Int. J. Chem. Sci.: 14(2), 2016, 1148-1156
ISSN 0972-768X
www.sadgurupublications.com

NUMERICAL ANALYSIS OF CENTRIFUGAL PUMP IMPELLER FOR PERFORMANCE IMPROVEMENT

S. KALIAPPAN, M. D. RAJKAMAL and D. BALAMURALI

Department of Mechanical Engineering, Velammal Institute of Technology, CHENNAI – 601204 (T.N.) INDIA

ABSTRACT

The impeller of an existing Mather & Platt centrifugal pump with both geometry and performance known was analyzed and redesigned using an integrated. The purpose of the redesign was to achieve improved impeller performance at the duty point. Fluid dynamics and geometry modeling parts of the design /analysis system were systematically applied. To analyze the existing impeller say impeller A, which was designed using conventional (routine in industry) hydraulic layout procedures. To develop a new impeller say impeller B, using a coupled, multilevel design and optimization techniques like impeller redesign. The analysis results are presented for impellers A in this Paper and Comparisons of the CFD results for both impellers reveal internal flow features that explain the improved impeller B performance levels in Paper. Computational Fluid Dynamics (CFD) is a tool which aids in conducting or simulating flow through a pump virtually using computer software. An attempt has been made in this paper to improve the performance of a typical centrifugal pump impeller.

Key words: CFD, Centrifugal pump, Impeller.

INTRODUCTION

The subject impeller of this paper is from a high specific speed centrifugal pump stage which was designed by Mather & Platt Company. The data which support the work described herein were derived from and were used to reconstruct the key balding features of the tested impeller. Digitally reconstructed baseline impeller is referred to as impeller A. The new impeller, whose geometry was generated by the application of the agile system, is referred to as impeller B, which could serve as a replacement for impeller A. Within the redesign process, two steps were undertaken in order to improve the baseline pump impeller A. To analyze the performance of the baseline impeller A using the multilevel analysis tools available within the Agile system, which includes 1D, Q3D, and fully 3D, Navier-Stokes

*Author for correspondence; E-mail: kaliappa@yahoo.com, kamalaerodynamics@gmail.com, balamurali.srs@gmail.com

solvers. To redesign impeller A to yield higher efficiency, using the same Agile tools, in particular its 3D geometry definition and CFD performance prediction codes. The results presented are limited to the BEP operation, at which impeller A was analyzed, and for which impeller B was developed. However, for both impellers, flow calculations were conducted at the part load operation, but these will be reported separately. Future work on this case will include the instability analysis. No attempt was made at this point to include the instability effects in this analysis due to the lack of information available in the public domain for the pump A spiral volute geometry. Detailed investigations at part load operation will require coupled impeller-volute CFD computations.

Introduction to CFD

CFD is a technique of replacing Partial Differential Equations governing the fluid flow by a set of algebraic equations and solving them using digital computer. Any fluid flow in the universe is governed by a set of equations. The equations are the governing equations,

- Continuity equation
- Momentum equation
- Energy equation

Governing equations in partial differential form.

Continuity equation

The continuity equation in algebraic form is given as –

$$\rho 1\ A1V1 = \rho 2\ A2V2$$

The continuity equation in integral form is given as –

$$\partial\partial t\ \rho dv\ v + \rho V.dS\ s = 0$$

The continuity equation in Partial Differential Form is given as –

$$\partial\rho\partial t + \nabla.\ \rho V = 0$$

Momentum equation

$$\partial\partial t\ \rho \mathrm{V}\ dV\ v + \rho \mathrm{V.dS\ V}s = -\ p\ \mathrm{dS}\ s + \rho f dV\ v + \mathrm{F\ viscous}$$

Energy equation

$$v + \mathrm{Q\ viscous} - p\mathrm{V.dS}\ s + \rho\ \mathrm{f.\ V}\ dV\ v + \mathrm{W\ viscous} = \partial\partial t\ \rho\ e + \mathrm{V22}\ vdV$$
$$+ \rho\ e + V22\ s\mathrm{V.dS}$$

The agile engineering design system

An overview of the Agile Engineering Design System, adopted from Fig. 1. The underlying concept of the agile system is that it consists in the computer implementation of the design and analysis tools which organize and execute the integrated and automated multilevel (1D-Q3D-3D) approach, and allow their use either in the structured way, as shown in Figure 1 (i.e., from the 1D, through the Q3D to the fully 3D design and analysis), or permit their use in a more flexible fashion.

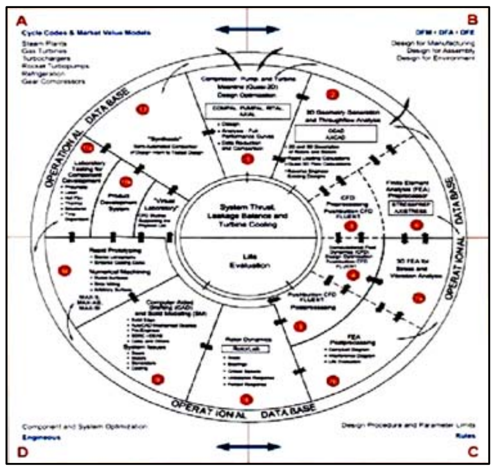

Fig. 1: Agile engineering design system

Baseline impeller geometry: 1d & 3d design models

Duty point parameters of the research pump under consideration are: Flow rate Q = 0.196 m^3/s, Head = 5.8 m, Rotational speed = 750 rpm. The baseline impeller geometry was derived from the data and the impeller pattern maker drawings published in Table 1.

Table 1

Parameter	Units	Impeller A Inlet	Impeller A Outlet	Impeller B Inlet	Impeller B Outlet
R - at the shroud	mm	135	162	128	161.5
R - at the hub	mm	43.3	162	43.3	148.5
Beta, at the shroud	deg	20	27	25	36
Beta, at the hub	deg	60	33	50	36

Cont…

Parameter	Units	Impeller A Inlet	Impeller A Outlet	Impeller B Inlet	Impeller B Outlet
B, imp. width	mm	—	87	—	74
Lax, imp. axial length	mm	—	167	0	159
LE inclination angle	deg	45	—	54.5	—
TE inclination angle	deg	—	90	—	80
Blade number	—	7	7	6	6

These data were used to create: The one dimensional model for impeller A t the three-dimensional digital model for impeller A. The baseline impeller shown with the pattern maker blade sections created in the 3D digital model is presented in Fig. 2 The three-dimensional image of the digital model of impeller A is displayed in Fig. 3

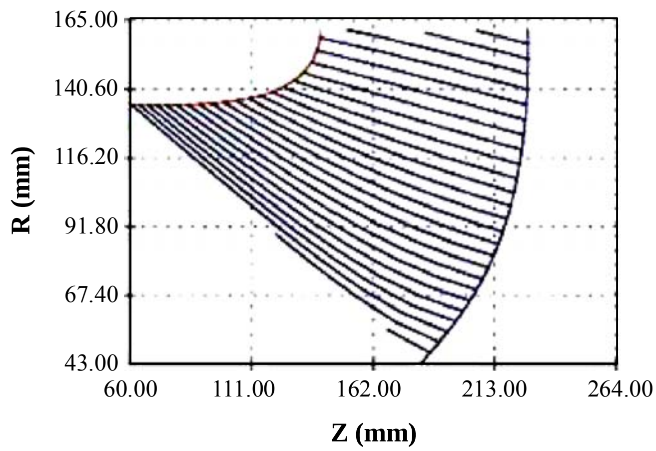

Fig. 2: Impeller A blade cross-sections

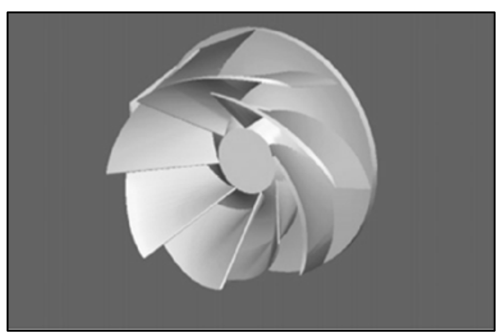

Fig. 3: Impeller A 3 D model

Viscous flow evaluations for Impeller A

One dimensional analysis of Impeller A

The set-up of the 1D model for Impeller A analysis consisted of two parts: geometrical modeling fluid dynamics modeling. Geometrical modeling required the input of the Table 2 data into the one dimensional Code, PUMPAL which was run in the analysis mode in order to generate the LE and TE velocity triangles, loss levels and impeller and stage efficiency predictions. Fluid dynamic modeling consisted of selecting design parameter levels.

Table 2

Parameter	Units	Impeller A Inlet	Impeller A Outlet	Impeller B Inlet	Impeller B Outlet
Flow departure at the tip	Deg	1.69 inc	—	2.64 inc	—
Flow departure at the RMS	Deg	14.4 inc	13 (dev angle)	3.39 inc	13.64 (dev angle)
Flow departure at the hub	Deg	26.5 inc	—	4.85 inc	—
E (sec. zone/total area)	—	—	0.426	—	0.28
Slip Factor (US definition)	—	—	0.731	—	0.76
Internal impeller loss	—	—	0.077	—	0.073
Impeller exit mixing loss	—	—	0.055	—	0.04
Rotor T-T efficiency	—	—	0.83	—	0.87
Stage T-T efficiency	—	—	5.8	—	5.8

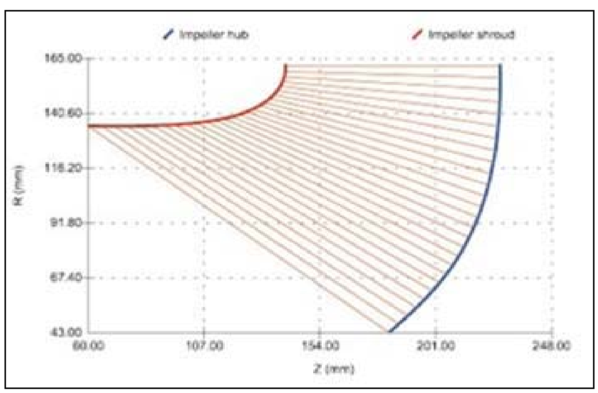

Fig. 4: Meridional profile of impeller A with meshing for Q3D analysis

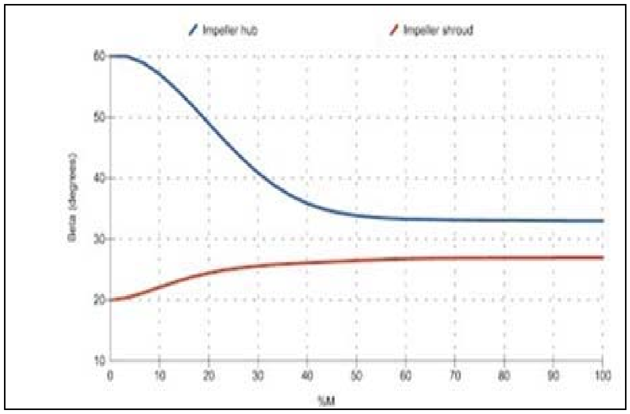

Fig. 5: Blade angles distributions along hub and shroud for impeller A

Impeller A - 3d geometry model and Q3d flow analyses

The 3D impeller geometric model was created in the CCAD code by inputting the number of blades, the definition of the hub and shroud contours, and the associated hub and shroud blade normal thickness and wraps angle distributions. Appropriate curve fitting procedures (in CCAD) were employed to represent the hub and shroud contours and thickness distributions and to deduce the hub and shroud blade angle Distributions. Geometry and fluid dynamics features available in CCAD were used to analyze impeller A properties, such as the meridional channel and the blade passage areas, as well as blade loading. Q3D in viscid flow analysis was performed to derive

Information on blade loading for the existing impeller

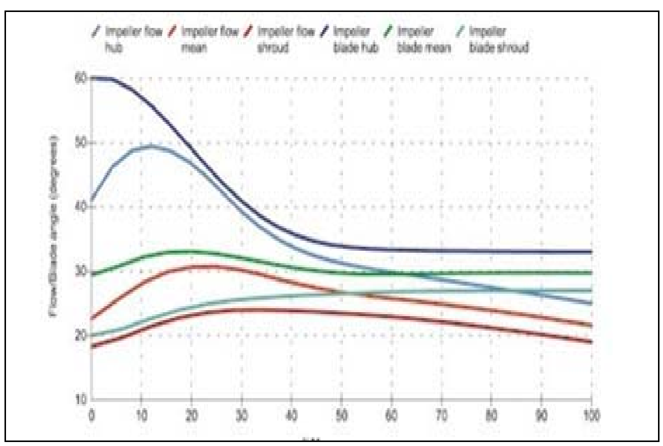

Fig. 6: Flow and blade angles distributions along three blade

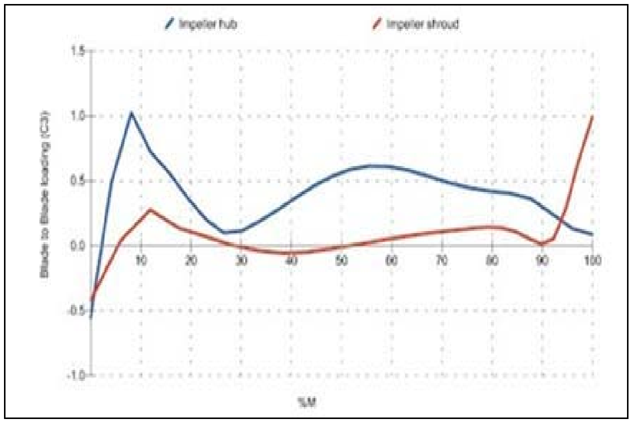

Fig. 7: Impeller A blade loading predicted from simplified Q3D procedure

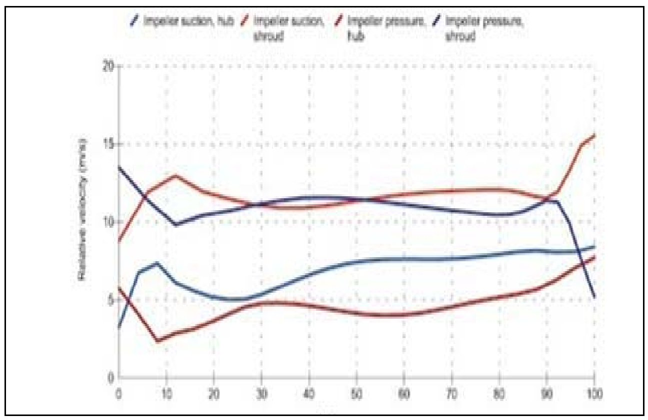

Fig. 8: Impeller A relative velocities distributions from simplified Q3D

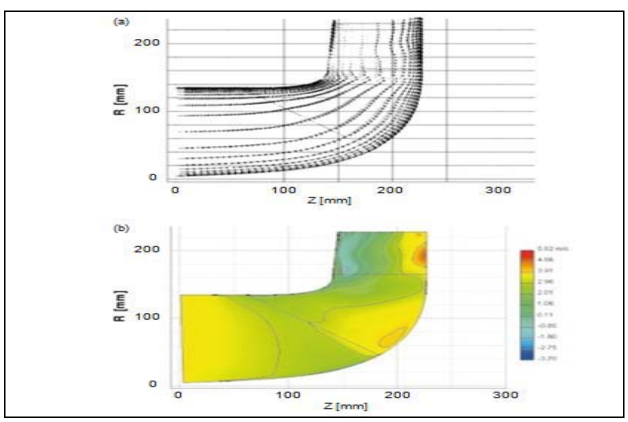

Fig. 9: Impeller A meridional velocity vectors at mid-passage

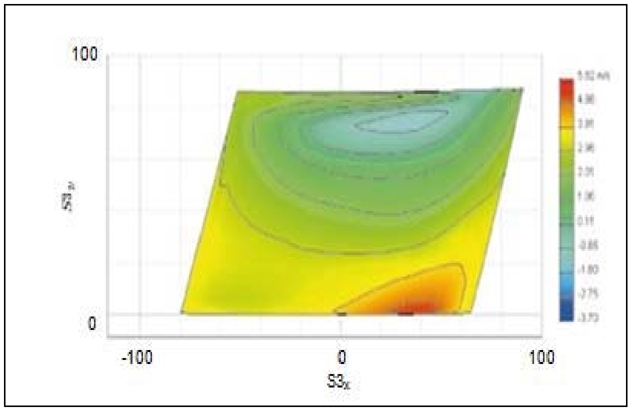

Fig. 10: Impeller A meridional velocity distribution and velocity distribution at mid-passage immediately after TE

CONCLUSION

Flow evaluations

The three-dimensional analyses of impeller A indicated a possibility for improving the hydraulic layout of this impeller, primarily by providing better flow guidance in the passage. The purpose of the redesign effort was to increase the impeller efficiency by reducing both the hub-to-shroud cross flow, and the tendency for secondary flow generation, as well as the reduction of the backflow region at the outlet of the impeller. Design modifications to achieve these goals included: The modification of impeller A's meridional contours more appropriate for its specific speed (i.e., by introducing more of a mixed-flow impeller shape via changes in the positions of the LE and TE).

These design modifications are aimed at reducing axial to radial turning and the associated passage curvatures in the meridional plane The redesign of the impeller blade angles to reduce excessive incidence angles along the leading edge of impeller A. The redesign of the blade angle distributions between leading and trailing edges of the blade. Experience with impeller designs within this range of specific speeds indicates a possibility for a decrease in blade number from 7 to 6, as well the reduction of the impeller inlet and exit widths. This was carried out along with a reduction of the impeller inlet diameter and an increase in impeller back sweep. These redesign steps, combined with the above mentioned three modifications, were aimed at decreasing the impeller internal losses by the reduction of the secondary flows as well as the size and the location of the wake regions in the impeller passages.

REFERENCES

1. A. E. A. Aqeel, Technology Engineering Software Limited CFX TASC flow Computational Fluid Dynamics Software: Primer Manual, Version 2.10 (2000).
2. J. Wang, Piechnajanusz et al. A novel design of composite water turbine using CFD, J. Hydrodynamics, **24(1)**, 11-16 (2012).
3. K. Srinivasan. Rotodynamic pumps (centrifugal and axial), New age international (2008).
4. R. I. Lewis, Turbomachinery Performance Analysis, John Wiley, New York (1996).
5. J. M. Lighthill, A new method of two -dimensional aerodynamic design (1945).
6. S. Rajendran and K. Purushothaman, Analysis of a centrifugal pump impeller using ANSYS- CFX, Int. J. Engg. Res. Technol., **1(3)** (2012).
7. M. B. Flathers and G. E. Bache, Aerodynamically induced radial forces in a centrifugal gas compressor, J. Engg. Gas Turbines & Power (1999).
8. R. K. Byskov and C. B. Jacobsen, Flow in a centrifugal pump impeller at design and off-design conditions-Part II: Large eddy simulations, J. Fluids Engg. (2003).
9. W. N. Dawes, P. C. Dhanasekaran, W. P. Kellar and A. M. Savill, Reducing bottlenecks in the CAD-to-Mesh-to-Solution cycle time to allow CFD to participate in design, J. Turbo Machinery (2001).
10. S. L. Chen and W. T. Wang, Computer aided manufacturing technologies for centrifugal compressor impellers, J. Mater. Process. Technol. (2001).

Accepted : 20.05.2016