Design and analysis of a radial centrifugal pump for cryogenic helium based application

J Das1,2, V J Lakhera1 and B Sarkar2

1Mechanical Engineering Department, Nirma University, Ahmedabad, India
2ITER-India, Institute for Plasma Research, Ahmedabad, Gujarat, India

Email: 15mmrerex01@nirmauni.ac.in

Abstract. A centrifugal pump is a rotordynamic machine that has wide spread application in industry. After analytical design, it is important to check the machine performance by some numerical method to be assured of machine performance before manufacturing. In the present study, a centrifugal pump conforming to a specific process requirement for cryogenic helium fluid has been developed in detail. This involves analytical design derivation of the stator and rotor part, computer aided design (CAD) modelling, meshing and detailed computational dynamic analysis (CFD) analysis. In line with analytical prediction, it is seen from simulation results that the designed machine meets the given requirements with an achieved hydraulic efficiency of 88%.

1. Introduction

A centrifugal pump is a rotordynamic machine used for transport of incompressible process fluids by the conversion of rotational kinetic energy (through work input) to hydrodynamic energy (e.g. potential head) of the fluid flow, for some nominal design condition. A single stage pump typically constitutes of a rotor (impeller wheel) and stator (volute passage) section. The impeller wheel basically imparts potential head to fluid flow. Volute flow passage guides the impeller exit flow to pump outlet while contributing to static pressure recovery as well. When required, for additional pressure head generation, diffuser row with static vane row is also used [1].

In the present study, a pump with nominal process specification has been developed while targeting a hydraulic efficiency of 85% for cryogenic helium fluid based application for end client. Literature review suggests that full emission type centrifugal impeller designs are preferred over partial emission counterparts, where in addition to pressure head, higher hydraulic efficiency is also desired [2,3]. In this regard, an impeller wheel (for full emission type pump) with single curvature blading has been analytically developed. From this the computer aided design (CAD) model has been derived and supporting computational fluid dynamics (CFD) analysis has been done to compare the simulation results with projected analytical performance.
2. Methodology

Figure 1 above summarizes the approach adopted for deriving the hydrodynamic design of the pump system. Initially based on defined process specification type of pump (whether reciprocating, radial or axial type) is chosen based on Head (H) vs flow (Q) curves [3]. Then based on Stepanoff’s approach [4], specific speed of the system is worked out to get the basic wheel dimensions, namely inlet and outlet wheel diameter (D₁, D₂), and corresponding blade height (b₁, b₂) along with number of blades (Z). Point by Point (PbP) [5] method has been used to derive the single curvature blading for this design. Next a volute with trapezoidal cross section based on conservation of angular momentum approach has been developed considering favourable performance at moderate specific speeds. The developed impeller wheel and volute are meshed appropriately and assembled together (while checking for rotational direction) for detailed Computational Fluid Dynamics (CFD) investigation, using appropriate boundary conditions and turbulence models.

3. Analytical Calculation

To achieve the design specification as mentioned in section 1, the analytical design of the impellers wheels and volute passage has been derived based on approach shown in Figure1 with corresponding equations and governing relations. Fluid under consideration is considered to be incompressible and its density (ρ) is taken as 139.87 kg/m³ from HEPAK® Excel add in [6].

3.1. Rotor Design

| S No. | Design parameter               | Abb. | Unit | Value |
|-------|--------------------------------|------|------|-------|
| 1.    | Number of full length blades   | Z    |      | 9     |
| 2.    | Inlet diameter                 | D₁   | mm   | 51    |
| 3.    | Outlet diameter                | D₂   | mm   | 84    |
| 4.    | Blade inlet angle              | β₁   | deg. | 17.32 |
| 5.    | Blade outlet angle             | β₂   | deg. | 22.50 |
| 6.    | Inlet blade height             | b₁   | mm   | 16.23 |
| 7.    | Outlet blade height            | b₂   | mm   | 11.32 |
| 8.    | Avg. blade thickness           | δ     | mm   | 1.2   |

Figure 1: Design methodology for the cryogenic centrifugal pump
Table 1 above summarizes the impeller wheel dimension. Basic sizing reveals that the impeller comes out to be a radial wheel with a specific speed ($N_S$) of around 46 (mks units) and a nominal rotational speed of 13,000 rpm (revolutions per minute).

**Table 2: Blade profile points of the impeller wheel**

| S No. | Radius ($r_n$) [mm] | Angle ($\theta_n$) [deg.] |
|-------|---------------------|--------------------------|
| P1    | 25.5                | 0.0                      |
| P2    | 28.8                | 21.5                     |
| P3    | 32.1                | 38.9                     |
| P4    | 35.5                | 52.5                     |
| P5    | 38.7                | 62.6                     |
| P6    | 42                  | 73.0                     |

Table 2 above summarizes the single curvature blade profile obtained with Point by Point (PbP) method. On the profile guide curve, average thickness is assigned symmetrically. Then this closed sketch is extruded to generate the required blade shape. This single blade is symmetrically patterned around central axis to obtain the desired number of blades, i.e. 9 in this case.

3.2. Stator Design

The volute flow passage is an important element of pump design to achieve high pressure head while minimizing shock losses. For the present study, volute with trapezoidal cross section has been designed based on the recommendation in following reference [4]. The spiral guide curve is given by the following equation

$$\theta = \frac{360 \cdot C}{Q'} \int_{R_3}^{R} \frac{b \, dr}{r} \quad \text{deg.} \quad (1)$$

Here $C$ is a constant derived from pump head ($H$), rotation speed ($N$) and hydraulic efficiency ($\eta_h$). ‘$\theta$’ (volute angle) and ‘$r$’ which varies from $R_3$ to $R$ in volute design along the increasing angles while moving from throat area towards discharge. $Q'$ (revised volumetric flow) is derived by dividing the actual volumetric flow ($Q$) by volumetric efficiency ($\eta_v$). Based on the analytical calculations above, the CAD model of the volute is developed to match the abstract.

**Table 3: Major geometrical parameters of volute**

| Design parameter                  | Abb. | Unit | Value   |
|-----------------------------------|------|------|---------|
| Trapezoidal cross section angle   | $\alpha_{TS}$ | deg. | 40      |
| Volute starting angle             | $\alpha_v$ | deg. | 14.16   |
| Volute inlet height               | $b_3$ | mm   | 16.80   |
Table 3 above summarizes the major dimensions of the designed volute. Finally the impeller wheel is assembled with the volute while ensuring correct rotational direction.

Figure 2 above depicts the computer aided design (CAD) model of developed impeller wheels and volute flow domain for CFD simulation to assess hydrodynamic performance. The shroud is sectioned at angle to reveal the internal blades of the impeller wheel. These models were developed using CATIA® V5.

4. Numerical Simulation

The flow physics in the fluid domain is described completely by equations given by Navier Stokes, assuming the domain to be a continuous system. However from a technical standpoint considering practical limitations pertaining to computation power, partial differential equations based on RANS (Reynolds Averaged Navier Stokes) formulation are solved using suitable turbulence closure models to predict flow behaviour. To achieve this objective, first from the developed CAD models, the respective fluid flow domain in extracted. Then adequate discretization of the flow domain under investigation is done with appropriate inflation near walls to resolve the viscous sublayer effects. The present CFD simulation has been done using the frozen rotor approach in the commercially available CFD solver ANSYS CFX®, version 19.

Figure 3 below presents the discretization detail of the fluid domains for simulation. Hexahedral elements with high orthogonality have been generated using TurboGrid® for impeller wheels and ICEM CFD® for volute flow passage. Menter’s SST turbulence model (which has been utilized here) is considered to be most suitable for turbomachinery flows [7].

While linear wall treatment been utilized on the hub, shroud and blade walls, logarithmic wall function approach has been utilized on volute walls with appropriate y+ values. After due mesh sensitivity analysis, the number of elements used comes out to be around 900000 for impeller wheel and 1030000 for volute flow domain.

In present analysis, SHe is considered to be incompressible. For geometry simplification, fillets on blade have been ignored and, hub and shroud clearance between impeller wheel and volute has been neglected. Hub, shroud, impeller blade and volute internal walls have been set to “adiabatic” condition. The effect of static heat inleak from ambient (300 K) to operation state (4.2 K) has not been considered. For this comparative study, roughness effects have been for simulation simplicity.
5. Results and Discussion

Extensive CFD analysis of the designed impellers was done to compare the simulation results with predicted analytical performance. This also yields good visualization of internal flow behaviour for further design investigation.

Figure 4 below shows the pressure contour plot for the designed pump assembly. The evolution of the generated pressure profile is uniform. Pressure rise obtained across the pump is 0.21 MPa and it slightly exceeds the required design criterion. In Figure 5 we see the internal flow distribution. Zones of low velocity are observed near blade walls due to viscous sublayer effects. High velocity region is found near the blade leading edge due to its non-optimal alignment with the fluid inlet flow field. Figure 6 reveals the velocity streamline plots for the designed pump. By and large the overall flow distribution is uniform and attached type with no observable localized flow recirculation.
Figure 5: Velocity vector plot at impeller mid span section

Figure 6: Velocity streamline in the impeller for nominal flow condition

Figure 7: Convergence behavior of monitoring parameters
Convergence monitoring is a major criterion to check the smoothness of simulation progress. For steady state simulation, it is primarily influenced by (i) geometry under consideration, (ii) boundary condition applied, (iii) type of mesh with inflation and (iv) turbulence model applied. As shown in figure 7, during CFD simulation the solution was run till a minimum residual convergence value of 1e-4 was attained, at 100 iterations. During this time no major fluctuations or saw teeth instabilities were observed. Further, it was also ensured that the two monitoring parameters set up during simulation, namely pressure difference ($\Delta P$) and hydraulic efficiency ($\eta_h$) attained stable profile well before or near the end of simulation run.

6. Conclusion
The present study outlines the methodology for analytical design derivation of centrifugal pump. Additionally, procedure for CAD modelling, meshing and detailed CFD analysis is also shown. It is seen that the designed pump based on Stepanoff’s approach successfully meets the given design requirements. The achieved hydraulic efficiency comes out to be 88%.

7. References
[1] Turton R K 1994 *Principles of Turbomachinery* (Springer Netherlands)
[2] Balje O E 1981 *Turbomachines: A Guide To Design Selection And Theory* (New York: John Wiley & Sons)
[3] Srinivasan K M 2008 *Rotodynamic Pumps (Centrifugal and Axial) First Edition* (New Age International Private Limited)
[4] Stepanoff A J 1957 *Centrifugal and Axial Flow Pumps: Theory, Design, and Application* (Berlin, Heidelberg: Springer Berlin Heidelberg)
[5] Vasandani D V P 1990 *Theory & Design of Hydraulic Machines* (New Delhi: Khanna Publishers)
[6] Arp V HEPAK®
[7] Mao J Y, Yuan S Q, Pei J, Zhang J F and Wang W J 2014 Applications of different turbulence models in simulations of a large annular volute-type pump with the diffuser *IOP Conf. Ser. Earth Environ. Sci.* 22 022019