Analysis of Effect of Impeller Parameters on Performance of Centrifugal Pump Using CFD

Mohankumar M |Hudson E Daniel Raj |M.Varatharaj

[1](PG Scholar, Engineering Design of Mechanical PG department, Sri Krishna College of Technology, Coimbatore, INDIA, mohankumar.manuneethi@gmail.com)

[2](Assistant professor, Mechanical PG department, Sri Krishna College of Technology, Coimbatore, INDIA, hudsondani@gmail.com)

[3](Head of the Department of EEE, Christ the King Engineering College, Coimbatore, INDIA, varatharaj_ms80@rediffmail.com)

***Abstract*—** *The impeller of a centrifugal pump plays an important role in converting the input driver energy into kinetic energy. The impeller is a complex structure and moreover the conventional trial and error method is cost consuming to experiment with. CFD analysis is a solution to this problem. This paper presents the numerical investigation of the effects that the pertinent design parameters, including the blade number, the inlet blade angle, trimmed impeller profile, and the impeller diameter, have on the steady state liquid flow in a three-dimensional centrifugal pump. Initially the impeller model is geometrically constructed based on certain performance modified parameters. The model is then meshed and CFD analysis was carried out. The inner flow fields, pressure and velocity distribution were predicted. Results obtained from the analysis were then compared with the actual results and inferences were made.*

Keywords*— *Circumferential velocity, Over filing,CFD, Impeller.

1. INTRODUCTION

A centrifugal pump is a non-positive displacement pump that imparts energy to a liquid. A centrifugal pump converts driver energy to kinetic energy in a liquid by accelerating it to the outer rim of a revolving device known as an impeller. The energy created is kinetic energy. The amount of energy given to the liquid corresponds to the velocity at the edge or vane tip of the impeller. Impeller is the rotating part of a centrifugal pump which imparts the kinetic energy to the system. The blades of the rotating impeller transfer energy to the fluid thereby increasing pressure and velocity. The fluid is sucked into the impeller at the impeller eye and flows through the impeller channels formed by the blades between the shroud and hub. The design of the impeller depends on the requirements for pressure, flow and application. The impeller is the primary component determining the pump performance. Pumps variants are often created only by modifying the impeller. The impeller accounts for nearly 17.5% of losses in a pump.

2. LITERATURE REVIEW

Chien-Hsiung Tsai, et al [1] studied the effect of installation of spoiler on aerodynamics and aero-acoustics of a vehicle, six cases were considered. Case 1 is the base model which corresponds to a car without a spoiler another five spoilers with different cross-sectional profiles was considered. Symmetrical boundary condition was applied on the plane where the actual computational domain was split into two.

On the plane upstream of the vehicle model, a constant velocity of 180 km/hr, which corresponded to Re = 3.6x106, was applied. This speed was used because 180 km/hr is almost the maximum speed reasonably achievable by a Honda S2000. This maximum speed was chosen over other speeds due to the fact that noise generation is more critical at a higher speed. Similarly, this boundary condition was also applied at the plane on top of the vehicle model as well as the plane opposite of the symmetrical plane. The plane downstream of the vehicle model was assumed a pressure outlet of 1 atm. Non-slip condition was applied on all of the solid surfaces. The combination of these boundary conditions implies the simulation of a vehicle model securely mounted in the test section of a wind tunnel.

Among these six cases, there are more positive pressure coefficients on the upper wing of the spoiler. Therefore, its lift force is the minimum among all models. For all the cases, the pressure coefficients in front of the bumper and the windscreen were positive, but that associated to the roof was always negative. Recall that case 1 corresponds to the vehicle model without the installation of a spoiler. If a spoiler (or a pair of spoilers) is installed on top of the trunk, the drag coefficient increases while the lift coefficient decreases.

R. B. Sharma, et al [2] in this investigation of paper Computational fluid dynamics (CFD) simulations of the steady flow field around passenger car models with and without Tail Plates presented and compared the simulated data to each other. The ANSYS-14.0 Fluent with the k-e steady model is used for the simulations of aerodynamics. In this analysis, the coefficient of drag has been reduced 3.87% and coefficient of lift is reduced 16.62%. Hence, the Tail Plates is the effective tool to reduce the drag force on vehicle.

The effects of different aerodynamic add-on devices on flow and its structure over a generic passenger car may be analyzed using CFD approach. The objective is to reduce aerodynamic drag acting on the vehicle and thus improve the fuel efficiency of passenger car. Hence, the drag force can be reduced by using add on devices on vehicle and fuel economy, stability of a passenger car can be improved.

Joseph Katz [3] studied the complexity of automobile and concluded that race car aerodynamics is comparable to airplane aerodynamics and is not limited to drag reduction only. The generation of downforce and its effect on lateral stability has a major effect on race car performance, particularly when high speed turns are involved. In the process of designing and refining current race car shapes, all aerospace-type design tools are used. Because of effects such as flow separations, vortex flows, or boundary-layer transition, the flow over most types of race cars is not always easily predictable. Due to the competitive nature of this sport and the short design cycles, engineering decisions must rely on combined information from track, wind tunnel, and CFD tests.

Ram Bansal, et al [4] analyzed different types of aerodynamic add-on devices used on the baseline car to get the results for the coefficient of drag and coefficient of lift. In the first case the spoiler is applied on the boot of the passenger car with the inclination angle 12◦ .The coefficient of drag is 0.3441 and the coefficient of lift is 0.1985.The percentage reduction in drag coefficient in comparison with baseline car is 2.02%and in coefficient of lift is 6%.Hence, drag force and lift force on the passenger car are reduced as proportional to drag coefficient and lift coefficient, respectively. In the second case the vortex generators are applied on the rear side at roof of the baseline car with inclination angle 12◦ . The coefficient of drag is 0.3471 and the coefficient of lift is 0.2085.

The percentage reduction in drag coefficient in comparison with baseline car is 1.17% and in coefficient of lift is 9.8%. Hence drag force and lift force on the passenger car are reduced as proportional to drag coefficient and lift coefficient, respectively. In the third case the tail plates are applied on the rear side: one is at the rear side of the roof garnish and the other is at the tail bumper of the passenger car.

The coefficient of drag is 0.3376 and the coefficient of lift is 0.1926. The percentage reduction in drag coefficient in comparison with base line car is 3.87% and in coefficient of lift is 16.62%. Hence drag force and lift force on the passenger car are reduced as proportional to drag coefficient and lift coefficient, respectively. In the fourth case the spoiler and VGs together are applied on the rear boot and rear side at the roof of the passenger car. The coefficient of drag is 0.3359 and the coefficient of lift is 0.1875.

The percentage reduction in drag coefficient in comparison with base line car is 4.35% and in coefficient of lift is 18.83%. Hence, drag force and lift force on the passenger car are reduced as proportional to drag coefficient and lift coefficient, respectively.

From the above analysis, it is found that spoiler with VGs is more effective add-on device to reduce the drag coefficient and lift coefficient which are applied on the passenger car when the car is running on the road. The drag coefficients and drag forces are proportional to each other so when the drag forces are reduced, lift forces are also reduced because it is proportional to the lift coefficient.

Tank Nilesh R, et al [5] presented that transient condition is inevitable for cross flow condition. Stagnation pressure is high at windward side compare to steady case, so more prone to yawing moment and side force which leads to instability of the vehicles. Tapered front section will slightly decrease the yawing moment and side force effects on overtaking vehicle. Notchback exhibits highest yawing moment and square back will lowest while reverse for side force.

Rounding of the edges of front end and hood and hood sides increase the yawing moment and leads to lower pressure on leeward side of front end. Increase in side projection area will leads to decrease in side force at rear end but will increase in yawing moment due to increment in length. Low placed cowl with less extent and greater wind shield angle reduce the front side force and yawing moment.

Jesper Marklund [6] in the paper presented that passenger cars are bluff bodies aerodynamically, with the pressure drag dominating the friction drag. Reducing the pressure drag has the greatest potential and the focus should be at this. The exterior body of a passenger vehicle, such as the upper and under-body, is responsible for more than 60 % of the drag. The rest is related to cooling flows and wheels. It is therefore relevant to study the vehicle body alone, and do so with simplified models of vehicle proportions. Simplified bluff bodies were used to study the overall effects of the body shape with regard to ground proximity, ground simulation, yaw and pitch angles.

The rear-end shape was seen to have a major effect on the flow distribution around the body. The effect of drag and lift dependence on ground proximity is very different depending on rear-end shape.

- ✓ A body of vehicle proportions will have reduced drag by a nose-down pitch as long as the flow does not generate a negative camber.
- ✓ There is an optimum drag relative pitch angle and yaw angle of a body.
- ✓ A square back model will have a larger drag increase by a yaw angle than other shapes.
- ✓ Drag reduction of an under-body diffuser, applied to a simplified bluff body is depending on the upper body. A model with large backlight angle will have better drag reduction by a diffuser than a square back model.
- ✓ Diffusers applied to passenger vehicles, with road vehicle boundary conditions, follow the same trend as diffusers applied to simplified bluff bodies. The main part of lift and drag of a passenger car is related to the main body.

- ✓ The sedan vehicle has greater potential to reduce drag by using a smooth floor and under-body diffuser than the wagon. The diffuser generates an upsweep and makes the wake more symmetrical in side view, for the sedan. For the wagon the wake is already symmetrical and the diffuser can even generate an asymmetric wake.

- ✓ Wake symmetry is very important for optimum pressure recovery of passenger vehicles.

- ✓ The effect of correct ground simulation is shown to be important since the flow around the body will be re-distributed to have more flow under the body.

Gopal P, et al [7] experimental investigated on the measurement of the variation of pressure coefficient and dynamic pressure on the roof of a utility vehicle with and without vortex generators (VG), the following conclusions were made:

- ✓ The value of pressure coefficient without VG is minimum whereas its value was observed to be maximum with VG having yaw angle of 15°.
- ✓ The pressure coefficient can be increased with the inclusion of VG by around 17% at a velocity of 2.42 m/s.
- ✓ The values of pressure coefficient don't change significantly with increase in velocity for various values of yaw angle.
- ✓ Dynamic pressure over the surface of the vehicle roof increases with addition of VG which is favorable for avoiding flow separation and the consequent losses.
- ✓ The value of CD is reduced by 90% with the addition of VG at a velocity of 2.42 m/s and a minimum of 20% reduction in drag is obtained for VG with a yaw angle of 10°
- ✓ It is observed that VG with a yaw angle of 15° will be useful at lower velocity
- ✓ The value of CL decreases with increase in velocity with and without VG and the results revealed that at higher velocity the value of CL remains constant for VG with varying yaw angles.

Mahmoud Khaled, et al [8] concludes that unlike the lift force, the drag is smaller when the body lower face outlet is centered over the width of the model. Positioned on the left or the right, the body lower face air outlet induces the same effect on the aerodynamic torsor. The further towards the top the vertical air outlet is, the lower the induced drag and the higher the lift force. The aerodynamic drag is independent of the position of the air outlet in the wheel arch. However, placing this outlet at the rear of the wheel arch decreases the lift force.

The drag coefficient as well as the pitch momentum coefficient increase with the inlet/outlet section ratio up to 0.7, beyond which they become almost constant.

The lift coefficient decreases with the ratio between the inlet and outlet section also up to 0.7, from which it varies only slightly. It is also shown that increasing the distance in the *X* direction between the cooling module and the engine block from 6 to 20 cm reduces the drag coefficient by 1.4%, the cooling drag coefficient by 17.4%, and the lift coefficient by 1.8%. Also, shifting the cooling module in the right direction (through the *Y* direction) from the engine block, i.e., varying the distance *Y* between the two components, reduces the cooling drag by 1.1%, the cooling drag coefficient by 12.8%, and the pitch momentum coefficient by 1.5%.

With an air inlet opening centered in the *Z* direction, one reduces, with respect to classical air inlet position, the drag coefficient by 1.3%, the cooling drag coefficient by 56.4%, and the pitching momentum coefficient by 3.6%.

Xingjun Hu, et al (9) proposed that when the diffuser angle varied from 0□to 12□, the total aerodynamic drag coefficients of car first decrease and then increases, while the total aerodynamic lift coefficients decrease. There is a diffuser angle at which the sedan can obtain the minimum drag coefficient. From the calculation results, we can find when the diffuser angle varied from 0° to 12°, pressure distributing on the top of body and the front of body changed little, with the increase of diffuser angle, the distribution area of positive pressure on the rear of the body first increases and then decrease.

The distribution area of positive pressure reach peak at diffuser angle of 6°. Difference of positive pressure distribution on the rear of the body lead to differential pressure of the body surface varies from case to case, which results in the total aerodynamic drag coefficients of car first decreasing and then increasing while diffuser angle changes. Negative pressure is generated at the underbody interface and the region of the negative pressure become larger and larger. At the same time, the positive pressure generated at the edge of underbody decrease. When diffuser angle is changed to 9.8 degree, there is not any positive pressure distributing at the edge of underbody.

The increase of negative pressure distribution and the decrease of the positive pressure distribution on the underbody lead to increase in differential pressure of the body surface, which results in decreasing of total aerodynamic lift coefficients, found the negative pressure peak at the start of the diffuser. The streamline behind the vehicle can be seen for diffuser angles from 0 to 12 degrees. From the streamline and pressure contour of the sedan, it can be found when the diffuser angles varies from 0 degree to 12 degrees, the flow field after the sedan has an obvious change. It can be seen obviously the wake structures after the car of all cases are different.

3. OBJECTIVE

The impeller is the major part of a pump that can be modified to improve its performance. The physical testing of various pump models is a laborious process.

The trial and error method is time consuming and has many disadvantages**.** CAD and CFD analysis are useful tools that reduce considerable time that is usually lost in physical testing.

CAD aids in constructing the geometrical profile on a computer and CFD analysis provides the necessary virtual simulation without using any physical effort. Hence an impeller with the performance modified specification is constructed and the flow through it is virtually analyzed using CFD tools to determine the effect of the parameters on the performance.

4. METHODOLOGY

A. EXISTING PUMP DATA

Table 1 Existing Pump Data

Pump Std	IS:14220
Motor Rating	3.7kw / 5 hp
Speed	2780 rpm
Total head	24 m
Discharge	8 lps
Delivery size	50 mm
Voltage	380
Overall η	43.5%

B. PARAMETERS

The key parameters that influence the impeller are the number of blades, inlet angle, outer diameter and trimmed exit vanes.

OUTER DIAMETER

As per BEE recommendations, impeller diameter reductions greater than 5% to 10% of the maximum will increase the NPSHR (net positive suction head required). The outer diameter of the impeller determines the circumferential velocity of the exit vanes.

INLET VANE ANGLE

The initial kinetic energy developed by the fluid depends on the inlet vane angle of the impeller. The velocity triangles are used to calculate the inlet vane angle.

From inlet velocity triangle,

$\tan \theta = v_{f1} / u_1$

NUMBER OF BLADES

The selection of number of blades is crucial because, selecting a higher or lower number might result in considerable performance reduction. The number of blades is given by

$Z = [\ 6.5 * (D_2+D_1 / D_2 - D_1) * \sin(\beta_1+\beta_2)\ ] / 2$

TRIMMED EXIT VANES

The concept of underfiling the exit vanes at the outlet is extensively used in the industrial field to improve the efficiency. Over filing will increase the pump discharge, especially for large circulating pumps.

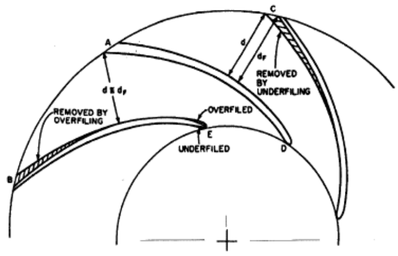


Fig 1 Impeller trimming.

The exit angle of the fluid will change resulting in a higher head at design flow,

Table 2 Process Parameters

PARAMETERS	VALUE
No. of blades	6
Outer Dia	170mm
Blade inlet angle	37 º
Trimmed exit vane	Overfiled profile

5. MODEL CONSTRUCTION

Computer Aided Drawing is a valuable tool which aids in the construction of a geometric model in a computer. The impeller model was created in CATIA V5 software.

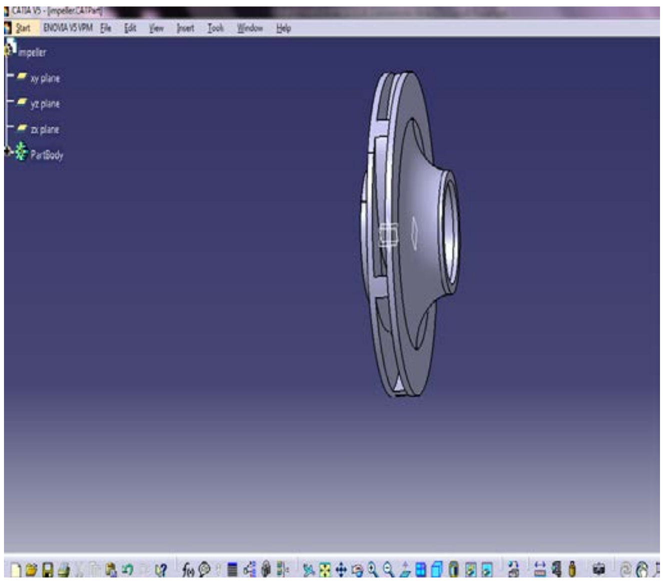


Fig 2 Impeller model 1 in CATIA V5.

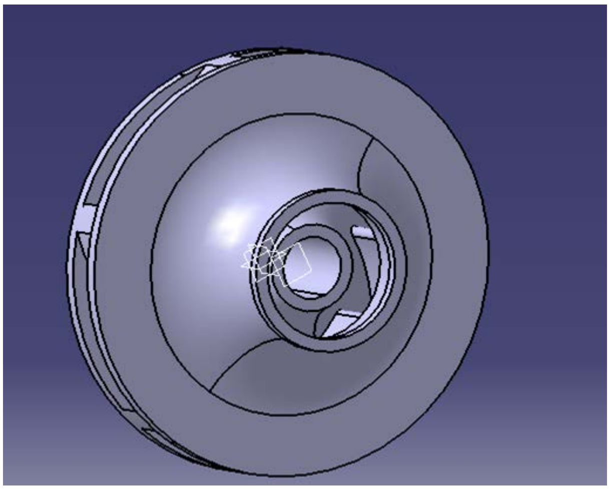


Fig 2 Impeller 2 in CATIA V5.

CFD ANALYSIS

The geometric model created with the help of CATIA V5 software was exported to the SolidWorks Flow Simulation software for carrying out the CFD analysis. CFD analysis was carried out by specifying the input data

6. RESULTS AND DISCUSSIONS

Numerical simulations were carried out to predict the performance and to determine the discharge for the given input data. Successive iterations were performed in the Flow simulation solver to obtain the flow rate, pressure distribution, peripheral velocity distribution, torque and efficiency of the impeller. The circumferential velocity contours indicate high velocity values near the trimmed exit vane regions.

This is a testament to the fact that impeller trimming has a significant effect on circumferential velocity developed by the exit vanes. The torque on the impeller obtained from the CFD simulation is 8.99 Nm and the

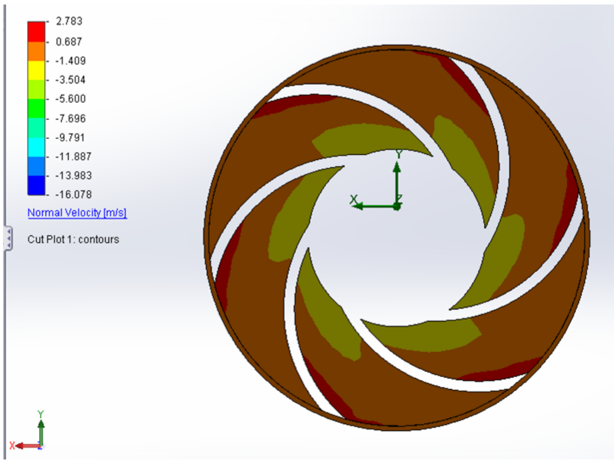


Fig 3 Velocity Distribution.

Discharge obtained is 0.009 m^3/s. The velocity and pressure distribution near the impeller eye are uniformly distributed which indicate the absence of low pressure areas that are mainly responsible for cavitation

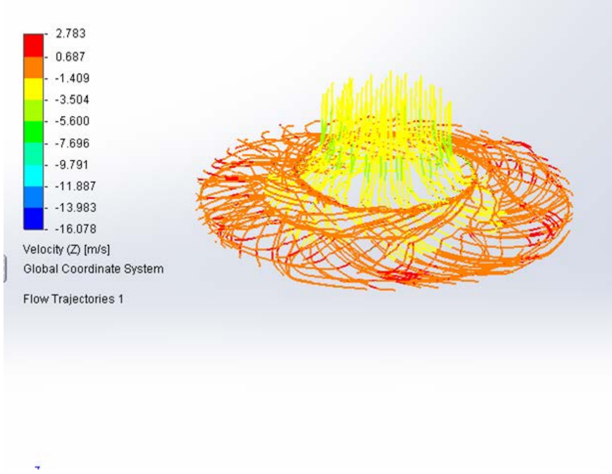
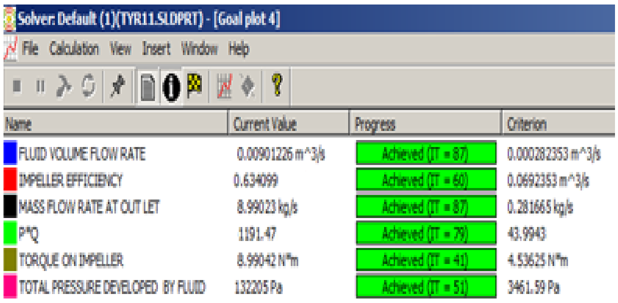


Fig 4 Flow Trajectory



Solver: Default (1)(TYR11.SLDPRT) - [Goal plot 4]

File Calculation View Insert Window Help

Name	Current Value	Progress	Criterion
FLUID VOLUME FLOW RATE	0.00901226 m^3/s	Achieved (IT = 87)	0.000282353 m^3/s
IMPELLER EFFICIENCY	0.634099	Achieved (IT = 60)	0.0692353 m^3/s
MASS FLOW RATE AT OUTLET	8.99023 kg/s	Achieved (IT = 87)	0.281665 kg/s
P*Q	1191.47	Achieved (IT = 79)	43.9943
TORQUE ON IMPELLER	8.99042 N*m	Achieved (IT = 41)	4.53625 N*m
TOTAL PRESSURE DEVELOPED BY FLUID	132205 Pa	Achieved (IT = 51)	3461.59 Pa

Fig 5 Solver depicting output data

7. CONCLUSION

CFD analysis was carried out using Solidworks Flow Simulation software to determine the discharge and to validate the role of key impeller parameters in improving the performance of the centrifugal pump. The pressure distribution, velocity distribution and torque of the system were obtained in this analysis. There is further scope for future work to optimize the performance variables to achieve higher efficiency values. The entire pumping theory and every component and part of the centrifugal pump can be thoroughly studied to identify further process variables to improve the efficiency of the system.

NOMENCLATURE

Z	Number of blades of impeller
D_1	Impeller eye diameter (mm)
D_2	Outlet diameter of impeller (mm)
β_1	Blade inlet angle (deg)
β_2	Blade outlet angle (deg)
H	Head (mm)
N	Pump speed (rpm)
Q	Flow rate (lps)
P	Pressure (Pa)
u	Peripheral velocity (mm/s)

References

[1]. Chien-Hsiung Tsai, Lung-Ming Fu, Chang-Hsien Tai, Yen-Loung Huang, Jik-Chang Leong (2009): **"Computational aero-acoustic analysis of a passenger car with a rear spoiler"**, pp 3661–3673

[2]. R. B. Sharma, Ram Bansal (2013): **"CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction",** pp 28-35

[3]. Joseph Katz (2008): **"Aerodynamics of Race Cars"**, pp 27-63

[4]. Ram Bansal, R. B. Sharma (2014): **"Drag Reduction of Passenger Car Using Add-On Devices"**

[5]. Tank Nilesh R, R. Thundil Karuppa Raj (2012): **"Numerical Simulation of Aerodynamic forces acting on Passenger Vehicle While Overtaking"**, pp 55-63

[6]. Jesper Marklund (2013): "**Under-body and Diffuser Flows of Passenger Vehicles"**

[7]. Gopal P, Senthilkumar T (2012): **"Aerodynamic Drag Reduction In A Passenger Vehicle Using Vortex Generator With Varying Yaw Angles"**

[8]. Mahmoud Khaled, Hicham El Hage, Fabien Harambat, Hassan Peerhossaini (2012): **"Some innovative concepts for car drag reduction: A parametric analysis of aerodynamic forces on a simplified body"**

[10]. Xingjun Hu, Rui Zhang, Jian Ye , Xu Yan , Zhiming Zhao (2011): "**Some innovative concepts for car drag reduction: A parametric analysis of aerodynamic forces on a simplified body"**

[11]. Khalid.S.Rababa (2011), "The Effect of Blades Number and Shape on the Operating Characteristics of Groundwater Centrifugal Pumps", European Journal of Scientific Research, pp 243- 251.

[12]. L.Subramaniam and S.Sendilvelan, (2012), "Modal Analysis of a Centrifugal Pump Impeller", European Journal of Scientific Research, pp 5- 14

[13]. Manivannan.A (2010), "Computational Fluid Dynamics of a mixed flow Pump Impeller", International Journal of Engineering, Science and Technology, pp 200-206

[14]. Wen-Guang LI (2011), "Inverse design of Impeller blade of centrifugal pump with a singularity method", Jordan Journal of Mechanical and Industrial Engineering, pp 119-128