Design and Analysis of Low Specific Speed Centrifugal Pump Impeller Passage

Mr.Nilesh A.Patil[1], Mr.Nilesh M.Shinde[2], Mr. Rajendra A Patil[3]
Department of MechanicalEngineering [1,2,3]
R. C. Patel Institute of Technology, Shirpur
Dist.Dhule, 425405, India
nileshpatil4089@gmail.com [1]
nmshinde@ymail.com [2]
raj.mech3987@gmail.com [3]

Abstract—Recently, a divergent pump in the scope of low particular pace, for example, ns 0.25, pulls in consideration as a substitute for positive uprooting pump in light of the fact that of vibration and commotion issue as well as because of late request towards little size and high rotational velocity it is currently unavoidable to build up another outward pump with superior in low particular rate range. This paper manages outline and stream investigation of low, particular velocity radiating pump impeller section. The streams in turbo-machines are extremely basic wonders. The paper depicts the stream examination of D and E sort impeller utilizing ordinarily accessible code. The outcomes were investigated for speed dissemination, weight conveyance on impeller cutting edges of D and E sort. As the spillage stream diminishes outright tangential speed of the liquid, the stream is diverted toward the impeller hub as an aftereffect of decline in radial power.

Keywords—low particular rate pump, numerical arrangement

*****

I. INTRODUCTION

A diffusive pump the particular velocity is under 0.25 then it is low particular pace pump. Pump is a mechanical gadget to build weight vitality of fluid. In the greater part of the cases pump is utilized for raising liquids from darling to more elevated amount. This is because of weight distinction in bay and outlet. Low weight at channel and high weight at outlet or conveyance end. Its motivation is to change over vitality of a prime mover (an electric engine or turbine) first into speed or dynamic vitality and afterward into weight vitality of a liquid that is being pumped. The vitality changes happen by excellence of two primary parts of the pump, the impeller and the volute or diffuser. The impeller is the turning part that proselytes driver vitality into the dynamic vitality. The volute or diffuser is the stationary part that changes over the active vitality into weight vitality.

Table 1 Dimension of pump [1]

Flow Rate	$Q\,(m^3/\text{sec})$	$3.75 * 10^{-4}$
Head	H (m)	1.1
Pump speed	n (rpm)	700
Specific Speed	n_s	0.24

A. Design of low specific speed pump

Impeller is designed on the basic of design flow rate, pump head, rotation speed and pump specific speed. For design calculation, the design parameters are taken as follows:

B. Design of impeller

Determination of the geometrical features of the impeller is generally accomplished in the following order: a) The "eye" radius r_e b) The exit radius r_2 or r_{t2}, and c) The exit width b_2 or, in the case of mixed- and axial-flow impellers, the hub exit radius r_{h2} all of which form the starting point for d) Shaping the hub and shroud profiles figure and finally e) Construction of the blades.

The eye: The inlet radius r_e can be found from the following formulas:

$$r_e = \left[\frac{Q}{\pi \Omega \phi_e \left(1 - \frac{r_s^2}{r_e^2}\right)} \right]^{\frac{1}{3}} \tag{1}$$

$$\Omega_{ss} = \frac{\sqrt{\pi \phi_e \left(1 - \frac{r_s^2}{r_e^2}\right)}}{\left(\tau/2\right)^{0.75}} \tag{2}$$

The exit radius r_2: This is found from head-coefficient ψ by means of the equation for r_2

$$\psi = \frac{g \Delta H}{\Omega^2 r_2^2} \tag{3}$$

The exit width b_2: The exit width can be calculated from following equation

$$b_2 = \frac{Q}{2\pi \Omega r_2^2 \phi_i \varepsilon} \tag{4}$$

Hub and shroud profiles: With the eye and the outlet sizing established, the two are connected by specifying the hub and

ISSN: 2321-8169
54 - 57

shroud profiles. The procedure is self explanatory an accepted geometry can be achieved by following these guidelines:

Maintaining the meridional flow area $2\pi r_{b1} b_1$ at the blade leading edge at about the same as it is at the eye, namely $\pi\left(r_e^2 - r_s^2\right)$, but then gradually increasing it versus meridional distance to the generally larger value already established at the exit, namely $2\pi r_2 b_2$.

Table 2 Dimension of impeller D and E

Test impeller	D	E
Type	Semi-open	Semi-open
Z	6	4
t(mm)	4	2.37
b_1 (mm)	14.4	14.4
b_2 (mm)	3.24	3.24
c(mm)	1	1
c/b_2	0.125	0.125
$r_2(mm)$	59	59
β_1 (deg)	90	17.32
β_2 (deg)	90	30
N_s	0.24	0.24

Table 3 Development of the Hub and shroud profile

Serial No.	Radial distance from axis of rotation to centre of circle defining impeller passage width, r_b (mm)	Width of an impeller or other blade passage in the meridional plane, b(mm)
1	10.8	14.4
2	16.74	10.83
3	21.48	9.11
4	24.09	8.04
5	27.81	7

II. INTRODUCTION OF CFD:

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow (in a special form) over a region of interest, with specified (known) conditions on the boundary of that region.The approximation of a continuously varying quantity in terms of values at a finite number of points is Called discretization. The fundamental elements of any CFD simulations are

1. The fluid continuum is discretised; i.e. field variables (u, v, w, p...) are approximated by their values at a finite number of nodes.

2. The equations of motion are discretised; i.e. approximated in terms of the values at the nodes: differential/integral equations (continuum)=algebraic equations(discrete)

3. The system of algebraic equations is solved to give the values of all variables at the nodes.

III. MODELING AND PHYSICS DEFINATION

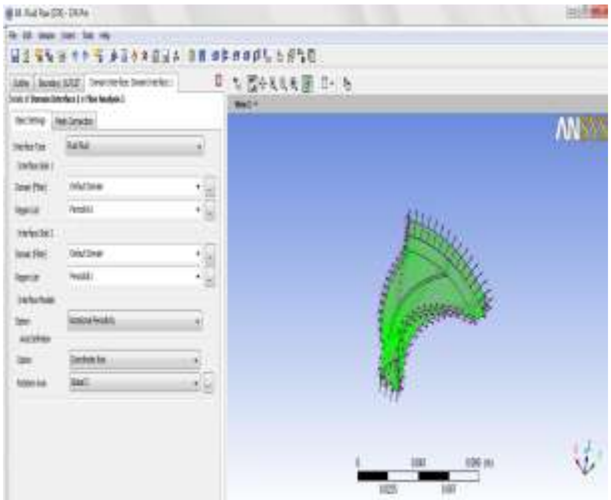

Fig.1 CFX-PRE USER INTERFACE

After the compilation of modeling called the object in cfx-pre here we have to define a physics of a problem like boundary condition (reference pressure and temperature, domain type, interface type, turbulence model), property of flow etc. For particular this analysis the rotation speed is 750 rpm and 100000 Pa inlet pressure.

IV. RESULT OF FLUID FLOW ON IMPELLER BLADES IN CFX POST

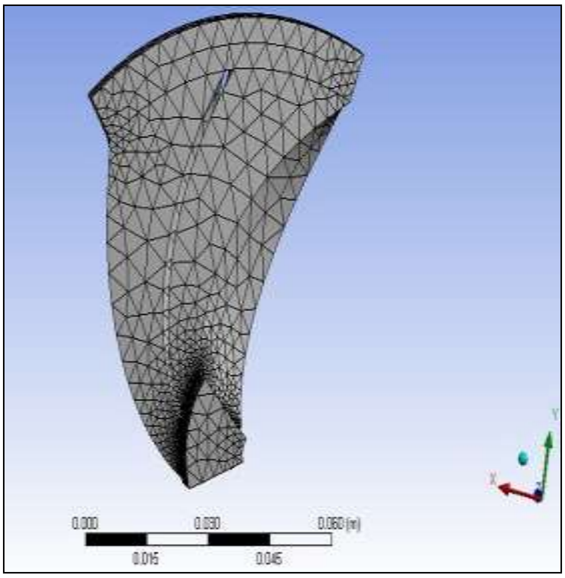

Fig.2 Mesh of blade for impeller E

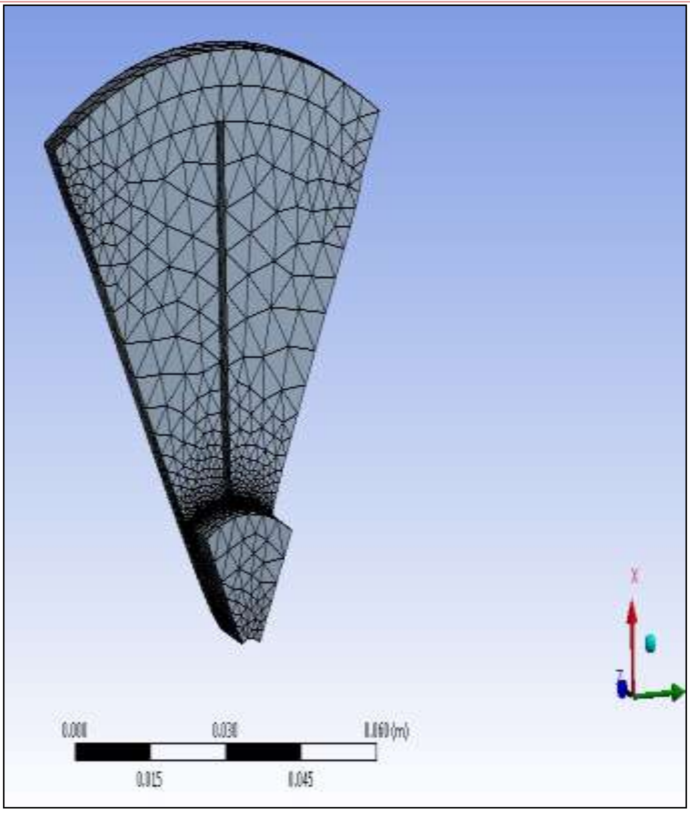

Fig.3 Mesh of blade for impeller D

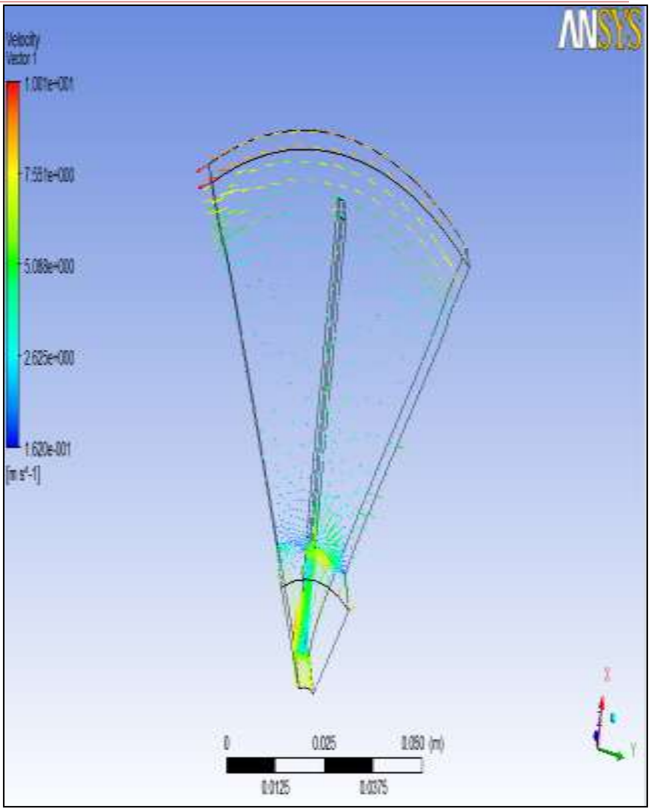

Fig.5 Velocity distribution along the blade impeller D

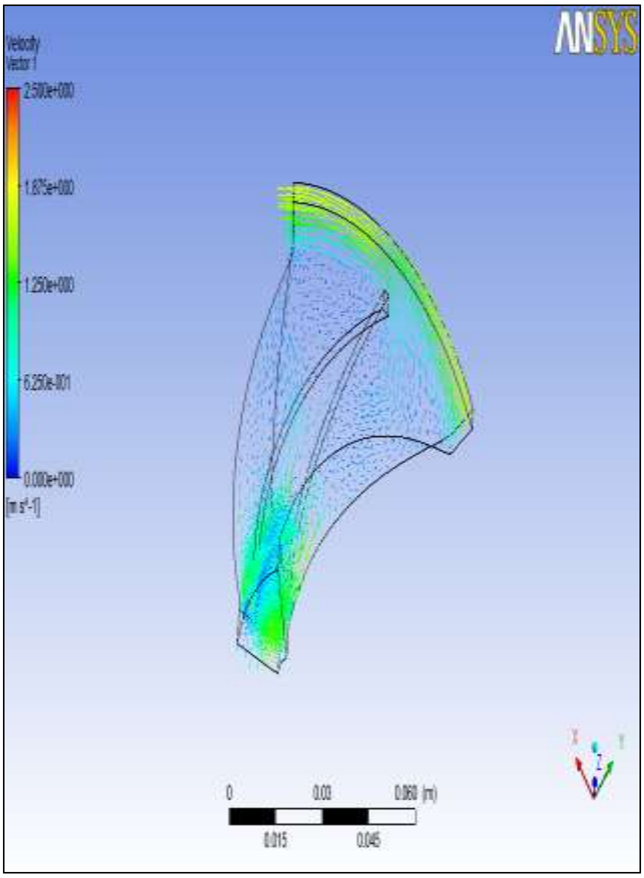

Fig.4 Velocity distribution along the blade impeller E

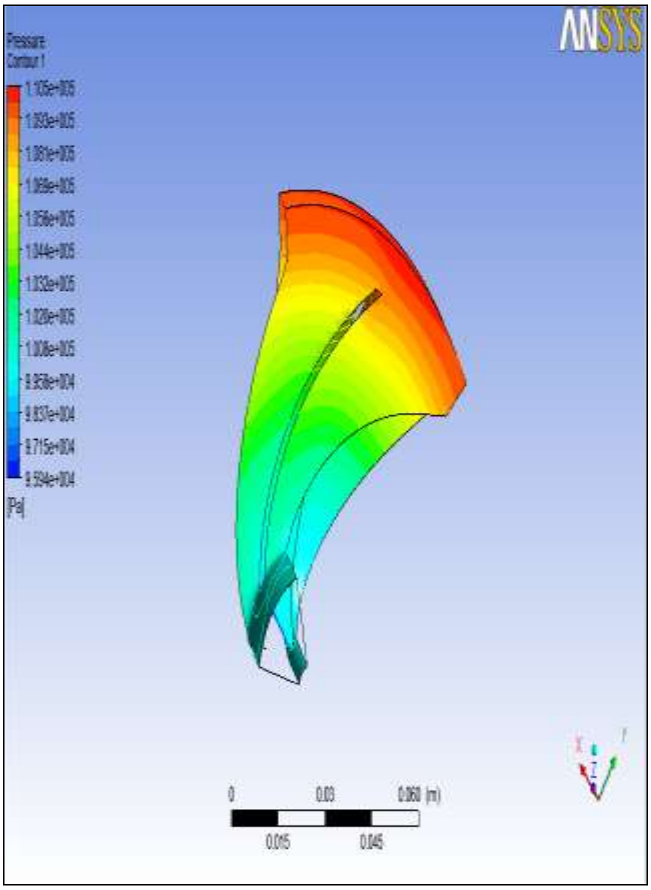

Fig.6 Pressure distribution along the blade for impeller E

.

ISSN: 2321-8169
54 - 57

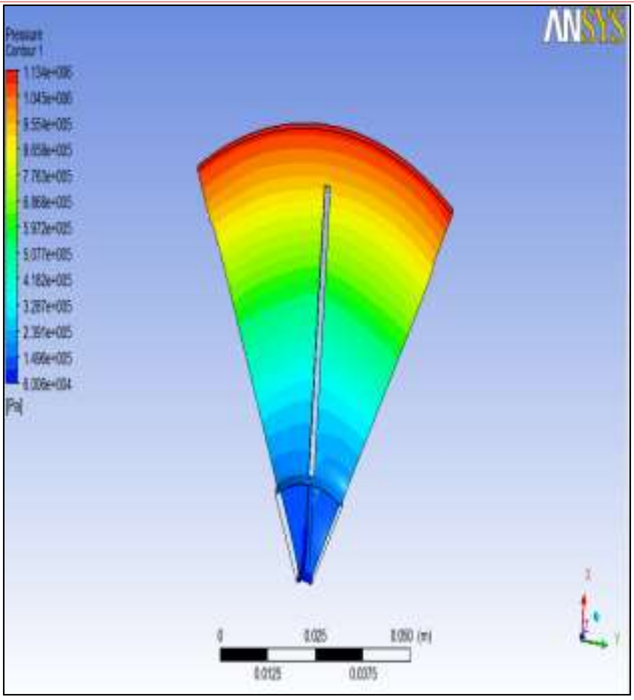

Fig.7 Pressure distribution along the blade for impeller D

V. RESULT AND DISCUSSION

The flow analysis is carried out from post processing results in CFX Post. Velocity and pressure results are discussed as bellows

1. Maximum velocity is observed at the inlet portion of the blade E and D as shown in fig 4 and fig 5.

2. Maximum pressure is observed at pressure side of blade and minimum pressure is observed at suction side of the blade as shown in fig 6 and fig 7.

3. The relative flow of semi-open impeller D shows that through-flow along the confined narrow region of the blade suction side goes out to the impeller outlet but the entire flow direction in the impeller passage deflects towards the blade pressure side from the blade suction side and low velocity region seen in impeller passage.

4. As the leakage flow reduces absolute tangential velocity of the fluid, the flow is deflected toward the impeller axis as a result of decrease in centrifugal force.

References

[1] Junichi Kurokawa,Jun Matsui Young-Do Choi1, "Performance and Internal Flow characteristics of a very low specific speed centrifugal pump," Journal of Fluids Engineering, vol. 128, pp. 341-349, march 2006.

[2] W Liu1, W Jian1 and P Wei 2 H Chen1, "Impellers of low specific speed centrifugal pump based on," IOP Publishing, 2010.

[3] WANG Tong1, GU ChuanGang1 & SHU XinWei1,2 ZHANG Bin1*, "Blade optimization design and performance investigations of an ultra-low specific speed centrifugal blower," Technological Sciences, vol. 54, pp. 203–210, January 2011.

[4] WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai LIU Houlin*, "Effects of Blade Number on Characteristics of Centrifugal Pumps," CHINESE JOURNAL OF MECHANICAL ENGINEERING, vol. 23, 2010.

[5] "Fluid flow in a rotating low specific speed centrifugal impeller passage," Fluid Dynamics Research, vol. 24, pp. 275-292, july 1999.

[6] Yue LI Wei-Dong CAO Yun LIANG Xiao-Di ZHANG Yi GAO, "Hydraulic Optimization of Super-low Specific Speed Multistage Flameproof Submersible Pump with Inner Motor for Coal Mine," Third International Conference on Information and Computing, 2010.

[7] "Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics," International Journal of Rotating Machinery, vol. 9, pp. 49-61, 2003.

[8] M G Rose, "Low flowrate effects in a centrifugal pump impeller," Journal of Power and energy, vol. 218, pp. 417-427, april 2004.

[9] H W Oh and M K Chung, "Optimum values of design variables versus specific speed of centrifugal pump," Proc Instn Mech Engrs, vol. 213, pp. 219-226, february 1999.

[10] Jianjun Feng Friedrich-Karl Benra Hans Josef Dohmen, "Investigation of Periodically Unsteady Flow in a Radial Pumpby CFD Simulations and LDV measurement," Journal of Turbomachinery, vol. 133, pp. 1-10, january 2011.