

Effect of Blade Geometry on Hydraulic Performance of the Centrifugal pump

Ravi Teggin¹, Shashank M Hebbal², M.S.Hebbal³P.G. Student, Department of Mechanical Engineering, BEC, Bagalkot, Karnataka, India¹P.G. Student, Department of Mechanical Engineering, BEC, Bagalkot, Karnataka, India²Professor, Department of Mechanical Engineering, BEC, Bagalkot, Karnataka, India³

ABSTRACT: The hydraulic performance of a centrifugal pump greatly depends on its geometrical parameters of the vane profile. In the literature the guidelines are available to determine these parameters. However in many cases the guidelines end up with pointing the range of parameters than pointing a specific value. Therefore, the process of designing the impeller does not guarantee the optimum performance. In the present paper Computational Fluid Dynamics (CFD) analysis of flow through the impeller performed on different impeller configuration is presented. Different configuration of the impellers is obtained by considering the impellers with the vane outlet angle ($23^{\circ}, 25^{\circ}, 27^{\circ}$) and number of vanes (five, six, and seven). ANSYS BladeGen, ANSYS TurboGrid and ANSYS CFX modules are used for geometric modelling, meshing and solution respectively. Based on the CFD analysis it is found that, the hydraulic efficiency is maximum for impeller with outlet vane angle equal to 25° and having seven vanes.

KEYWORDS: Centrifugal pump, Impeller, Grid dependence test, Computational fluid dynamics (CFD), Vane outlet angle.

I. INTRODUCTION

Computational Fluid Dynamics is a very powerful tool which provides a cost effective method of predicting the flow pattern in the hydraulic passage of impellers. It helps to identify the abrupt changes in the flow parameters, recirculation and flow separation zones. Such deficiencies can be addressed effectively by suitable modifications to vane and shroud geometry. The use of CFD tools to analyse the flow field in turbo machines and to predict performance parameters has gained enormous popularity in recent years. Sedlar and Mensik [1] give examples of calculations of radial flow pump stages with semi-axial and radial diffusers.

Prime objective of impeller is to increase the efficiency and at the same time decrease the Net Positive Suction Head (NPSH) required. In the work of E.C.Bacharoudis [2], the vane outlet angle with $20^{\circ}, 30^{\circ}$ and 50° are investigated and it is reported that the increase in vane outlet angle causes a significant improvement in hydraulic efficiency. But it does not ensure the stable performance when operated away from the duty point. According to Val. S. Lobanoff et.al [3], for approximately 15 to 25 percentage of raise in head from best efficiency point to shut off, the number of vanes recommended is five to seven and outlet angle 23° to 27° . Therefore, the impellers configurations having the above parameter ranges are considered for investigation to identify the optimum impeller configuration. Problems associated with a hydraulic design of centrifugal pumps have been discussed in many works [4-6]. In these works the hydraulic design of centrifugal pump is based on theory of one dimensional streaming with correction to finite blade count. Hibocan.P [7] and Hibocan.P et.al [8] emphasized the utility of synergy of hydraulic methods based on the assumption of the one dimensional flow model and CFD simulation, which is based on an assumption of three dimensional streaming.

Numerous studies have been carried out to improve hydraulic performance of centrifugal pumps, including their performance [9-11]. Although recent pumps show a high performance, there is a scope for more advanced systematic approach such as optimization to achieve more striking performance improvement.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 7, July 2016

II. MATHEMATICAL MODELING OF IMPELLER

2.1 Design and geometric modelling of impeller

In order to carry out the flow analysis through impeller of a centrifugal pump, it is essential to have geometric model of the pump. To determine the various dimensions of the impeller, certain minimum basic inputs are essential and they are specified in Table 1.

Table 1: Design specifications used for study

Parameter	Magnitude
Flow rate (Q) at Best Efficiency Point	4500 m ³ /h
Head (H)	60 m
Rotating speed (N)	600 rpm
Density (ρ) of working fluid (Water)	1000 kg/m ³

In addition to this, hydraulic efficiency, volumetric efficiency, mechanical efficiency, vane outlet angle and number of vanes are the additional required inputs to determine the basic dimensions of the impeller. The Hydraulic efficiency as a function of specific speed and flow rate are available in literature. Similarly graphs for volumetric efficiency versus flow rate and approximate mechanical efficiency as a function of flow rate and operating speed are available. These values are referred from CF Turbo 9.0 help manual and are given in Table 2. Based on the literature given in [3] the vane outlet angle and the number of vanes are considered for design of base impeller.

Table 2: Parameters

Parameters	Magnitude
Hydraulic efficiency (η _h)	0.94
Volumetric efficiency (η _v)	0.98
Mechanical efficiency (η _m)	0.99
Vane outlet angle (β ₂)	25 ⁰
No. of vanes (Z)	6

The specific speed of the pump is calculated by the equation (1).

$$N_s = \frac{N\sqrt{Q}}{(H)^{3/4}} \quad (1)$$

Where, 'Q' is flow rate in m³/s. The specific speed of the pump for the duty parameters at best efficiency is 31.1.

The impeller dimensions determined are listed in Table 3.

Table 3: Impeller parameters

Parameter	Calculated values according to [3]	Values from VISTA CPD
Inlet dia (D ₁)	611 mm	579 mm
Outlet dia (D ₂)	1175 mm	1088 mm
Impeller outlet width(b ₂)	85 mm	120 mm
Blade thickness (S _u)	12.5 mm	30 mm
Shaft dia (d)	120 mm	124 mm
Inlet blade angle	(β ₁) =15.5 ⁰	(β _{1h}) =32.69 ⁰ , (β _{1s}) =22.02 ⁰

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 7, July 2016

Three dimensional geometry (Fig. 1(a)) is created in ANSYS BladeGEN using the data of base impeller from VISTA CPD. The flow passage about one vane, half on either side is considered as computational domain (Fig. 1(b)) to make use of cyclic symmetry of the flow passage. This will reduce the computational time. The computational domain is meshed in ANSYS TurboGrid.

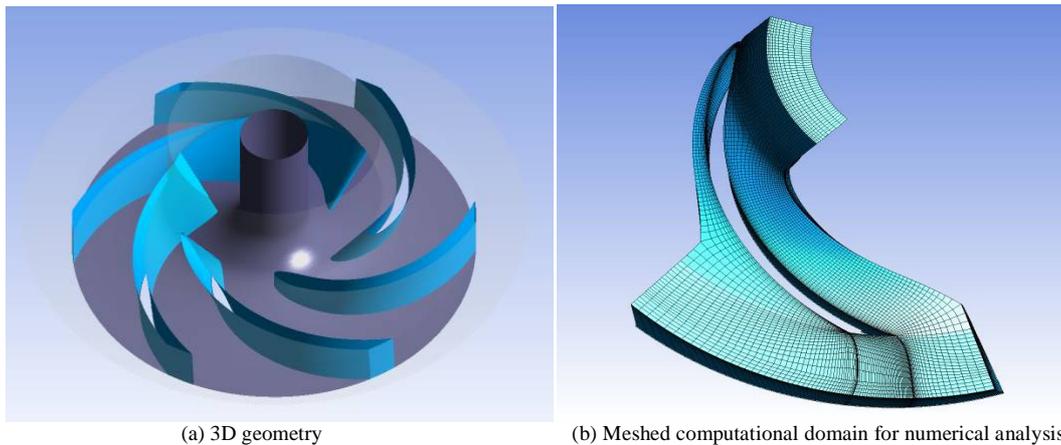


Fig.1. 3D Model of impeller

A grid dependence test (Fig.2) of the impeller was conducted to confirm the convergence of the results (head and efficiency). It is found that the result converges when 1,00,000 nodes are used for analysis. This grid system assures less than 0.1% performance error.

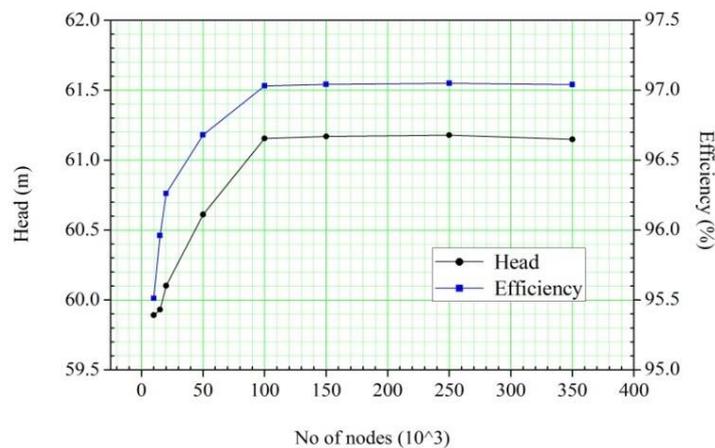


Fig.2. Results of grid dependence test

2.2 Performance evaluation of the impeller using CFD

ANSYS CFX is used to evaluate the hydraulic performance of the impellers with different configurations specified in Table 4. A high-resolution technique with an accuracy of second order or higher was used. The shear stress transport (SST) turbulent model is adopted for numerical analysis. Similar strategy is used in [12].

The reference pressure was set to 1 atm. Boundary condition of zero atm. pressure is applied at inlet of the impeller. Keeping these boundary conditions, different mass flow rate are applied at outlet of the impeller to evaluate the performance of impeller at different flow rates. The performance of the base impeller obtained from the analysis is

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 7, July 2016

depicted in Fig.3. From the results of the analysis it is found that, the head and efficiency of the base impeller at designed flow rate is 61.1 m. and 97 percentage respectively.

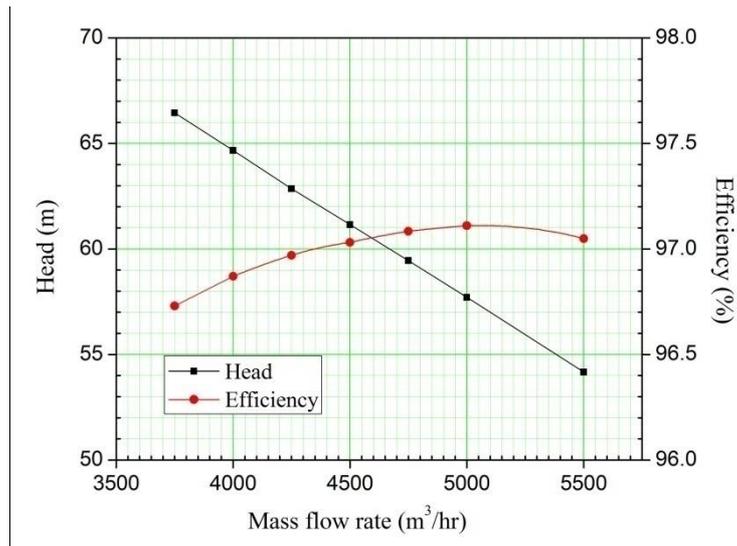


Fig.3. Performance curve for the base impeller

Best Efficiency of 97 % is resulted at 5000 m³/h flow rate, which is off the design point by 11.1 %. Therefore, impellers with different configurations (Table 4) are considered for further analysis. Vane outlet angle of 23⁰, 25⁰, 27⁰ and five, six, seven number of vanes are considered as design variables to study their effect on hydraulic performance. Same boundary conditions are applied on all the impellers considered for analysis and the results (head and efficiency) of different impeller configurations are enumerated in Table 4.

Table 4: Impellers configurations

Impeller Configuration no.	Vane outlet angle	Number of vanes	At Design point	
			Head (m)	Efficiency (%)
1	23 ⁰	5	60.16	95.31
2	25 ⁰	5	61.10	96.21
3	27 ⁰	5	61.49	95.73
4	23 ⁰	6	60.30	96.06
5	25 ⁰	6	61.16	97.03
6	27 ⁰	6	61.73	96.31
7	23 ⁰	7	60.51	97.00
8	25 ⁰	7	61.31	97.60
9	27 ⁰	7	62.00	97.41

III. RESULTS AND DISCUSSIONS

3.1 Effect of vane outlet angle on head

The effect of vane outlet angle on the head developed for impeller with five, six and seven number of vanes is shown in Fig.4 (a), Fig.4 (b) and Fig.4 (c) respectively. Head developed is large in case of impellers having larger vane outlet angle. A maximum head of 61.5 m and minimum of 60.2 m at design point (Fig. 4a) is developed for the impeller having 5 vanes with 27⁰ and 23⁰ vane outlet angles respectively. Similar trend is observed in case of impeller with six

and seven number of vanes also. The difference between maximum and minimum head developed is 1.3 m, 0.4 m and 1.5 m for impellers with five, six and seven vanes respectively.

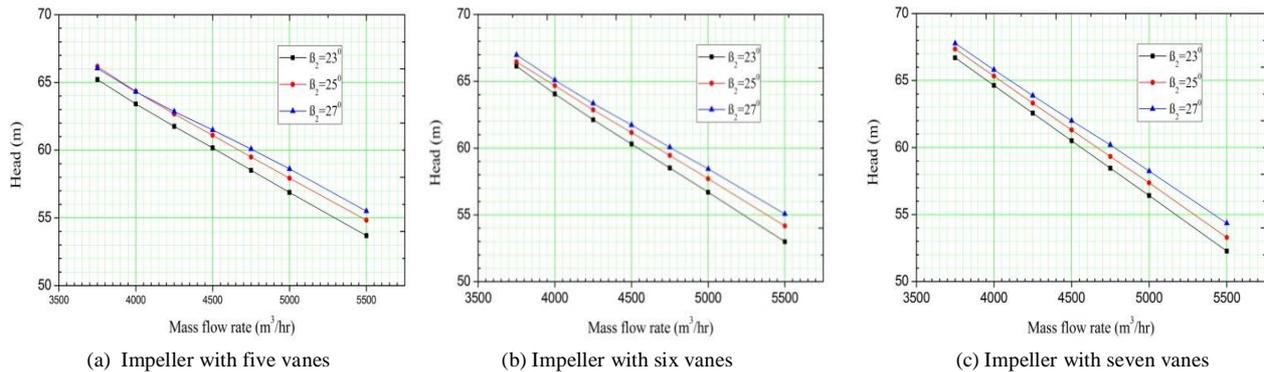


Fig.4. Head versus mass flow

3.2 Effect of vane outlet angle on Efficiency

The efficiency (97.6 %) at design point for the impeller with 7 vanes and 25° vane outlet angle is maximum. The efficiency at duty point is less than this for impeller with vane outlet angles 23° and 27°. The variation of efficiency at different flow rates for impellers with five, six and seven vanes are presented in Fig.5 (a), Fig.5 (b) and Fig.5 (c) respectively.

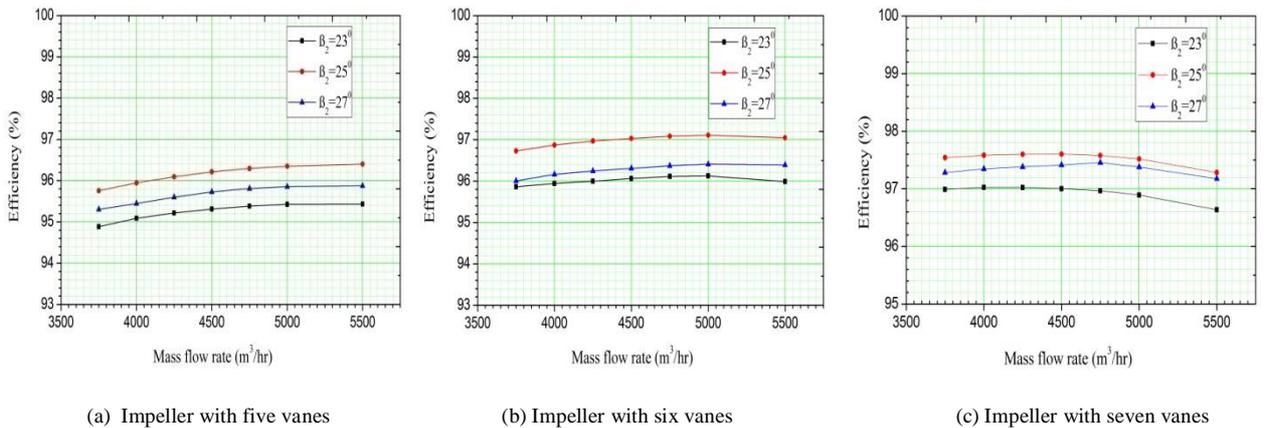


Fig.5. Efficiency versus mass flow

3.3 Effect of number of vanes on head

The values of head at flow rate of 4750 m³/hr for different exit vane angles have insignificant deviation. The decrease in head developed is observed as the mass flow rate increases from that point. The head increases with vane number as the mass flow reduced below the design point. The total deviation of head at design point is 0.35 m, 0.22 m and 0.51 m from Fig. 6(a), Fig. 6(b) and Fig.6(c) respectively.

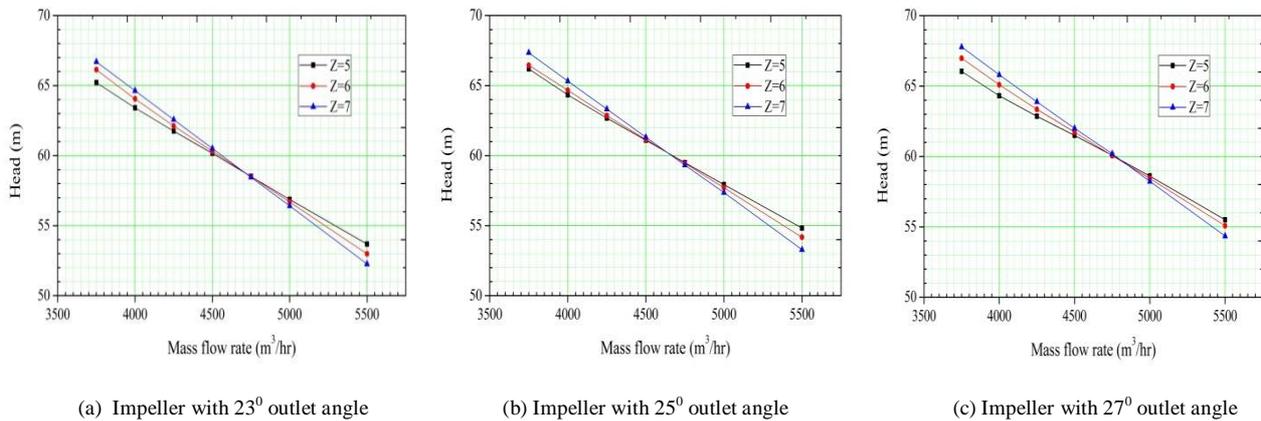


Fig.6. Head versus mass flow

3.4 Effect of number of vanes on efficiency

The maximum efficiency at design point attained is 97 % for impeller with 23° vane outlet angle and seven vanes. It decreases by 1.7 points for impeller with 5 vanes. From Fig.7 (a), Fig.7 (b) and Fig.7(c), it is inferred that the efficiency decreases with decrease in vane number for the range of number of vanes considered for analysis. The BEP is at design point only for the impeller with 7 vanes and 25° outlet angle (Fig. 4.7(b)).

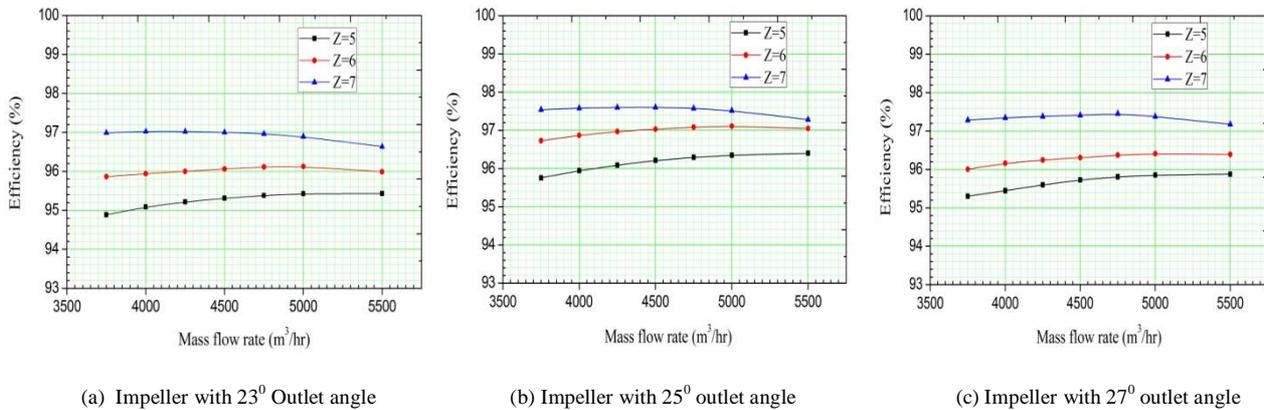


Fig.7. Efficiency versus mass flow

The performance curve for impeller configuration 8 is illustrated in Fig. 8. Results show that the best efficiency for this impeller is at designed flow rate. The pressure and velocity distributions at meridional section for this impeller are shown in Fig.9 (a) and Fig.9 (b) respectively.

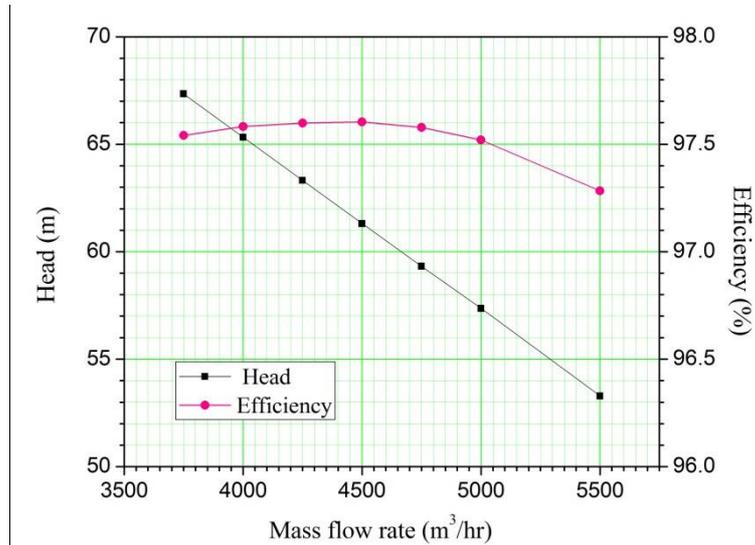


Fig.8. Performance curve for the impeller having 7 vanes and 25° outlet angle

Pressure increases gradually from inlet to outlet. Velocity decreases along the meridional section from inlet to outlet. Also the maximum velocity is observed at shroud and the minimum velocity at the hub where the flow direction changes from axial to radial. No voids and flow separation and abrupt changes in flow pattern are observed.

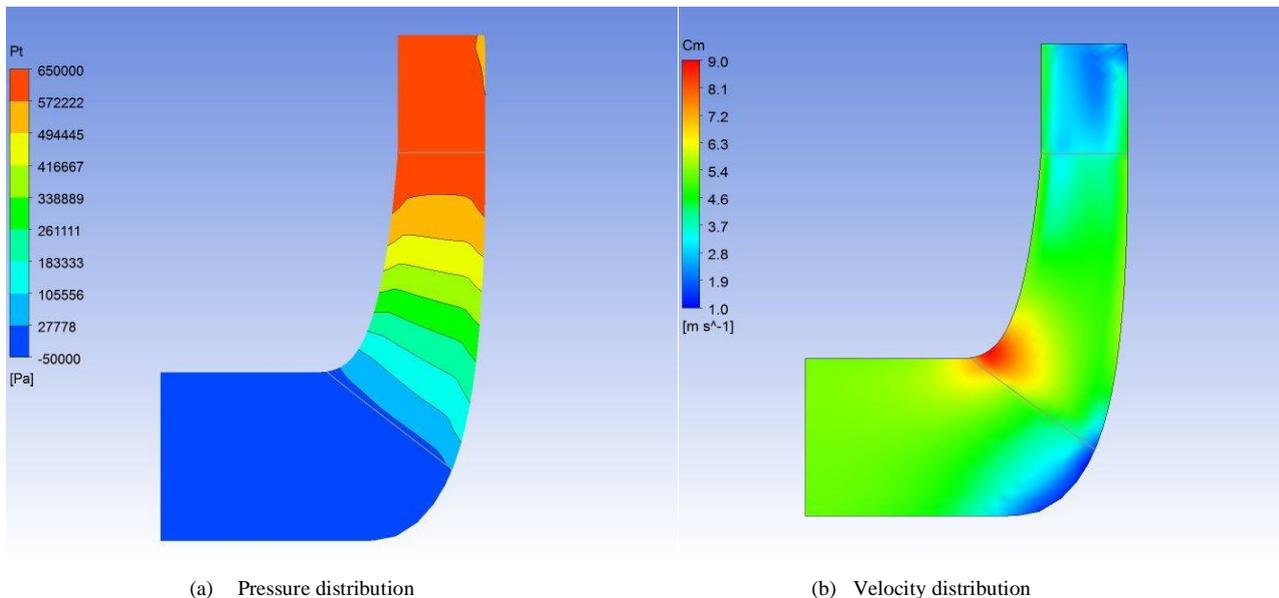


Fig.9. Pressure and Velocity distribution along meridional section

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 7, July 2016

IV. CONCLUSIONS

At design point, head increases with increase in the outlet blade angle from 23° to 27° . Efficiency at design point is maximum for impeller with 25° vane outlet angle having seven vanes. Both efficiency and head increases as the number of blades increases from five to seven. The best efficiency point of the base impeller is off the design point.

REFERENCES

- [1] Sedlar M and Mensik P, "Investigation of rotor/stator interaction influence on flow fields in radial flow pumps", Proc.3rd European conference on Turbomachinery, 1999, IMechE, pg.1017-1025.
- [2] E.C. Bacharoudis et.al "Parametric study of a Centrifugal pump Impeller by Varying the Outlet blade Angle", The open Mechanical Engineering Journal, 2008, Volume2, pg. 75-83.
- [3] Val. S. Lobanoff, Robert R. Ross, "Centrifugal pumps Design and application", second edition, A practical preference stressing hydraulic design, 1985, pg 28-36.
- [4] Strycek. O "Centrifugal Pumps, Hydrodynamicke cerpadla", 2.Vydanie, Vyadavatel'stvo STU, Bratislava 1994. (in Slovak)
- [5] Paciga, Strycek, Ganco, "Pumping EquipmentCerpacia Technika, Sjf STU, Bratislava 1984. (in Slovak)
- [6] J. F. Guilich, "Centrifugal pumps", second edition, Springer, ISBN 978-3-642-12823-3, 2008.
- [7] HLbocan, P, "Hydraulic design of the impeller in 3D environment". The thesis.Hydrallycky navrh obezneho kola v 3D prostredi. Bratislava; Sjf STU, Diplomava praca.2010. (in Slovak)
- [8] HLbocan, P, Varchola M "Hydraulic design of a mixed flow impeller in 3D environment",2010.
- [9] S.cao, G.Peng and Z. Yu, "Hydrodynamic design of rotodynamic pump impeller for multiphase by combined approach of inverse design and CFD analysis", Journal of Fluids Engineering, 127 (2005), pg. 330-338.
- [10] M. Asuaje, F. Bakir, S. Kouidri and R. Rey, "Inverse design method for centrifugal impellers and comparison with numerical simulation tools", International Journal for Computational Fluid Dynamics, 18 (2), 2004, pg.101-110.
- [11] A. J. Stepanoff, Centrifugal and axial flow pumps: theory, design and application, John Wiley Sons Inc. 1986.
- [12] Joon-Hyung Kim et. al, "Design techniques to improve the performance of a centrifugal pump using CFD", Journal of Mechanical Science and Technology, 2014, pg 215-225.