



A PROCEDURE FOR OBTAINING COMPLEX GEOMETRY AND FLUID FLOW SIMULATION OF CENTRIFUGAL TURBOPUMP IMPELLER

Mehdi Bakhshesh¹, Seyed Saied Bahrainian²

¹Shohadaie Hovaizeh University of Technology, Iran

²Assistant Professor of Mechanical Engineering, Shahid Chamran University

*Correspondence author: Fax: +98 612 4231020 Email: mehdi.bakhshesh@shut.ac.ir

ABSTRACT

Geometrical solid modeling is the first step in flow simulation within solid boundaries with complex or unknown geometry. For complex rotating geometry such as turbo pump impellers, ordinary measuring tools cannot be used directly due to the existence of complex curvature and acute angles. Therefore, a combination of direct measuring tools such as vernier, French curves and indirect measuring methods such as casting and solid modeling software must be used to obtain the dimensions of the unknown body. In this paper, a new procedure for obtaining complex geometry of solid bodies is presented. An industrial turbo-pump impeller that employed in Bidboland gas refinery has been chosen to represent the complex geometry. The proposed procedure is based on making small chuck casting from the complex passages and simultaneously making use of available computer aided design (CAD) software. Dimensions obtained from the small casts and curve fitting methods and simultaneously using solid modeling software gave the complete unknown geometry.

NOMENCLATURE

ρ	density
V	velocity
P	pressure
μ	viscosity
ω	vorticity Vector
k	turbulent kinetic energy
ε	dissipation rate of turbulent kinetic energy

INTRODUCTION

Employing computational fluid dynamics (CFD) for simulation of fluid flow require accurate geometrical dimensions of the solid body. One such application is

the simulation of flow through cascades of impeller blades. However, prior to flow simulation the (CAD) geometry must be prepared. This process can take anywhere from hours to weeks, especially if the solid passage has an unknown or complex geometry. The physical flow domain would then have to be extracted from the solid geometry. The next step is to generate a computational grid in the extracted geometry. The discretized extracted geometry now serves as the computational flow domain. The last step is to do the flow simulation over the computational grid.

The above CAD and CFD procedure given for a simple solid geometry may be a trivial task. However, for a complex geometry and realistic flow conditions, the above procedure can in fact be a very difficult and time consuming task.

In many instances, the dimensions of the complex geometry are unknown, which further complicates the modeling and simulation process. This paper describes a new procedure for geometrical modeling of complex geometry. To demonstrate the above CAD-CFD process for a realistic complex body as applied to rotating machinery, a centrifugal impeller of an industrial turbo pump was selected. The unknown geometrical dimensions of this impeller further exaggerate the complexity of the modeling process. Figure 1 illustrates the centrifugal impeller showing the blades at inlet and exit. The passages formed by the axial inlet and the radial outlet constitute volumes with complex geometry. The primary objective is to obtain geometrical model of the passages. The unknown dimensions and curvatures of the passage geometry make this objective particularly challenging.



a: blades at inlet b: flow passage exit

Figure 1

Centrifugal impeller of an industrial turbo pump

GEOMETRICAL MODELING OF FLOW PASSAGE REGIONS

To obtain a geometrical model, dimensions of the solid body must be measured. Different methods can be used to measure geometrical dimensions. These methods include direct measurement using different length measuring devices such as ruler, micrometer, and other sophisticated measuring tools such as laser scanner. But, each method or measuring tool depending on the complexity of the body's geometry has limitations. The existence of curvatures and acute angles within impeller passages has limited the direct measuring devices. Therefore, other approaches must be taken to obtain the overall geometry of the impeller. The approach taken here includes the simultaneous use of, direct measuring tools, and indirect measurement such as casting.

In fact, the procedure referred to here relies on small casts from complex passages and their simultaneous modeling with CAD software. Figure 2 shows a sample chuck cast from one passage of the impeller. Measurement of the chuck cast is much easier and straightforward than the original passage. Transfer of data from the cast to CAD software is the next step in this process.



Figure 2

A sample chuck cast from one passage of the centrifugal impeller

It should be noted that accuracy or smoothness of the chuck cast is not important. Furthermore, the cast does not require representing a complete shape of the entire passage. This is because of the fact that only data from selected slices through the cast is sufficient enough to construct the geometry using any CAD software. By transferring the measured lengths from

slices of the chuck cast to CAD software, passage cross sections along the blade curvature are obtained. The arrangement of a few passage cross sections along blade curvature can be seen in Figure 3. Most CAD software can use curve fitting options to simply connect the coordinates of the provided cross sections. There is no limitation for the quantity and locations of the selected cross sections. It is, however, important to supply the inlet and outlet cross sectional data. Also, care should be taken that selected locations of cross sections conveys the required and sufficient data to keep the integrity of the original passage curvature.



Figure 3

Arrangement of passage cross sections along blade curvature.

If the final blade or fluid passage curvature is not formed by the provided data, the number of cross sectional data can be increased to reach the correct final shape. Figure 4 shows the final passage geometry along with the measured cross sections. It can be seen that by connecting the coordinates of the cross sections the final blade curvature is obtained.

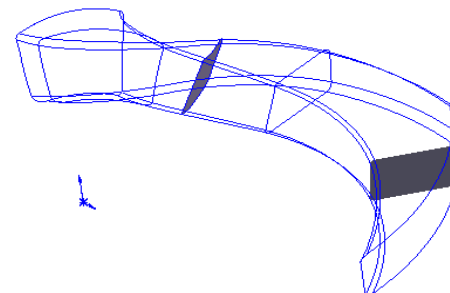


Figure 4

Final passage geometry along with measured cross sections

Mathematically speaking, the final passage geometry is obtained at this point. However, the wire frame geometry has no mathematical significant as far as fluid simulation is concerned. The next step is to construct solid volume geometries that represent the fluid flow passage. For this reason, the wire frame passage is turned into a solid volume. This process is performed by connecting two-dimensional

curvature. Figure 6 illustrates this procedure where cross-sections are connected along the blade curvature to form a partial solid volume. The solid volume is advanced by filling the empty volume up to the next passage cross sectional area along the blade. Upon completion, the solid volume that represents the fluid flow passage is shown in Figure 5.

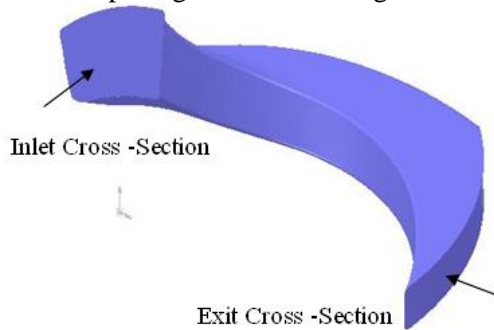


Figure 5
Solid volume representing one fluid flow passage of a centrifugal impeller

The pump impeller selected for this study contains seven symmetrical fluid passages. This can know be modeled by simply arranging seven passage volume geometries around axis of rotation. By doing so, the solid volume that represents fluid flow passages of a centrifugal impeller is obtained. These passages that are shown in Figure 6 are sufficient enough to be used for computational fluid dynamic simulations. Nevertheless, for consistency, next section details the required steps needed to reach a complete impeller model.

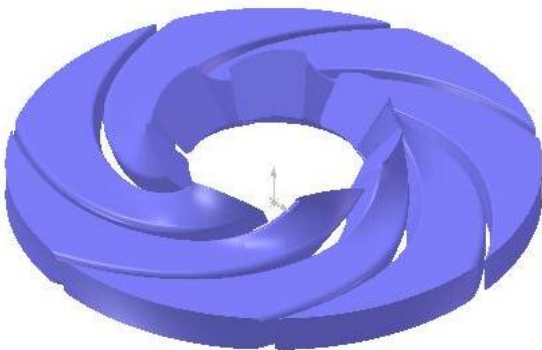


Figure 6
Solid volumes that represent fluid flow passages of a centrifugal impeller

GEOMETRICAL MODELING OF TURBO PUMP IMPELLER

To model a complete impeller, it is divided into different simpler geometries. These geometries include back and front disks, blades, and passages described in the previous section. Back and front impeller disks have accessible geometries and their

dimensions can easily be measured using a long range micrometer. Blade and passage curvatures have essentially the same geometry. That is the blade geometries are actually sidewalls to the passages. By adding the seven passage walls (impeller blades) to that of the back impeller disk a combination is formed. If the front impeller disk is added to the above geometry, a complete three dimensional impeller solid model is obtained. The complete impeller solid model is shown in Figure 7a. To show geometrical details a cross sectional view is shown in Figure 7b. By comparing this solid model to the illustrations given by Figure 1, one can see the geometrical similarity.

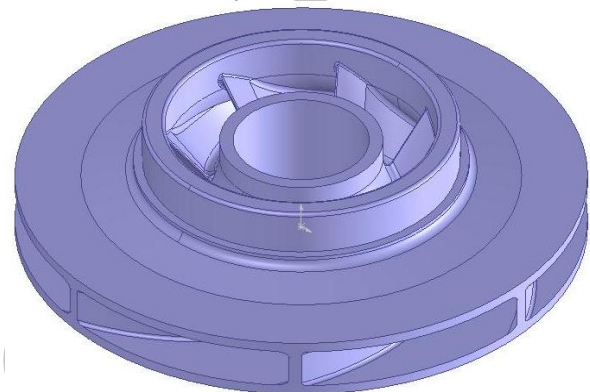


Figure7a
Impeller solid model

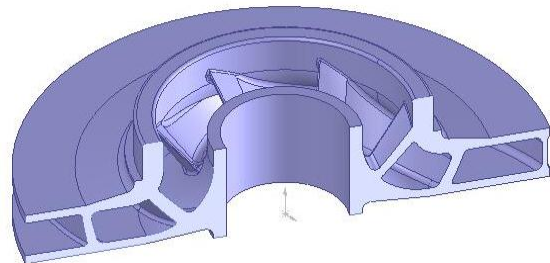


Figure 7b
Cross sectional view of impeller solid model

COMPUTATIONAL GRID GENERATION

To discretize the computational domain of the example presented in this paper, because of the complexity of impeller geometry, 3-D tetrahedral unstructured grids is used. Again, making use of symmetry, computational grid output of seven passages was obtained to give a computational grid output for the impeller. This final grid is shown in Figure 8. Figure 9 shows close views of final surface faces on passage walls.

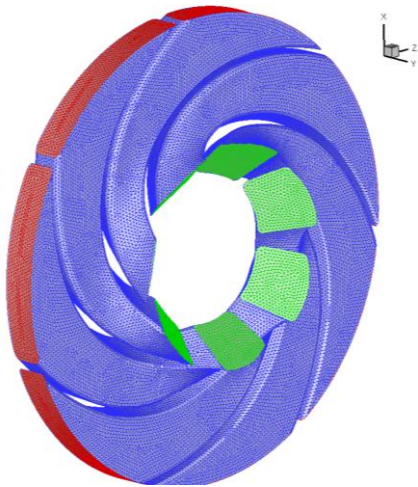


Figure 8
computational grid inside seven passages of the centrifugal impeller

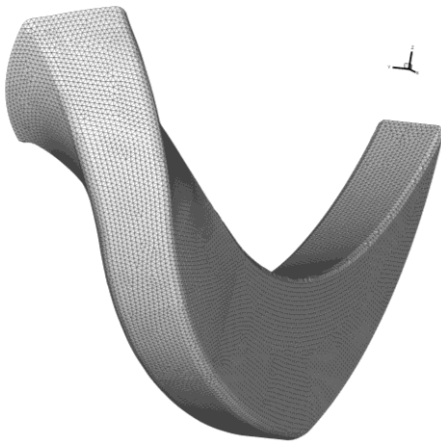


Figure 9
close views of final surface faces on passage walls.

GOVERNING EQUATIONS

Governing equation for flow fluid contain continuity and momentum equations. These equations can be written in the rotating coordinate system as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{\bar{V}}) = 0 \quad (1)$$

$$\frac{\partial}{\partial t} (\rho \bar{\bar{V}}) + \nabla \cdot (\rho \bar{\bar{V}} \otimes \bar{\bar{V}}) = -\nabla P + \nabla \cdot \left(\mu \left(\nabla \bar{\bar{V}} + (\nabla \bar{\bar{V}})^T \right) \right) + \frac{\partial}{\partial x_j} (-\rho \bar{V}_i \bar{V}_j) + S_M \quad (2)$$

Where $\bar{\bar{V}}$ is the average velocity, P is the pressure, ρ is the density, μ is the viscosity, and S_M is the source term. For flows in a rotating frame of reference that are rotating at the constant rotation speed, the effects of Coriolis are modeled.

$$S_M = -\rho [2\Omega \times U + \Omega \times (\Omega \times r)] \quad (3)$$

Where, r is the location vector. The vorticity vector, ω , is defined as

$$\omega = \text{curl } V \quad (4)$$

Reynolds stress term $\overline{\rho V_i' V_j'}$ is the unknown and this new turbulence term must be modeling and attribute suitable quantity. Conservative form of the $k-\varepsilon$ model that are used here are given by the following equations:

$$\frac{\partial(\rho k)}{\partial t} + \text{div}(\rho k U) = \text{div} \left[\frac{\mu_t}{\sigma_k} \text{grad } k \right] + 2\mu_t E_{ij} \cdot E_{ij} - \rho \varepsilon \quad (5)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \text{div}(\rho \varepsilon U) = \text{div} \left[\frac{\mu_t}{\sigma_\varepsilon} \text{grad } \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon}{k} 2\mu_t E_{ij} \cdot E_{ij} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (6)$$

BOUNDARY CONDITION AND INITIAL CONDITIONS:

The following boundary condition can be set:

- Inlet boundary: A constant mass-flow rate was specified at the inlet of the calculation domain for each computation. Various mass-flow rates were specified so as to study design and off-design pump conditions.
- Outlet boundary: In the outlet of the calculation domain, the gradients of the velocity components were assumed to be zero.

Flow is assumed to be at a uniform state for initial condition. The flow pressure and velocity components are set to initial value of inlet boundary condition. A solution of certain chemical fluid, (DEA) has employed as working fluid. Molecular weight, Density, Viscosity, and weight percentage of DEA fluid are 105.14, 1.095 (gm/cc), 350 (Cp), and 25.8% respectively. The operation conditions of pump are consisted of flow rate 205 (m³/h), inlet pressure 0.78 (MPa) outlet pressure 6.27 (MPa), and rotational speed 3300 (rpm).

RESULTS AND DISCUSSION

Figure 10 shows static pressure contours. The blades are rotating in the counter-clockwise direction. It is seen that by forward movement of fluid in passages and rotation of impeller, pressure increases from inlet to outlet. In figure 11, variation of relative



velocity between two adjacent vanes for two different radiuses has been shown. It is seen that relative velocity in suction side more than relative velocity in pressure side.

It is because of flow fluid in impeller tends to rotate in contrary direction of impeller rotation.

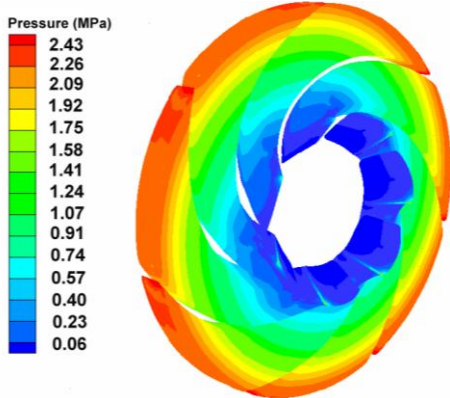


Figure 10
static pressure in impeller

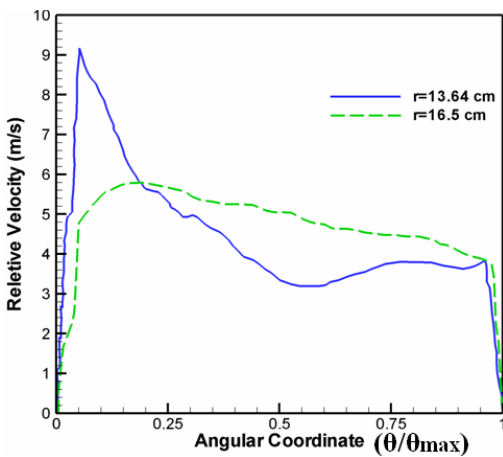


Figure 11
Variation of relative velocity in passage between two adjacent vanes

Figure 12a shows relative velocity vector in passage between two adjacent vanes for 205 m³/h flow rate and 3000 rpm. It is seen that fluid flows from inlet to outlet uniformly. In Figure 12b relative velocity vector for same flow rate and 3300 rpm has been shown. It is observed vortices are made near the impeller outlet and pressure side. Also by fixing rotational speed and reduction of flow rate to 159 m³/h vortices are made in passage. Figure 13a shows distribution of relative velocity vector in 159 m³/h flow rate and 3600 rpm rotational speed in vicinity inlet of passage. In this state, rotational speed is increased and flow rate is decreased. From this Figure it is specified that a pair of strong re-circulating

vortices are made in inlet at two region vicinity both pressure and suction side. Fluid is allowed to be flow only at very small region between two vortices regions. This phenomenon cans very undesirable influence on the turbopump performance. In addition, distribution of relative velocity vector at 159 m³/h flow rate and 3600 rpm rotational speed is shown in vicinity outlet of passage in Figure 13b. It has been seen that vortex flow is generated at pressure side of passage contrary to rotational direction of blades. The simulation clearly shows that the formation of vortex. Vortex founding in impeller cause to generation unsteady forces. Since turbo pump impeller is rotating, so started to vibrate because of these forces. It is clear that impeller motion affects on the flow fluid. In fact this vortices cause to pump vibration with high frequency. This phenomenon call vortex induced vibration. By increase of flow rate or reduction of rotational speed, vortex velocity is decreased in compare of velocity flow in streamline direction and hence reverse flow occurred lesser. For this reason, distribution of velocity vector becomes uniformed in the passages.

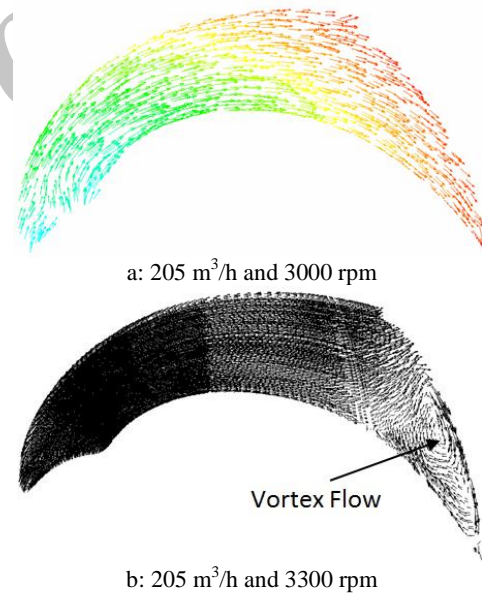


Figure 12
relative velocity vectors in middle plane

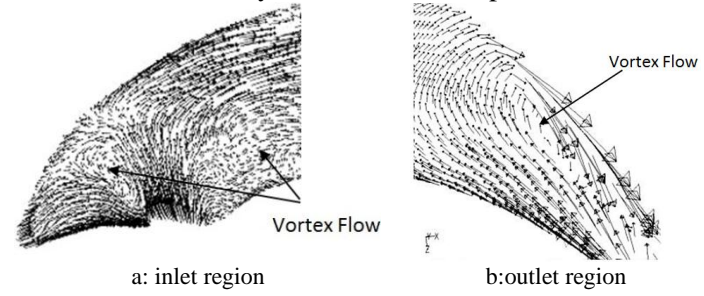


Figure 13
Distribution of the relative velocity vectors

in middle plane: 159 m³/h and 3600 rpm

CONCLUSION

By using presented procedure in this paper, complex geometry of solid bodies can be obtained. By this procedure geometry of impeller turbopump with its details and good accuracy is obtained. This approach includes the obtaining geometrical coordinates from complex passages and their modeling with CAD software simultaneously. Simulation of flow fluid in obtaining geometry presented results that are well-proportioned to the function of actual turbopump. Distribution of Relative velocity vectors show that for high rotational speeds, vortices are generated between two adjacent vanes. In addition, it is shown that low flow rates can also cause vortex generation. It is concluded that an increase in rotational speed and a simultaneous decrease of flow rate can intensify the vortex generation.

KEYWORDS

solid modeling, complex geometry, turbo pump impeller, casting

REFERENCES

- [1] King, M., Fisher, M., 2006. "A CAD-centric Approach to CFD Analysis With Discrete Features". *Computer-Aided Design & Application*, Vol. 3, Nos. 1- 4, pp. 279-288.
- [2] Schlichting, H., 1979. *Boundary Layer Theory*, Translated by Dr. J. Kestin, Seventh Edition, McGraw-Hill.
- [3] Launder, B. E. and Spalding, 1974. "The Numerical Computation of Turbulent Flow", *Comput. Methods Appl. Mech. Eng.*, Vol. 3, pp. 269-289.
- [4] Dixon., 1982. *Fluid Mechanics, Thermodynamics of Turbomachinery*, University of Liverpool.
- [5] Zhou,W., Zhao, Z., Lee,T. and Winoto, S., "Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics", *International Journal of Rotating Machinery*, 49-61, 2003.
- [6] Shepherd, D. G., *Principles of Turbomachinery*, Macmillan Publishing Co., INC. New York, 1956.