \frac{P_{t1}}{\gamma} + \frac{V_{t1}^2}{2g}$$

where z_1 is the elevation of the water surface relative to the inlet pressure tap and P_{t1} and V_{t1} are the pressure and velocity in the pipe at the inlet pressure tap, respectively. Part of the same flow system is a tank placed on a mechanical scale. Using appropriate valves, the flow can be diverted to this weigh tank. Together with a timer, this system can be used to calibrate the 2" paddle wheel flow meter.

III. Indentations And Equations

In the calculations of the discharge, efficiency, water horse power and shaft power, the following formulas were undertaken;

$Q = \frac{(70X40X5X10-6)}{t}$

Discharge (Q) = l X b X h / t

where l = length of tank = 70 cm
b = breadth of tank = 40 cm
h = height of tank = 5 cm
t = time required to reach 5cm of height

$H_T = (13.6 \times 9.81 \times H_S + H_D \, X \, 9.81 \, X \, 10^4)/ 9810$

Total Height (H_T)
Suction Height (H_s)
Discharge Height (H_D)
WHP = 1000 X Q X HT

WHP = Work Horse Power
Q = Discharge
Total Height (H_T)

SHP = (N x 3600)/ (t_c X EMC)

SHP = Shaft Horse Power

N = No. of flashes = 10

t_c = time for N flashes

Energy meter constant =6400

SHP = (10 x 3600)/ (t_c X 6400)

EFFICIENCY (η) = WHP*100/SHP

IV. Result And Conclusion

According to the experiment conducted, our main objective of understanding the relationship of discharge with Head, pump efficiency, and RPM was done effectively by plotting the results.

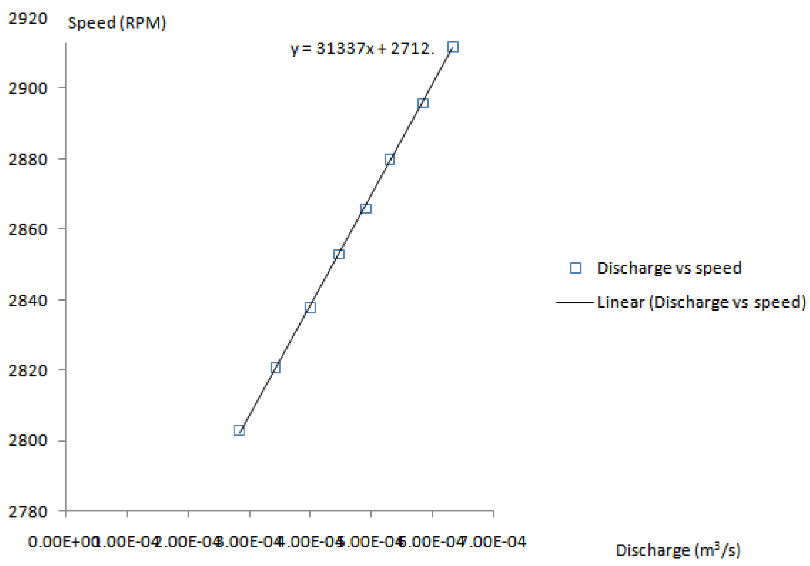

Fig. 6. Discharge vs speed

Thus it was observed that as the speed of the motor increased the the discharge also increased

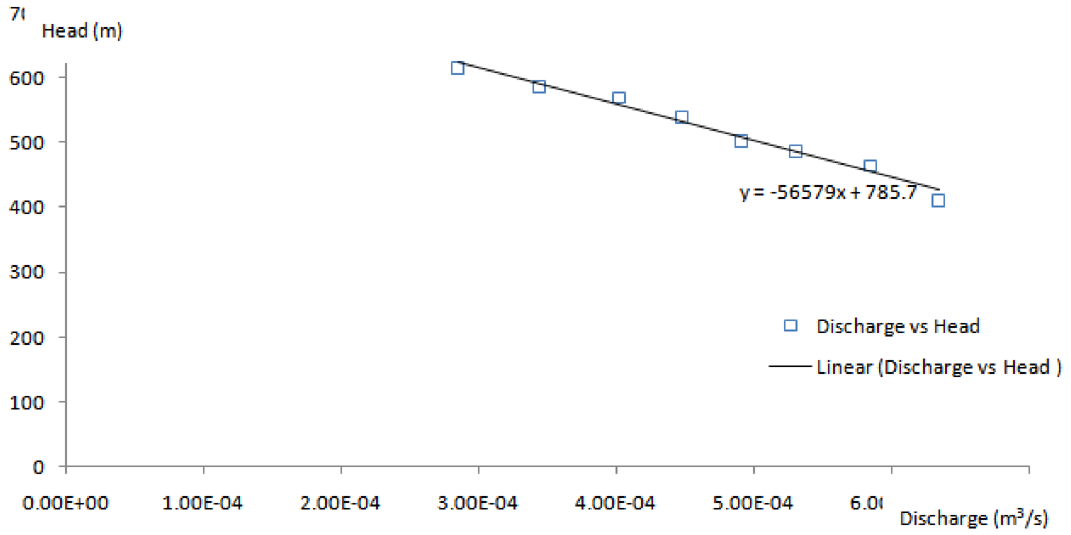

Fig. 7. Discharge vs Head

As the Discharge increased the Head also increased.

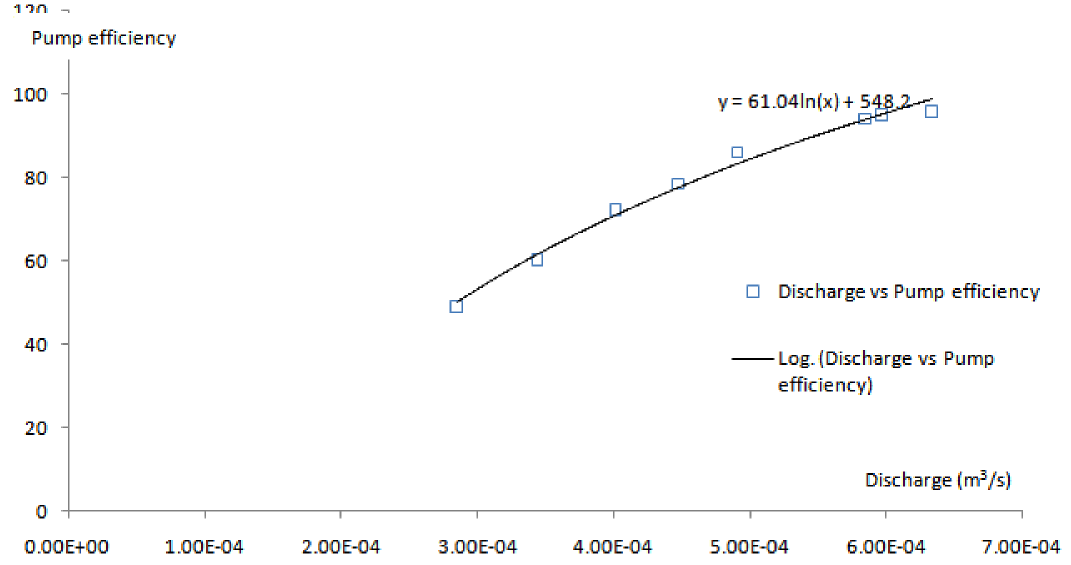

Fig. 8. Discharge vs Pump efficiency

For higher Discharges the Pump efficiencies also increased.

References

[1]. https://andersonpumpprocess.wordpress.com/2013/03/20/centrifugal-pump-advantage-disadvantage/

[2]. https://pumpbiz.com/blog/cat/pump-mentenance/post/right-centrifugal-pump-impeller-three-type/

[3]. https://www.nuclear-power.net/nuclear-engineering/fluid-dynamics/centrifugal-pumps/casing-volute/

[4]. http://userwww.sfsu.edu/ozer/302-EXP7%20S13.pdf

[5]. https://www.sciencedirect.com/science/article/pii/S1877705813001033

[6]. Hedi, L., Hatem, K.,Ridha, Z., 2010."Numerical flow simulation in a centrifugal pump*," International Renewable Energy Congress. Sousse, Tunisia.* pp. 300-304.

[7]. Mentzos, M.,Filios, A.,Margaris, P.,Papanikas, D.,2004. "A numerical simulation of the impeller-volute interaction in a centrifugal pump," *Proceedingsof International Conference from Scientific Computing to Computational Engineering. Athens*, pp. 1-7.

[8]. Mentzos, M., Filios, A., Margaris, P., Papanikas, D., 2005. "CFD predictions of flow through a centrifugal pump impeller," *Proceedings of InternationalConf. Experiments/Process/System Modelling/ Simulation/Optimization. Athens*, pp. 1-8.

[9]. Shah, S., Jain, S.,Lakhera, V., 2010. "CFD based flow analysis of centrifugal pump," *Proceedings ofInternational Conferenceon Fluid Mechanics andFluid Power. Chennai, India*, paper#TM08.

[10]. Bacharoudis, E.,Filios, A.,Mentzos, M.,Margaris, D., 2008.Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle. *TheOpen Mechanical Engineering Journal* 2, p. 75.

[11]. Anagnostopoulos, J., 2006. CFD Analysis and Design Effects in a Radial Pump Impeller. *WSEAS Transactions on Fluid Mechanics* 1 (7), p. 763.
[12]. Patel K.,Ramakrishnan, N., "CFD analysis of mixed flow pump."
[13]. Medvitz, R., Kunz, R., Boger, D., Adam, J.,Yocum, A., Pauley, L., 2002. Performance Analysis of Cavitating Flow in Centrifugal Pumps Using n Multiphase CFD. *Journal of Fluid Engineering* 124, p. 377.
[14]. Nohmi, M.,Goto, A.,Iga, Y.,Ikohagi, T., 2003."Cavitation CFD in a centrifugal pump," *Proceedings of International Symposium on Cavitation. Osaka,Japan,* pp. 1-7.
[15]. Caridad, J., Asuaje, M.,Kenyery, F.,Tremante, A.,Aguillon, O., 2008. Characterization of a Centrifugal Pump Impeller under Two-Phase FlowConditions. *Journal of Petroleum Science and Engineering* 63,p. 18.
[16]. Gonzalez, J., Fernandez, J., Blanco, E., Santolaria C., 2002.Numerical Simulation of the Dynamic Effects Due to Impeller-Volute Interaction in aCentrifugal Pump.*Journal of Fluids Engineering* 124,p. 348.
[17]. Wang, H., Tsukamoto, H., 2001. Fundamental Analysis on Rotor-Stator Interaction in a Diffuser Pump by Vortex Method.,*Journal of Fluids Engineering*123, p. 737.