

# Design of Centrifugal Pumps for Liquid Propulsion Rocket System

<sup>1</sup>Abdul Rehman Bangash <sup>2</sup>Sibghat Ullah Mahi <sup>3</sup>Haider Ali

Department of Aeronautics & Astronautics

Institute of Space Technology

Islamabad, Pakistan

<sup>1</sup>[aerohaider@hotmail.com](mailto:aerohaider@hotmail.com)

<sup>2</sup>[a.rehman\\_10@hotmail.com](mailto:a.rehman_10@hotmail.com)

<sup>3</sup>[sibghatullahmahi@gmail.com](mailto:sibghatullahmahi@gmail.com)

**Abstract**— High-speed pumps, i.e., pumps with angular speed are widely used in aeronautics, power plants, naval, Space Launch Vehicles, petrochemical industry and missiles. The use of pumps of high rotational speed takes into account the requirements to get high pressures with small dimensions, reduced mass and few stages. This research paper pertains to design and analysis of pumps for Liquid Oxygen (LOX) and Rocket Propellant 1(RP1) based 2.5 Ton propulsion system. Major issues in cryogenic Propulsion systems include weight and size constraints, high RPM's, High Structural Strength requirement, cavitation and ductile to brittle transition. Turbo machinery theory is modeled in MATLAB to attain Geometrical Pump parameters against the required Pressure Head and Volume Flow rate requirements. Pump parameters obtained from the MATLAB code are used for design of pumps in Creo Package. ANSYS Workbench (CFX, Turbogrid) is utilized for Computational Fluid Dynamics (CFD) analysis of the pumps. The calculated Geometrical Parameters and Results of CFD analysis are presented.

**Keywords**— Turbo machinery; Computational Fluid Dynamics (CFD); Centrifugal Pumps; Cryogenic; Rocket Propulsion; ANSYS; CFX; MATLAB

## I. INTRODUCTION

Centrifugal pump is a turbo machine which converts mechanical energy into pressure energy by means of centrifugal force acting on the fluid. It is categorized as rotor dynamic type of pump in which dynamic pressure is developed which empowers the lifting of liquids from lower level to higher level. Since centrifugal action causes lifting of liquid hence it is called as centrifugal pump. Centrifugal pump has high output and high efficiency compared to other types of pumps.

Flow inside the turbo machines, especially centrifugal are the most complex phenomenon in fluid mechanics. Turbo machinery flow's complexity are characterized by their three dimensional aspects, unsteadiness, multiphase, turbulent and interdependent on the geometric dimensions and operating conditions. For such intricate flow analysis, utilization of Computational Fluid Dynamics (CFD) tools is very effective, time and resources saving.

CFD simulation makes it possible to envisage the flow state inside a centrifugal pump, and provides valuable evidence about the centrifugal pumps hydraulic design. Simulation result

is used to compute or foresee the performance of a centrifugal pump to replace or decrease the experiments in the process of pump design. A great deal of effort and facility will be saved, as well as it helps in shortening the design cycle. Therefore, great development on centrifugal pump design must be achieved by CFD analysis of inner flow inside a centrifugal pump and following application of its results in pump design processes.

The development of LPRE pumps is carried out with a common design cycle including the theoretical development of the pump parameters using Euler equations and then using CFD tools to compare the theoretically calculated pump performance with the CFD results. The design cycle include the trade studies using a one dimensional mean line code which yields the initial velocity triangles and initial sub component size. Different custom codes are then used to design the 3D surface definitions for the major flow path in pumps. The 3D surfaces are then used to generate the grid analysis.

Turbomachine, as turbo pumps applied to the LPRE, are designed using empirical relations and turbomachinery equations that are developed based on experimental studies. The design of turbo pumps is a complex and laborious task that takes into consideration some objectives and restrictions. Turbomachinery design is an iterative process [8].

The main requirements of our LPRE engine is provided below in table 1:

Table 1: System Specifications

Vacuum Thrust	25 KN
Propellants	Kerosene / Lox
Pressurization	Pump Fed
Combustion Chamber Pressure	25 bar
Mixing Ratio O/F	2.2

## II. BACKGROUND

Numerical and experimental studies of centrifugal pumps using water and oils of varying viscosities for diverse outlet blade angles established that increasing Outlet blade angles

improves pumps performance. While increase in viscosity results in drastic decrease in performance due to frictional losses [1].

The three-dimensional flow in centrifugal pumps by means of CFD shows zones of low pressure where it would have the possibility of cavitation occurrence at leading edge of the blades of impeller [3] [5].

Oyama and Liou established an optimization method for pumps using genetic algorithm. This is meant to maximize head and minimize the power losses [4].

Rodrigues Cunh and Helcio Francisco Villa Nova have carried out cavitation modeling of a centrifugal pump and predicted cavitation occurrence while observing the pressure field, since it has a direct relation with the vapor pressure at the flow fluid temperature, thus making it possible to add improvements in the project of the equipment in order to prevent or to minimize the phenomenon of cavitation [2].

### III. OBJECTIVE

The main objective of this research paper is to design centrifugal pumps for LPRE, determine their performance parameters using empirical relations and investigate the flow inside the LOX centrifugal pump using the CFX module of CFD. Numerical results are then compared to the theoretical values estimated in the preliminary design process. Using these numerical results the resulting characteristics curves are obtained at design and off design operating conditions. Using these curves it is possible to determine best efficiency point (BEP) that is the point where we have the maximum hydraulic efficiency.

### IV. DESIGN OF LPRE PUMPS

The selected configuration of turbo pump assembly is such that, turbine and both fuel and oxidizer pumps are mounted on the same shaft, supported by two bearings. Turbine is mounted don the center of the shaft while pumps are attached one on each end of the shaft. This configuration simplifies the thermal analysis of the heat flow in the shaft of the turbo pump since the turbine operates at high temperature and LOX pump at low temperature.

LOX pump Positioned at one end is called cantilever pump, decreases inlet diameter and relative flow velocity at blade input [3]. It improves the suction capability of the pump and decreases the cavitation erosion.

Oxidizer and fuel pumps are designed against the requirements in Table2.

The specific work of the pump or Head 'H' can be defined as

$$H = \frac{p_2 - p_1}{\rho} + \frac{c_2^2 - c_1^2}{2}$$

Where,

$$\begin{aligned} p_1 &= \text{inlet static pressure} \\ p_2 &= \text{outlet static pressure} \\ c_1 &= \text{inlet velocity} \end{aligned}$$

$$c_2 = \text{outlet velocity}$$

The hydraulic efficiency of the impeller is defined as

$$\eta = \frac{H}{H_p}$$

Where,

$$\begin{aligned} H &= \text{actual specific work} \\ H_p &= \text{theoretical specific work} \end{aligned}$$

The initial design requirements for the LOX pump are given below in table 2:

Table 2: Pump Specifications

Description	Symbol	Value	Unity
Mass Flow Rate	dm/dt	5.6	Kg/sec
Volumetric Flow Rate	Q	0.008	m3/s
Temperature	T	95	K
Inlet Static Pressure	P <sub>1</sub>	0.4	MPa
Outlet Static Pressure	P <sub>2</sub>	40	MPa

The geometric features of the centrifugal impeller for the LOX pump are summarized in the table 3 below:

Table 3: Impeller Geometrical Features

Description	Symbol	value	Unity
External Diameter	D <sub>2</sub>	10.3	cm
Eye Diameter	D <sub>0</sub>	3.4	cm
Inlet Diameter	D <sub>1</sub>	3.325	cm
Inlet Impeller Width	b <sub>1</sub>	1	cm
Outlet Impeller Width	b <sub>2</sub>	1	cm
B <sub>1</sub>	B <sub>1</sub>	5.2	Degrees
B <sub>2</sub>	B <sub>2</sub>	23.5	Degrees
Blade Number	z	8	-
Rotational Speed	n	15000	Rpm

### V. ASSUMPTIONS

Simulation is based on following assumptions:

- Steady state condition.
- Constant fluid properties.
- Incompressible fluid flow.
- The walls are assumed to be smooth hence any disturbances in flow due to roughness of the surface were neglected.

### VI. ANALYSIS METHODOLOGY

CAD model of the pump is drafted in Creo package. Creo package is also used to define three dimensional fluid domains. CAD model is imported into ANSYS package where the conceptual computational mesh is generated using Turbo grid Module in ANSYS workbench. After generating mesh the

model is exported to the CFX module of ANSYS, where the processing and post processing of the results is performed.

The  $k - \epsilon$  turbulence model is used to cater for the turbulence of the flow due to its simplicity and robustness. The Standard  $k - \epsilon$  model is a semi-empirical model where the equations are based on empirical and phenomenological considerations.

The CFD code CFX treats the problem of flows in moving reference frames e.g. in turbo machinery, with four numerical approaches which are: rotating rotor frame, multi-reference frame, mixing plane and sliding mesh. Multi-reference frame (MRF) is the most advantageous model for turbo machinery simulations. For transient stator-rotor interface, sliding mesh provides better results than others approaches. MRF usually called frozen-rotor assumes that the flow is steady relative to the rotating frame, which simplifies the analysis. This model is appropriate for flows which have low interactions between the stator-rotor parts. The frozen-rotor method provides results global average at each prompt. The calculation results of steady values. The distributions of static, dynamic and total pressure on the surfaces of the impeller are studied. The behavior of the flow field in the interior is envisioned for one better understanding of the phenomena as secondary flow, recirculation, etc.

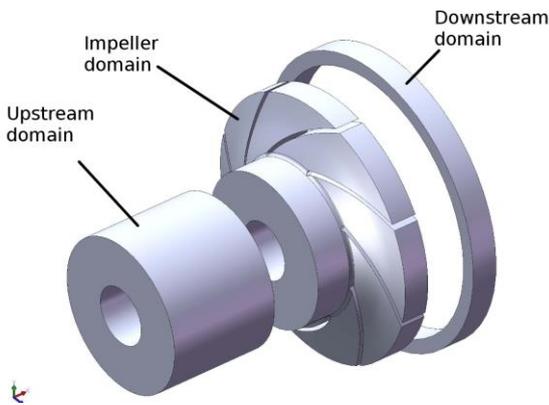


Figure 1: Fluid Flow Regions

### VII. BOUNDARY CONDITIONS

Boundary conditions are a set of conditions specified to determine the behavior of the solution for a set of differential equations at the domain's boundary. Mathematical solutions are determined with the help of boundary conditions of physical problems. These conditions specify the flow and thermal variables on the boundaries of a physical model.

The pump has different geometrical components like inlet, outlet, blades, hub and shroud. The pump inlet is defined as total pressure boundary condition and pump outlet as mass flow rate outlet. Rest of the surfaces were specified as wall boundary conditions. Rotating faces of impeller categorized as wall and zero slip or no slip wall condition is applied. Operating temperature is specified at the inlet.

### VIII. MESHING

An unstructured mesh is created using Turbo grid module of ANSYS workbench in both upstream and downstream domains. The geometry is generated with the aim of division into multiple blocks. Initially, the impeller is generated. After meshing one impeller channel it is then rotationally copied the necessary number of times. The impeller mesh is made with hexahedra and tetrahedral cells. The inlet channel is meshed for both pumps with prisms.

Meshing details are provided in Table 4 below:

Table 4: Mesh Details

<i>Fluid domain</i>	<i>Element</i>	<i>Number of cells</i>	<i>Nodes</i>
Upstream	hexahedral	185000	215543
Impeller	prismatic / tetrahedral	1345228	350672
Downstream	hexahedral	29930	39320

Figures 2 and 3 given below depict computational mesh of the impeller.

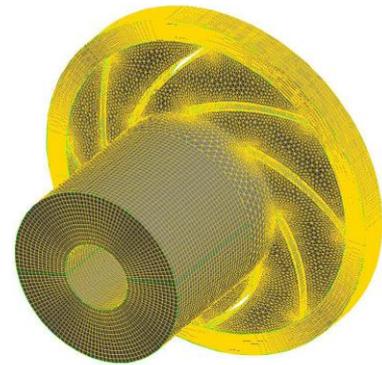


Figure 2: Impeller's Computational Mesh

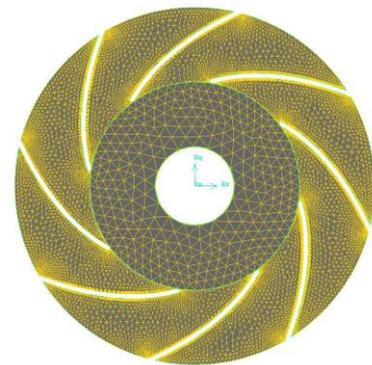


Figure 3: Impeller's Computational Mesh

## IX. RESULTS & DISCUSSION

The numerical results for ten operations points of the LOX centrifugal pump at temperature 90 K are depicted below. The power is in direct relation with the pressure and viscous moments on the surfaces of impeller (blades, hub, and shroud). The sum of these moments in about the axis of rotation is the torque. By multiplying torque with angular velocity the power is determined. The inlet pressure remains constant while the outlet pressure is variable, obtaining the resulting mass flow rate.

Figure 4 shows the characteristic curves of power (N), specific work (H) and hydraulic efficiency ( $\eta$ ) for different flow rates resulting from numerical simulation. These curves are obtained for steady-state, single-phase incompressible flow with constant rotation.

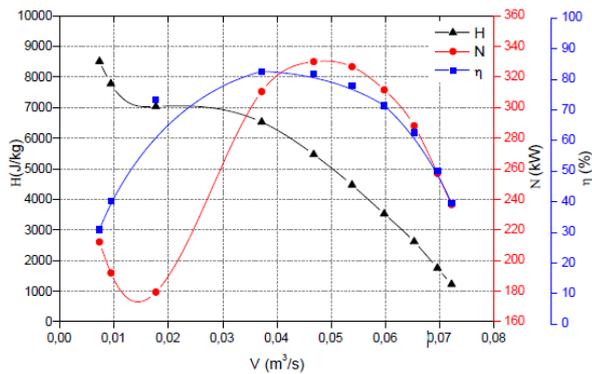


Figure 4: Impeller's Characteristic curves of specific work (H), power (N) and hydraulic efficiency ( $\eta$ ).

From the numerical results, the hydraulic efficiency of the impeller in the preliminary theoretical design is equal to the efficiency obtained by CFD (83%).

The numerical simulations in part-load region (volumetric flow rate below the BEP,  $V < 1$ ), the impeller has shown recirculation in both the suction and the discharge side, as well as the occurrence of secondary flow is detected. These phenomena increase the hydraulic losses, and to a great extent affecting the format of the characteristic curves of specific work, power and efficiency. Figure 5 shows the flow path lines in BEP and Fig. 6 portrays the part load regime. Another phenomenon that affects the format of the characteristic curve was the likely occurrence of rotating stall.

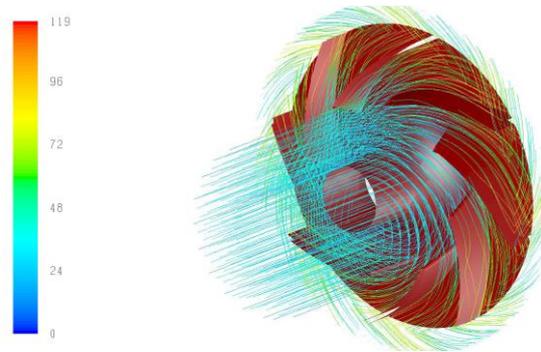


Figure 5: BEP for  $V = 1$ .



Figure 6: Partload regime for  $V < 1$ .

For part-load regime ( $V < 1$ ) the numerical results disclosed a stronger non-uniformity in the pressure and velocity distributions because of strong recirculation at the suction and discharge of the impeller, secondary flow, and flow separation at pressure side of blade, which can depict a possible presence of rotating stall, contributing strongly in the viscous energy losses of the impeller. The generation of a more refined mesh may improve the visualization of secondary flow phenomena, flow separation and rotating stall between impeller channels.

The pressure and velocity distributions for overload regime ( $V > 1$ ) presented smooth and uniform flow field in the channels. The static pressure showed a higher contribution in the total pressure as compared to the dynamic pressure.

For part-load regime ( $V < 1$ ) the numerical results displayed a stronger non-uniformity in the pressure and velocity distributions. This flow pattern was a result of strong recirculation at the suction and discharge of the impeller, secondary flow and flow separation at pressure side of blade, which can be explanation of a possible presence of rotating stall, which is contributing strongly towards the viscous energy loss of the impeller. A more refined mesh could improve the resolution of secondary flow phenomena, flow separation and rotating stall between channels of impeller.

## X. CONCLUSION

Despite its limits, the Standard turbulence model  $k - \epsilon$  was utilized to describe the turbulent flow due to its robustness and low computational cost. The rotor-stator interface Multi-Reference Frame (MRF) was used because it provides

satisfactory results for turbo machinery at steady state. The boundary conditions are properly taken into account at inlet (total pressure), outlet (static pressure) and moving walls of the impeller and stationary walls of the upstream and downstream domains.

The numeric results at different conditions allowed determination of characteristics curves for the centrifugal pump at constant rotational speed of 15000 rpm. About the characteristic curves attained by numerical results, the efficiencies calculated in the hydraulic design of the impeller and are numerically equal (82.4% and 82.6%, respectively).

The pump characteristic curve obtained numerically, presented strong instability for flows below the highest efficiency point obtained numerically, due to the presence of strong recirculation discharge, secondary flow and probable presence of rotating stall.

For the BEP ( $V = 1$ ), impeller has obtainable uniform pressure distribution and uniform flow field in the channel, without the existence of recirculating secondary flow, rotating stall or pre-rotating flow at the impeller inlet. At off-design condition ( $V \neq 1$ ), mainly at part-load regime ( $V < 1$ ), the flow pattern has displayed stronger non uniformity in the pressure distribution and velocity field. The main reason is the strong recirculation at the suction and discharge of the impeller,

secondary flow, and flow separation at pressure side of blade, which strongly contributes to impeller losses.

#### REFERENCES

- [1] M.H. Shojaee and F.A. Boyaghchi, "Studies of the influence of various blade outlet angles in a centrifugal pump when handling viscous fluids," *American Journal of Applied Sciences*, Vol. 9, pp. 718–724, 2007.
- [2] S.Rajendran and K.Purushothaman, "Analysis of a centrifugal pump impeller using ANSYS-CFX," *International Journal of Engineering Research & Technology*, Vol. 1, Issue 3, 2012.
- [3] S.S. Panaiotti, U.S Rohatgi, S.F. Timushev, V.A. Soldatov and B.N. Chumachenko, "CFD study of prospective 1st stage centrifugal impeller design," 5th Joint ASME/JSME Fluids Engineering Conference, San Diego, California USA 2007.
- [4] A. Oyama and M. Liou, "Multiobjective optimization of rocket engine pumps using evolutionary algorithm," *Journal of Propulsion and Power*, Vol. 18, No. 3, pp. 528–535, 2002.
- [5] M. J. Kim, H. B. Jin, and W. J. Chung, "A Study on Prediction of Cavitation for Centrifugal Pump," *World Academy of Science, Engineering and Technology*, Vol. 6, 2012.
- [6] M. Younsi, "A'eroacoustique et a'erodynamique instationnaire, num'erique et experimentale des ventilateurs centrifuges action," Ph.D. thesis, L' 'Ecole Nationale Sup'erieure d'Arts et M'etiers, 2007.
- [7] R. S. Muttalli , S. Agrawal , H. Warudkar, "CFD Simulation of Centrifugal Pump Impeller Using ANSYS-CFX," *International Journal of Innovative Research in Science, Engineering and Technology*, Vol. 3, Issue 8, 2014.
- [8] B.V. Ovsyannikov and B.I. Borovskiy, "Theory and calculation of feed units of liquid propellant rocket engines", Ohio:Foreign Technology Division,1973.