IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X
PP 33-41
www.iosrjournals.org

CFD Analysis of centrifugal pump impeller for performance enhancement

P.Gurupranesh [1], R.C.Radha [2], N.Karthikeyan [3],
[1] *Assistant Professor, Indra Ganesan College of Engineering, Trichy, India*
[2] *Assistant Professor, St Michael College of Engineering and Technology, Karaikudi, India*
[3] *Assistant Professor, Indra Ganesan College of Engineering, Trichy, India*

ABSTRACT: *Centrifugal pumps are used extensively for pumping water over short to medium distance through pipeline where the requirements of head and discharge are moderate. This project is devoted to enhance the performance of the centrifugal pump through design modification of impeller. Theories on pump characteristics are studied in detail. Vane profile of the impeller is generated using point by point method. The impeller is modelled in Solidworks 2012 software and CFD analysis is done using fluid flow simulation package. CFD analysis enables to predict the performance of the pump and a comparative analysis is made for the entire control volume by varying meshing*

Pump impeller models have been developed for critical design parameters of the pump. CFD analysis is done in the models to predict the pump performance virtually. Experimental analysis is to be carried out during the second phase of the project work.

Keywords: *centrifugal pump, impeller analysis, pump performance, pump characteristics, CFD analysis*

I. INTRODUCTION

A pump is a mechanical device for moving a fluid from a lower to a higher location, or from a lower to a higher pressure area. Mechanical energy is given to the pump and it is then converted into hydraulic energy of fluid. Pumps produce negative pressure at the pressure at the inlet so that the atmospheric pressure pushes the fluid towards the pump. The fluid coming into the pump is pushed the towards the outlet mechanically where positive pressure is generated. Pumps are classified in number of the ways according to their purpose, specifications, design, environment etc.

1.1 Selection of pump for performance enhancement

The design and performance analysis of centrifugal pump are chosen for the project work, because it is the most useful mechanical rotodynamic machine in fluid works which widely used in domestic, irrigation, industry, large plants and river water pumping system. These pumps are used at the place where the requirements of head and discharge are moderate. Extensive use of centrifugal pumps is because of the following reasons.

1. Initial cost of pump is low
2. Efficiency is high
3. Discharge is uniform and continuous flow
4. Installation and maintenance is easy
5. It can run at high speeds without the risk of separation of flow

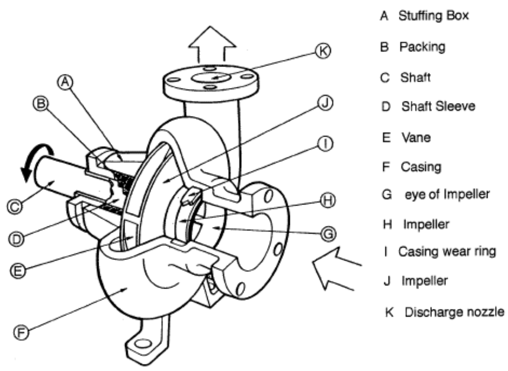


Fig1 Components of Centrifugal Pump

Many researches are going on in the field of centrifugal pump to improve the performance and to reduce the losses, such as turbulence loss, Shock losses, impeller friction losses, Volute friction losses, disk friction losses and recirculation losses and also power consumption

. Experimental investigations are generally carried out on pumps which are expensive, time consuming and limited to some extent. To reduce the number of experimental works, virtual analysis can be carried out on different pump models with the use of CFD packages and pump performance can be predicted.

1.2 Types of centrifugal pump

Centrifugal pumps are most often associated with the radial flow types. However, the terms "Centrifugal Pumps" can be used to describe all impeller type rotodynamic pumps including the radial, axial and mixed flow variations.

1.2.1 Radial Flow Pumps

These pumps are often simply referred to as centrifugal pumps. The fluid enters along the axial plane, is accelerated by the impeller and exists at right angles to the shaft (radially). Radial flow pumps operate at higher pressures and lower flow rates than axial and mixed flow pumps. The radial flow pumps, by its principle, are converse of the Francis turbine. The flow is radially outward, and the hence the fluid gains in centrifugal head while flowing through it. Because of certain inherent advantages, such as compactness, smooth and uniform flow, low initial cost and high efficiency even at low heads, centrifugal pumps are pumps are used in almost all pumping systems.

1.2.2. Axial flow pumps

Axial flow pumps differ from radial flow in that the fluid enters and exits along the same direction parallel to the rotational shaft. The fluid is not accelerated but instead "lifted" by the action of the impeller. They may be linked to a propeller spinning in the length of the tube. Axial flow pumps operate at much low pressures and higher flow rates that radial flow pumps.

1.2.3. Mixed flow pumps

Mixed flow pumps, as the name suggests, function as a compromise between radial and axial flow pumps , the fluid experiences both radial acceleration and lift and exists the impeller somewhere between 0-90 degrees from the axial direction. As a consequence mixed flow pumps operate at high pressure than axial flow pumps while delivering higher discharges than radial flow pumps. The exit angle of the flow dictates the pressure head-discharge characteristic in relation to radial and mixed flow.

1.3. General characteristics of centrifugal pump

Every centrifugal pump has two characteristics that are same; each has an impeller that forces the liquid being pumped into a rotary motion, each has a casing, which directs the liquid to the impeller. The liquid leaves the impeller as the impeller rotates.

The liquid leaves with high velocity and pressure than it had when it entered. There is a conversion of some of the velocity to pressure than takes place before the liquid leaves the pump; this partial conversion takes place in the pump casing. Head loss is associated with conversion and must be taken into account.

The size of the impeller and the pump casing vary greatly with the type of centrifugal pump. Centrifugal pumps are often classified by a type number known as the specific speed that varies with the shape of the impeller. Two main components of a centrifugal pump are the impeller and the casing. The impeller is a rotating component and the casing is a stationary component .The impeller is a rotating component. In centrifugal pump, water enters axially through the impeller eyes and water exits radially. The pump .Casing is to guide the liquid to the impeller, converts into pressure the high velocity kinetic energy of the flow from the impeller discharge and leads liquid away of the energy having imparted to the liquid comes from the volute casing.

In a centrifugal pump, the liquid is forced by atmospheric or other pressure into a set of rotating vanes enclosed within a housing or casing that is used to impart energy to a fluid through centrifugal force.

The design and performance analysis of radial flow centrifugal pump are chosen because it is the most useful mechanical rotodynamic machine in fluid works which is widely used in domestic, irrigation, industry, large plants and river water pumping system.

1.4. Objective

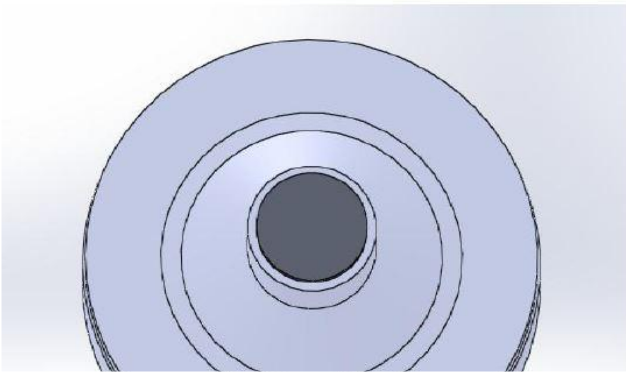
- ✓ To design a centrifugal pump impeller for the given specification.
- ✓ To provide design methodology for centrifugal pump impellers.
- ✓ Proposed for the Partial modification of the blade flow passage which affects the entire flow field.
- ✓ To provide centrifugal impellers that can operate more efficiently and quietly.
- ✓ To use a commercial CFD software to find the change in performance from initial design.

Table 1. Design Specification

S.No	**PARAMETERS**	**MODEL**
1.	Specific speed	973 rpm
2.	Power input to the pump	15.12hp
3.	Shaft diameter	17.69mm
4.	Hub diameter	25mm
5.	Outer diameter of impeller	0.2m
6.	Velocity of fluid at the impeller inlet	4.103m/s
7.	Eye diameter of impeller	76mm
8.	Inner diameter of impeller	74mm
9.	Inlet blade angle	20.3^o
10.	Impeller outlet area	5943.5mm2
11.	Impeller width at inlet	9.905mm
12.	Blade angle at outlet	24^o
13.	Number of blades	6

II. MODEL CONSTRUCTION

Three-dimensional model of an impeller was first created in Solidworks 2012 software and exported into STEP files. The STEP files were then imported into fluid flow simulation, the mesh generator. The fluid volume was split into a rotating fluid volume, a scroll volume , an inlet cone volume, and an inlet/outlet duct volume. The inlet and outlet ducts were intentionally set to simulate the actual measuring situation and to provide better boundary conditions for simulations. The flow was assumed fully developed when leaving the inlet and outlet ducts. The impeller wheel volume was defined as a rotating reference frame with constant speed, and other blocks were defined in a stationary frame. This setup is referred to as a "frozen rotor" model.



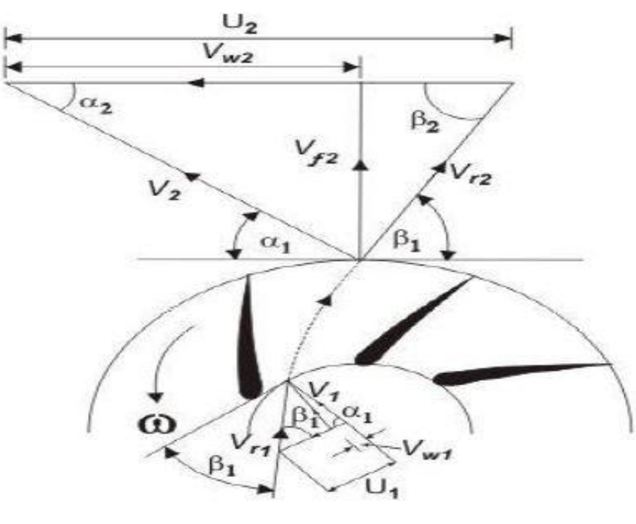


fig 2 Model of the impeller

fig 3 primary parameters of impeller

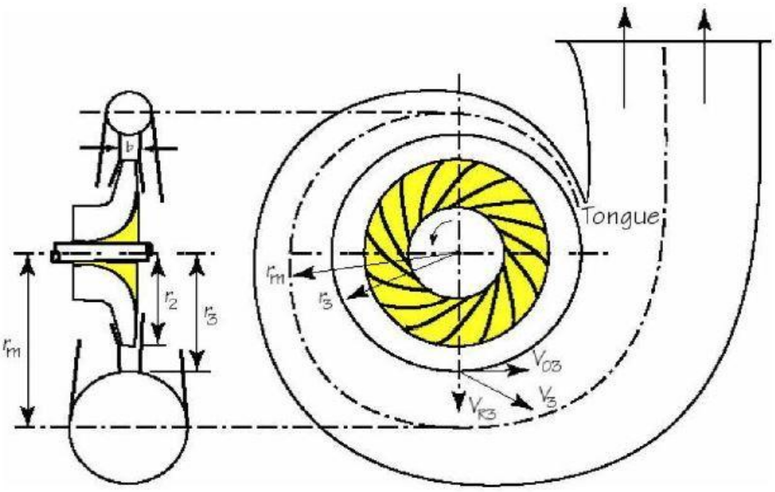


fig 4 parameters of pump

III. INTRODUCTION TO CFD

Computational fluid dynamics (CFD) uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow (the Navier- strokes and allied equations), for predefined geometries and boundary conditions. The result is a wealth of the predictions for flow velocity, temperature, density, and chemical concentrations for any region where flow occurs.

CFD analysis begins with a mathematical model of a physical problem, conservation of matter, momentum, and energy must be satisfied throughout the region of interest. Fluid properties and modeled empirically. Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, in viscid, two dimensional). Provide appropriate initial and boundary conditions for the problem. CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanical in the fluid region of interest. The solution is post processed to extract quantities of interest (e.g. lift, drag, torque, heat transfer, separation, pressure loss, etc.).

3.1 Practical advantages of employing cfd

The followings are among the many reasons why CFD is being widely used today:

3.1.1 CFD predicts performance before modifying or installing the systems:

Without modifying and/or installing actual systems or prototype, CFD can predict what design change is most crucial to enhance performance.

3.1.2 CFD provides exact and detailed information about HVAC design parameters:

The advances in HVAC/IAQ technology require broader and more detailed information about the flow within an occupied zone, and the CFD technique meets this goal better than any other methodize. Theoretical or experimental methods.

3.1.3. CFD saves cost and time:

CFD costs much less than experiments because physical modifications are not necessary. (Note that the cost and time for physical changes / modifications increase almost exponentially as the size of the system increases).

3.1.4 CFD is reliable:

Most importantly, numerical schemes and methods that CFD is based on are improving rapidly so that reliability on the results produced by CFD is getting very high. Increased reliability makes CFD a dependable tool in any design and analysis purpose. CFD as an engineer's tool the concept of virtual prototyping conceptual design engineering design CFD simulation data analysis final design (prototype).

3.2 LIMITATIONS OF CFD

3.2.1 Physical models

- ✓ CFD solutions rely upon physical models of real world processes (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.).
- ✓ The CFD solutions can only be as accurate as the physical models on which they are based.

3.2.2. Numerical errors

- ✓ Solving equations on a computer invariably introduces numerical errors.
- ✓ Round-off error: due to finite word size available on the computer. Round-off errors will always exist (through they can be small in most cases).
- ✓ Truncation errors: due to approximations in the numerical models. Truncation errors will go to zero as the grid is redefined. Mesh refinement is one way to deal with truncation error.

3.3 Boundary conditions

- ✓ As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model.
- ✓ Example: flow in a duct with sudden expansion. If flow is supplied to domain by a pipe, you should use a fully-developed profile for velocity rather than assume uniform conditions.

3.4 Discretization

Domain is discretized into a finite set of control volumes or cells. The discretized domain is called the "grid" or the "mesh". General conservation (transport) equations for mass, momentum, energy, etc., are discretized into algebraic equations. All equations are solved to render flow field.

IV. RESULTS AND DISCUSSION

Numerical simulations are carried out on the impeller model to predict its performance by giving its working conditions as input. Successive iterations are done by the software to obtain the characteristics such as efficiency, static pressure generated, pressure distribution, direction of flow, turbulence, fluid velocity. The fluid taken for the investigation is water.

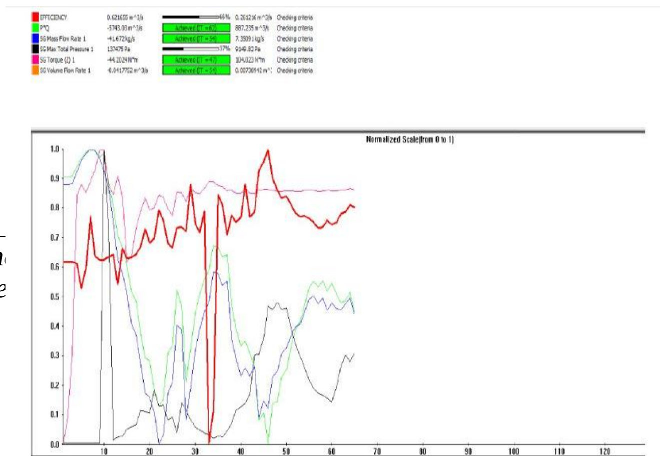


Fig 5 Efficiency Graph

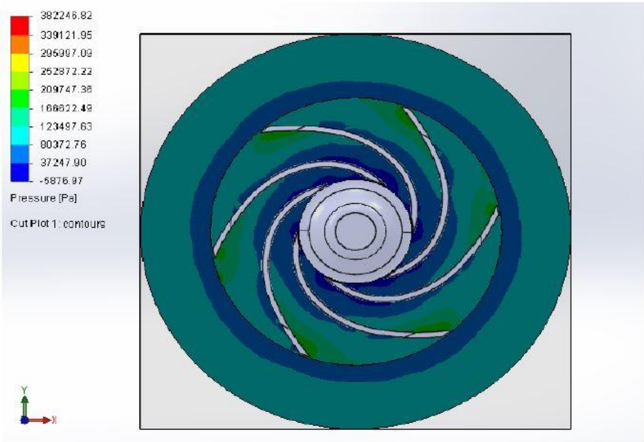


Fig 6 pressure diagram

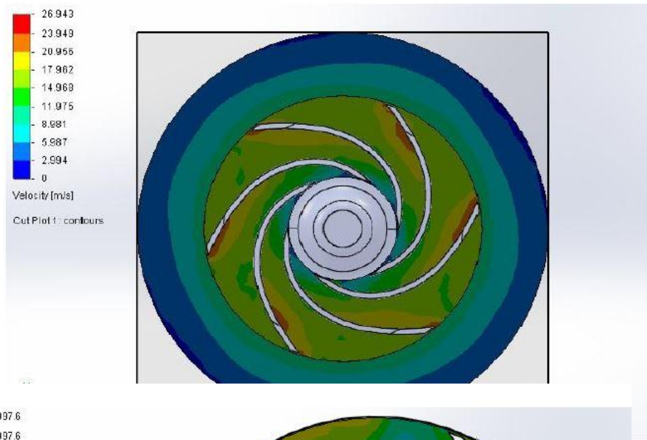


Fig 7 velocity diagram

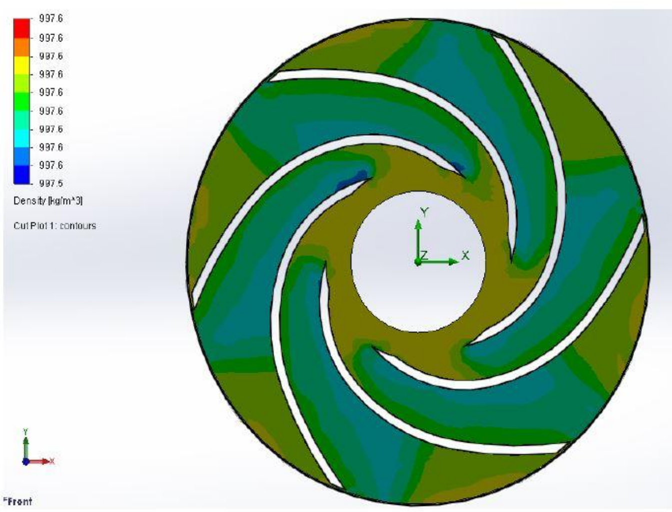


Fig 8 density diagram

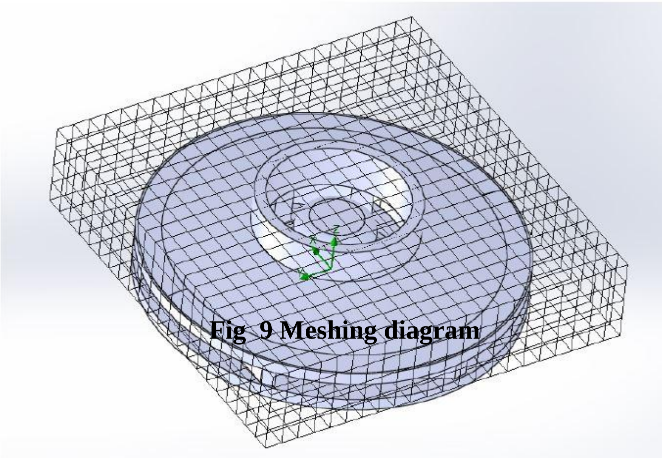
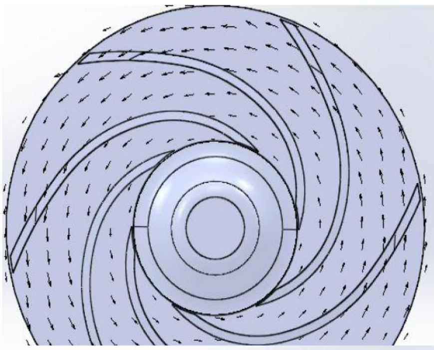


Fig 9 Meshing diagram



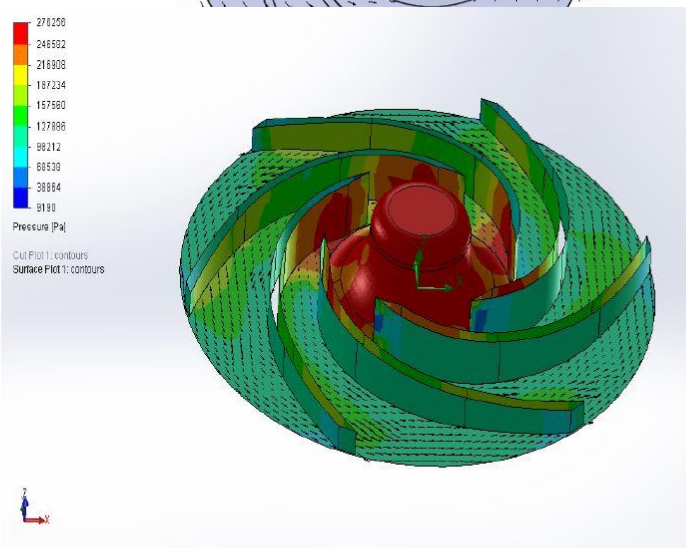


Fig 10 flow diagram

Fig 11 pressure

The above diagram enables to understand the distribution of efficiency, pressure, velocity, fluid flow diagram. The results obtained using fluid flow simulation is given below:

- ✓ Pressure developed in the fluid P= 38446.82 Pa and it is shown in the figure
- ✓ Fluid flow rate Q=0.0417752 m3/s and it is shown in the figure
- ✓ Efficiency ŋ= 62.16% and it is shown in the figure.

V. CONCLUSION

Pump impeller is designed for the given specification and numerical analysis is carried out in fluid flow simulation package. Contour plots are also obtained for the distribution of static pressure, velocity and wall shear stress. The following are the performance evaluation of the pump, arrived from CFD study. Overall efficiency of the pump is 61%, CFD results predict total head of 50 m.

REFERENCE

[1]. AEA Technology Engineering Software Limited (2000). CFX-TASCflow Computational Fluid Dynamics Software: Primer Manual, Version 2.10.
[2]. Ahn CS, Kim KY (2003). Aerodynamic design optimization of a compressor Rotor with Navier- Stokes analysis. Proceedings of the Institution of Mechanical Engineers. Part A, Journal of Power and Energy.
[3]. Asuaje M, Bakir F, Kouidri S, Rey R (2004). Inverse design method for centrifugal impellers and comparison with numerical simulation tools. International Journal of Computational Fluid Dynamics.
[4]. Byskov RK, Jacobsen CB (2003). Flow in a centrifugal pump impeller at design and off- design conditions—part II: large eddy simulations. Journal of Fluids Engineering.
[5]. Chen SL, Wang WT (2001). Computer aided manufacturing technologies for centrifugal compressor impellers. Journal of Materials Processing Technology.
[6]. Dawes WN, Dhanasekaran PC, Kellar WP, Savill AM (2001). Reducing bottlenecks in the CAD-to-Mesh-to-Solution cycle time to allow CFD to participate in design. Journal of Turbo machinery.
[7]. Fan HY (1998). An inverse design method of diffuser blades by genetic algorithms. Proceedings of the Institution of Mechanical Engineers. Part A, Journal of Power and Energy 212:261–268.
[8]. Flathers MB, Bache GE (1999). Aerodynamically induced radial forces in a centrifugal gas compressor. Journal Engineering for Gas Turbines & Power
[9]. Greenman RM, Roth KR (1999). High-lift optimization design using neural networks on a multi-element airfoil. Journal of Fluids Engineering
[10]. Langthjem MA, Olhoff N (2004). A numerical study of flow-induced noise in a two- dimensional centrifugal pump. Journal of Fluids and Structures.
[11]. Lewis RI (1996). Turbomachinery Performance Analysis. John Wiley, New York.
[12]. Lighthill JM (1945). A new method of two- dimensional aerodynamic design.

[13]. Lin BJ, Hung CI, Tang EJ (2002). An optimal design of axial-flow fan blades by the machining method and an artificial neural network. Proceedings of the Institution of Mechanical Engineers. Part C, Journal of Mechanical Engineering Science.
[14]. Majidi K (2005). Numerical study of unsteady flow in a centrifugal pump. Journal of Turbo machinery.
[15]. Medvitz RB, Kunz RF, Boger DA, Lindau JW, Yocum AM (2002). Performance analysis of cavitating flow in centrifugal pumps using multiphase CFD. Journal of Fluids Engineering.
[16]. Morishige K, Takeuchi Y (1997). 5-axis control rough cutting of an impeller with efficiency and accuracy. Proceedings of the 1997 IEEE International Conference on Robotics and Automation.
[17]. Nerurkar AC, Dang TQ, Reddy ES, Reddy DR (1996). Design study of turbo-machinery blades by optimization and inverse techniques.
[18]. Oh HW, Chung MK (1999). Optimum values of design variables versus specific speed for Centrifugal pumps. Proceedings of the Institution of Mechanical Engineers. Part A, Journal of Power and Energy.
[19]. Oyama A, Liou MS (2002). Multiobjective optimization of rocket engine pumps using evolutionary algorithm. Journal of Propulsion and Power.
[20]. Pak ET, Lee JC (1998). Performance and pressure distribution changes in a centrifugal pump under two-phase flow. Proceedings of the Institution of Mechanical Engineers. Part A, Journal of Power and Energy.
[21]. Pierret S, Van den Braembussche RA (1999) Turbo machinery blade design using a Navier-Stokes solver and artificial neural network. Journal of Turbomachinery.
[22]. Tanaka S, Isomura K, Togo SI, Esashi M (2004). Turbo test rig with hydroinertia air bearings for a palmtop gas turbine. Journal Micromechanics and Microengineering.
[23]. Tsay DM,Lin BJ (1997). Design and machining of globoidal index cams. Journal of Manufacturing Science & Engineering.
[24]. Wang WP,Wang KK (1986). Geometry modelling of swept volume of moving solids. IEEE Computer Graphics and Applications.
[25]. Yedidiah S (1991). An alternate method for calculating the head developed by a centrifugal impeller. ASME/JSME Fluids Engineering Conference.