

Design and CFD Analysis of Centrifugal Pump

Tilahun Nigussie, Edessa Dribssa

Department of Mechanical Engineering, Addis Ababa Institute of Technology (AAIT), Addis Ababa, Ethiopia

tilnigu@yahoo.com

Abstract— the purpose of this paper is to identify /observe and determine the pattern of velocity profile and pressure distribution by using CFD simulation program after the 3D design and modeling of the pump is made using Vista CPD. Basically, this paper revolves around the idea of investigating the effect and distribution of velocity profile and pressure within a pump having the following specification, Head = 20 m, Flow rate = 280 m³/hr, and RPM = 1450. 3D Navier–Stokes equations were solved using ANSYS CFX. The standard $k - \varepsilon$ turbulence model was chosen for turbulence model. From the simulation results it was observed that the pressure increases gradually from impeller inlet to outlet. The static pressure on pressure side is evidently larger than that on suction side at the same impeller radius. In addition to this, it was observed that, the velocity increases from impeller inlet until it enters the volute casing. It then drops to a minimum value at outlet region.

Keywords— Centrifugal pump design, CFD Analysis, Simulation, ANSYS CFX, Vista CPD, pressure distribution, CFD-Tool.

INTRODUCTION

Centrifugal pumps which belong to wider group of fluid machines called turbo machines are the most common type of pump used to move liquids through a piping system. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward or axially into a diffuser or volute chamber, from where it exits into the downstream piping system. Centrifugal pumps are typically used for large discharge through smaller heads.

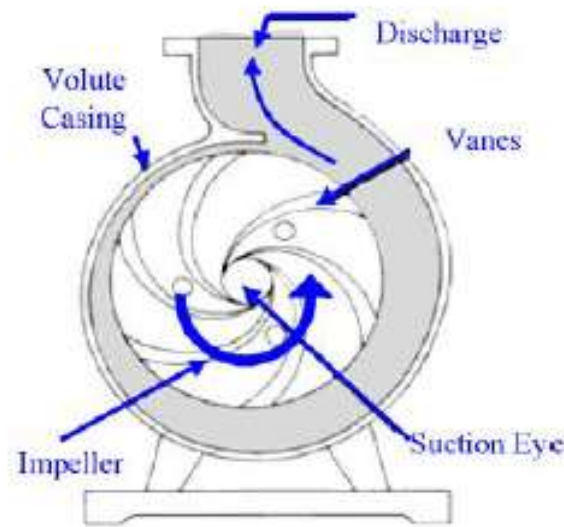


Fig 1: Liquid flow path inside a centrifugal pump [1].

Computational fluid dynamics (CFD) analysis is being increasingly applied in the design of centrifugal pumps. With the aid of the CFD approach, the complex internal flows in water pump impellers, which are not fully understood yet, can be well predicted, to speed up the pump design procedure. Thus, CFD is any important tool for pump designers. The use of CFD tools in turbo machinery industry is quite common today. Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labour intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world [2].

ANSYS Turbo system – R 14.5 which is one of the CFD tools offers a complete suite of software tools for comprehensive turbomachinery design and analysis. This system will provide streamlined workflow using Integrated, easy to use environment for all engineering simulations /Analysis using Vista TF, FLUENT, ANSYS FEA and CFX [4].

In a CFD model, the region of interest, a pump casing for example, is subdivided into a large number of cells which form the grid or mesh. In each of these cells, of which there may typically be 300,000, the PDEs can be rewritten as algebraic equations that relate the velocity, pressure, temperature, etc. in that cell to those in all of its immediate neighbors. The resulting set of equations can then be solved iteratively, yielding a complete description of the flow throughout the domain. Powerful graphical post-processors then display the results in an easily understandable way [4]. Therefore, in this paper 3D CFD analysis system using Vista CPD together with CFX code is used to simulate the fluid flow through a pump

CENTRIFUGAL PUMP DESIGN

The design of centrifugal pump is divided in two categories: Impeller Design and Volute Design. The detailed procedure of single volute casing and impeller design can be found in different literature; in this paper vista CPD for the design of centrifugal pump is used. The duty parameters required by the pump are assumed to be: 1. Head = 20 m, 2. Flow rate = 280 m³/hr, 3. RPM = 1450, 4. Density = 1000 Kg/ m³,

A. IMPELLER DESIGN USING VISTA TF V14.5

Input variables are used to give a basic starting point for the pump design. The head, volume flow, rotational speed and other parameters could be changed to the specific purpose. Various windows show the design parameters, like the angle and thickness distribution. The following fig-2 will demonstrate the entire workflow from input values in Vista CPD to final results by Vista design module. This way, manipulation of the geometry in BladeGen or BladeEditor will be possible and all the next steps will be automatically generated and results produced.

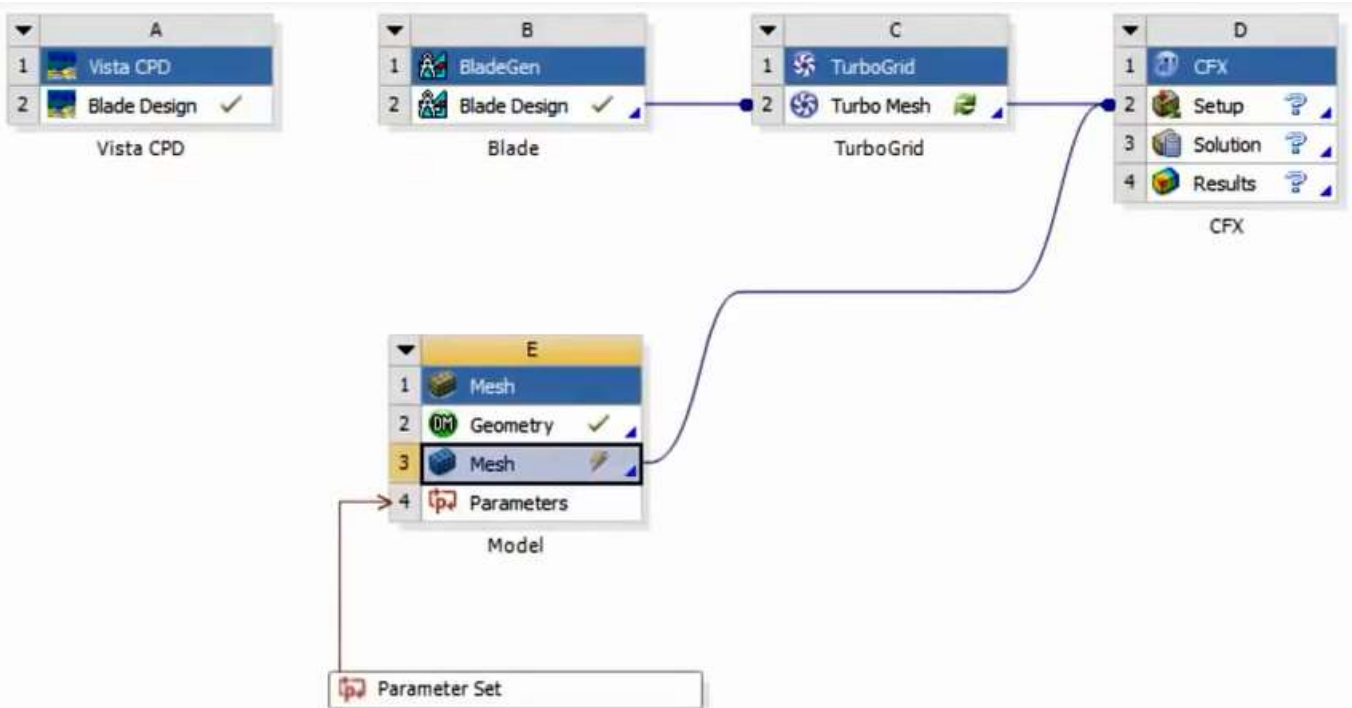


Fig.2: Flow chart of the project.

The screenshot displays the 'Impeller' design tool interface, divided into 'Operating conditions', 'Geometry', and 'Results' tabs.

Operating conditions:

- Units: SI, Imperial
- Duty:
 - Rotational speed: 1450 rpm
 - Volume flow rate: 280 m³/hr
 - Density: 1000 kg/m³
 - Head rise: 20 m
 - Inlet flow angle: 90 deg
 - Merid velocity ratio: 1.1
- Efficiencies:
 - Automatic (dropdown)
 - Hydraulic: 0.874
 - Volumetric: 0.97
 - Mechanical: 0.948
 - Pump: 0.804

Results (Overall performance):

Qs	Ns	nq	Nss	power (kw)
0.82	2242	43.4	3.15	19.5

Impeller inlet:

Dih (mm)	De (mm)	Thk (mm)
39.3	170.5	8.2

D1 (mm)	Cu1 (m/s)	Cm1 (m/s)	U1 (m/s)	W1 (m/s)	β 1 (deg)	β 1 (deg)	inc (deg)
83.1	0.00	3.55	6.31	7.24	34.52	29.37	5.15
127.3	0.00	3.95	9.67	10.44	24.19	22.21	1.98
171.5	0.00	4.34	13.02	13.72	18.44	18.44	0.00

Impeller exit:

D2 (mm)	B2 (mm)	lean (deg)	β 2 (deg)	W2 (m/s)
271.9	48.6	0.0	13.11	10.03

α 2 (deg)	C2 (m/s)	Wslip/U2	U2 (m/s)	Cu2 (m/s)
11.81	11.11	0.21	20.64	10.87

Fig.3: Input design parameters of pump using Vista CPD software.

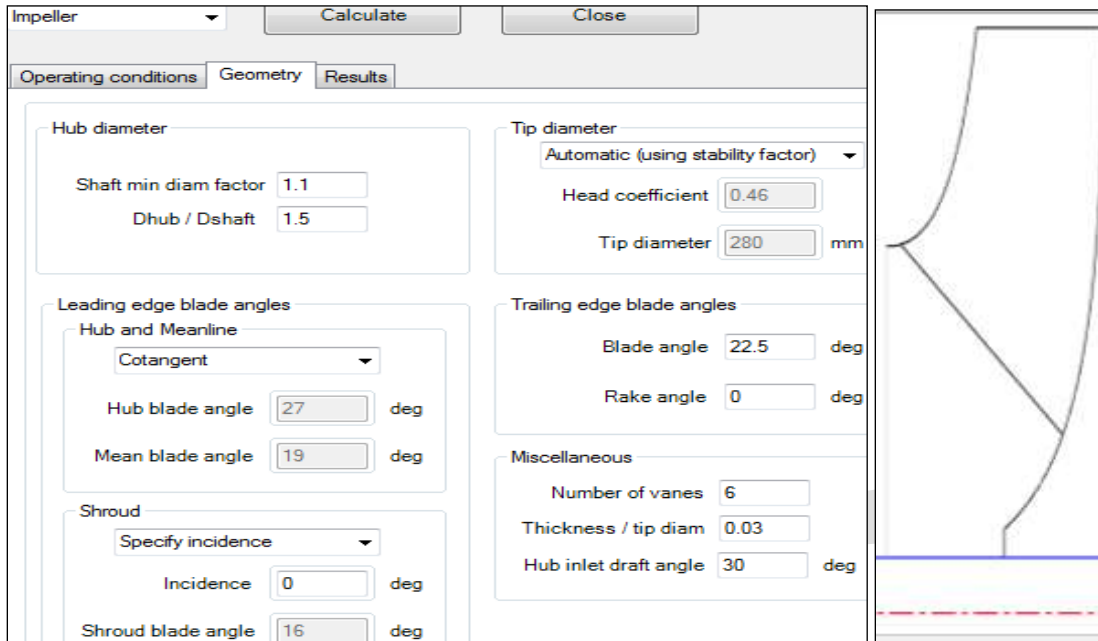


Fig.4: Geometrical parameters for Impeller of the pump using Vista CPD software

B. VOLUTE DESIGN

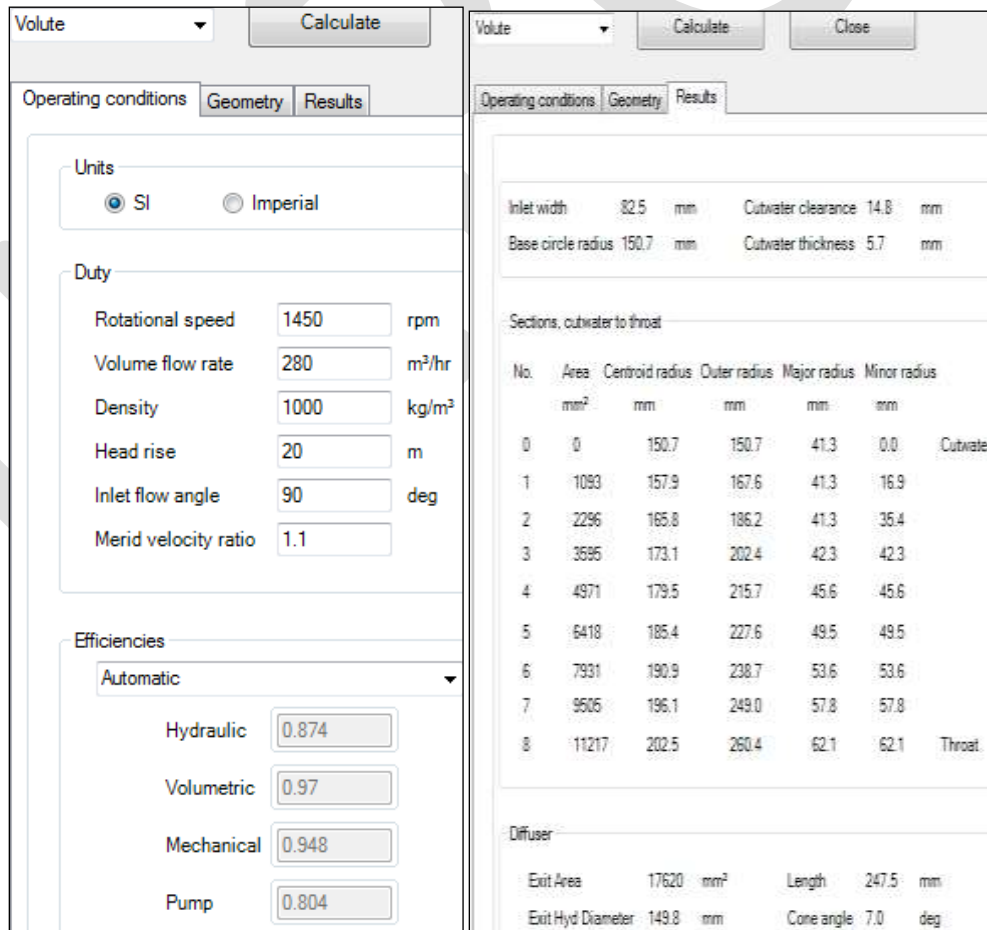


Fig.5: Output parameters using Vista CPD software.

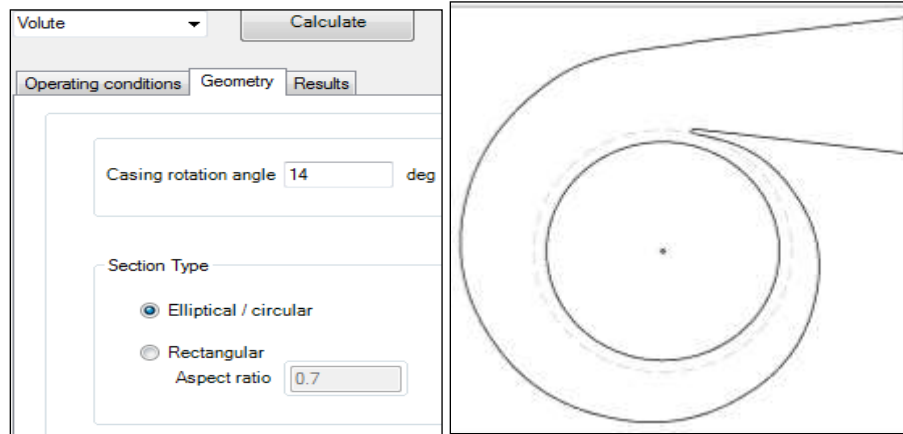


Fig.6: Geometrical parameters for volute using Vista CPD software.

MESH GENERATION

Once the pump geometry has been specified and a mesh has been created automatically, where the flow equations need to be solved.

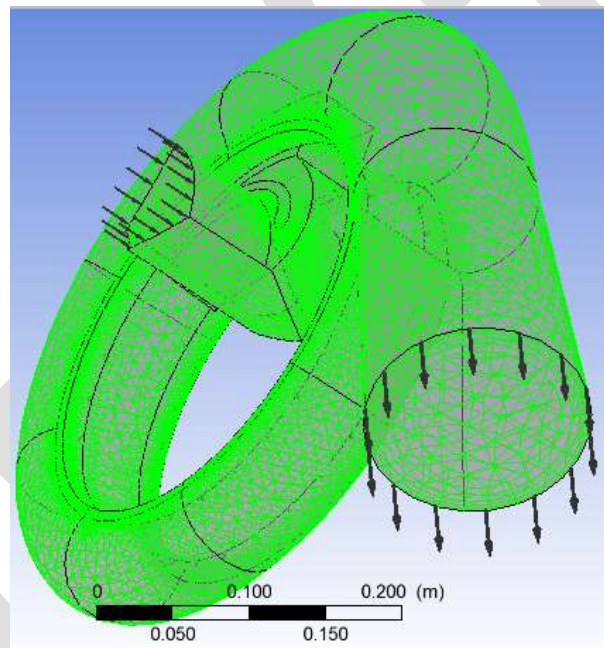


Fig.7: Automatic mesh profile for centrifugal pump.

ANALYSIS SETUP

Design points for a parametric study can be specified using the required duty of the pump in the setup steps:

Input Material: Material is also assigned to the parts of the pump as: Casing and Impeller: Aluminum alloy, Hydraulic Region: Water, Rotating part: Rotating region

Boundary Conditions: Boundary conditions are applied to the inlet and outlet of the pump i.e. 0 pa at inlet, 280 m³/hr at outlet, and 1450 RPM.

SOLUTION INITIALIZATION

Initialization in Ansys CFX is done by providing initial guess values to solve the governing equation so that the flow field variables can be solved by iteration toward the solution. The default automatic initialization for the velocity and static pressure is used to provide a start point to the solution.

CFD RESULTS

After analysis has been carried out the following results are obtained. The results are taken only when the convergence is obtained for the solution. As the solution iterated 1000 times and the pump impeller completed a full turn, following results are taken from different axis and cross-sections.

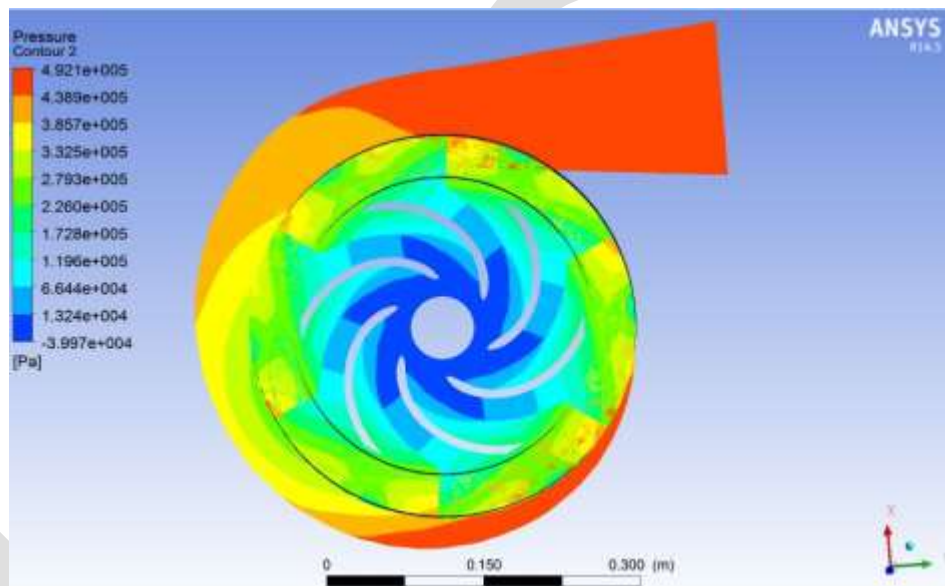


Fig.8: Pressure counter flow through the pump in the mid plane view for $Q=36\text{m}^3/\text{kg}$ (low flow rate)

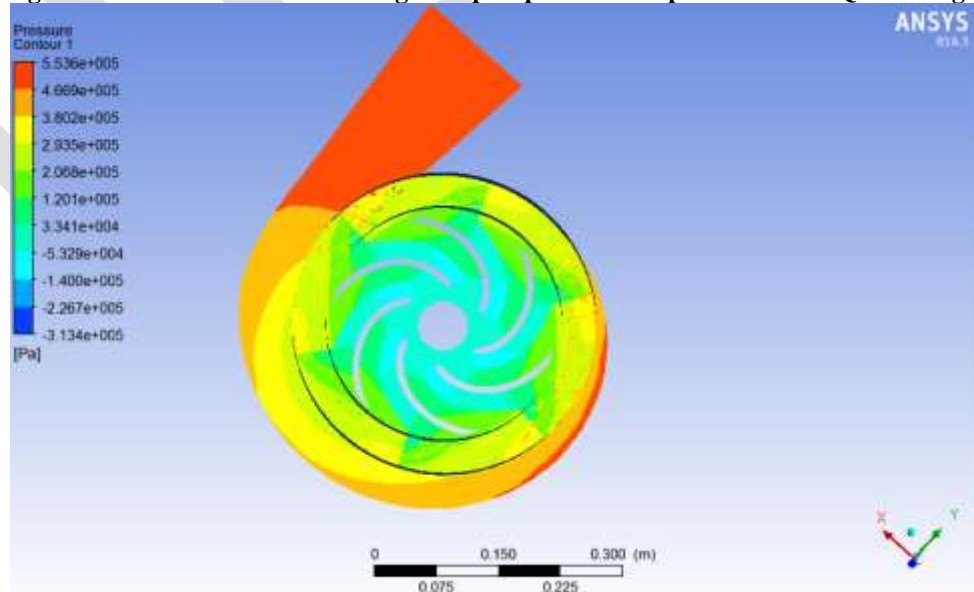


Fig.10: Pressure counter flow through the pump in the mid plane view for $Q=280\text{m}^3/\text{kg}$ (At design value of the flow rate)

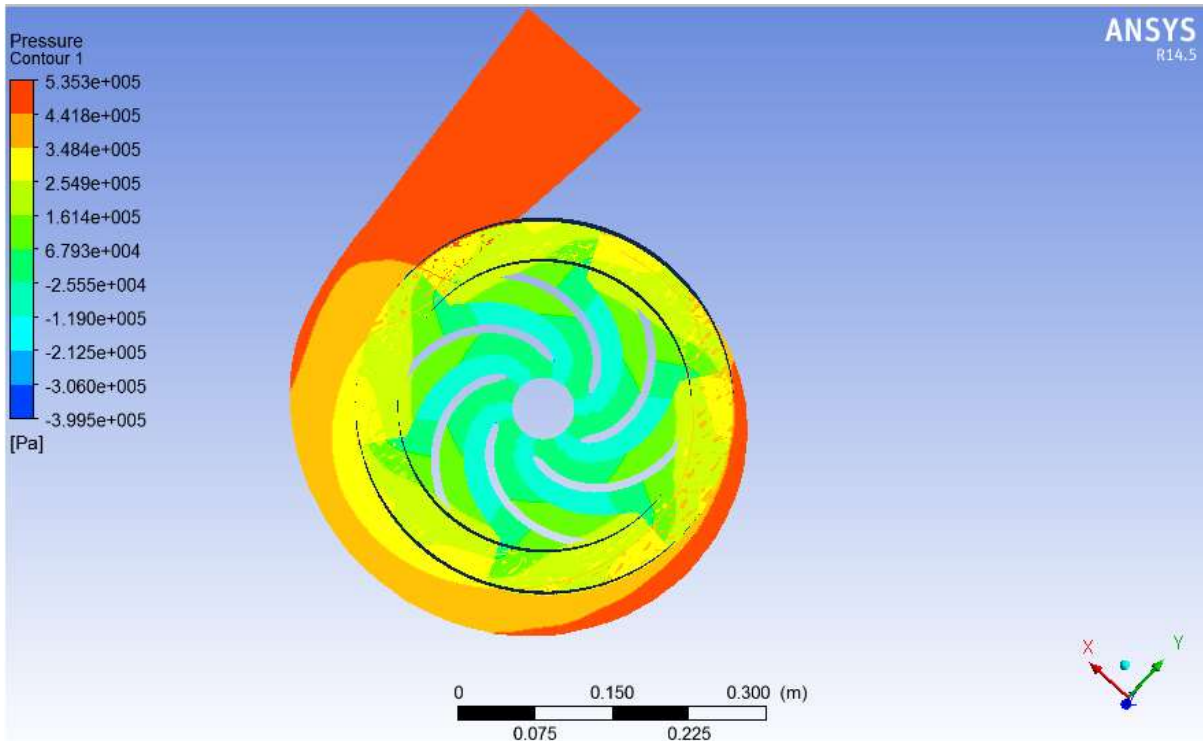


Fig.11 Pressure counter flow through the pump in the mid plane view for $Q=360\text{m}^3/\text{kg}$ (At high value of the flow rate)

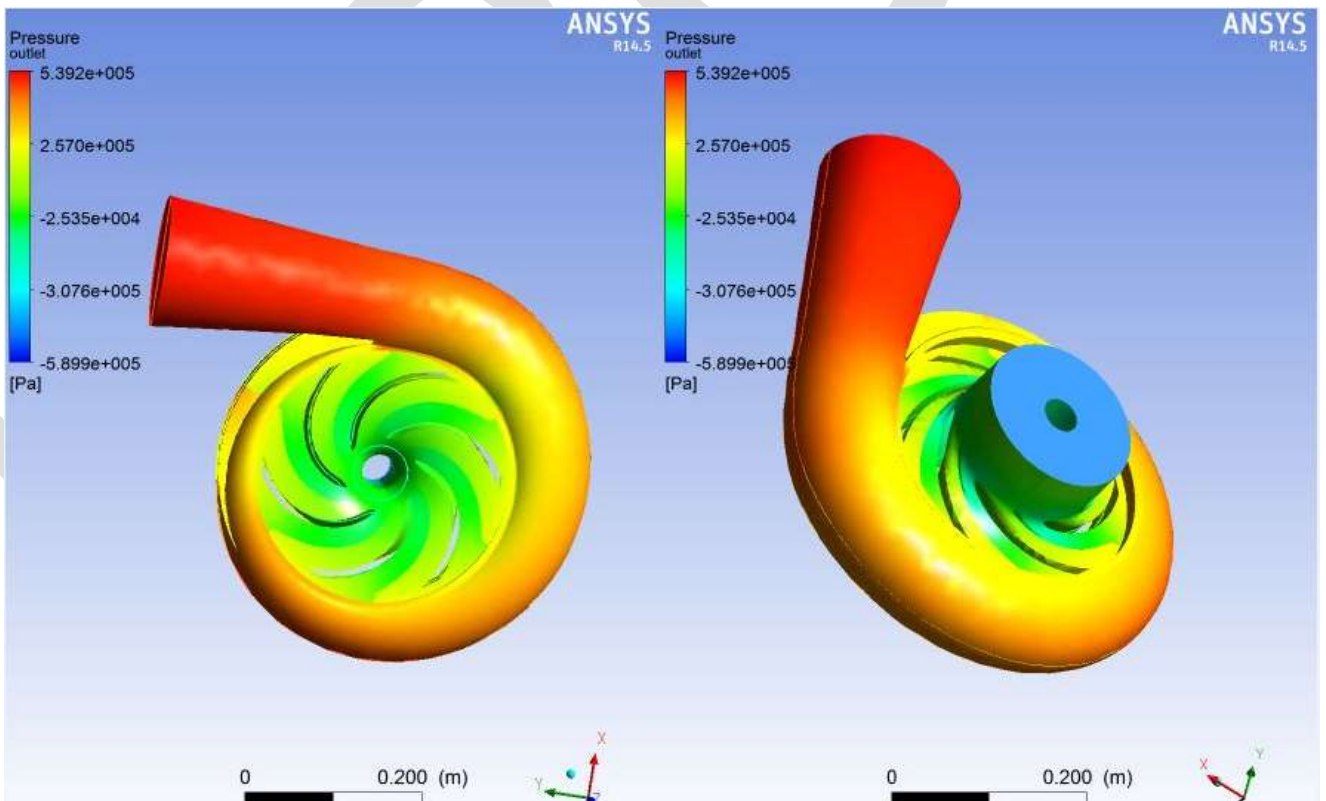


Fig.12 Pressure counters flow through the pump in the 3D view at design value of the flow rate

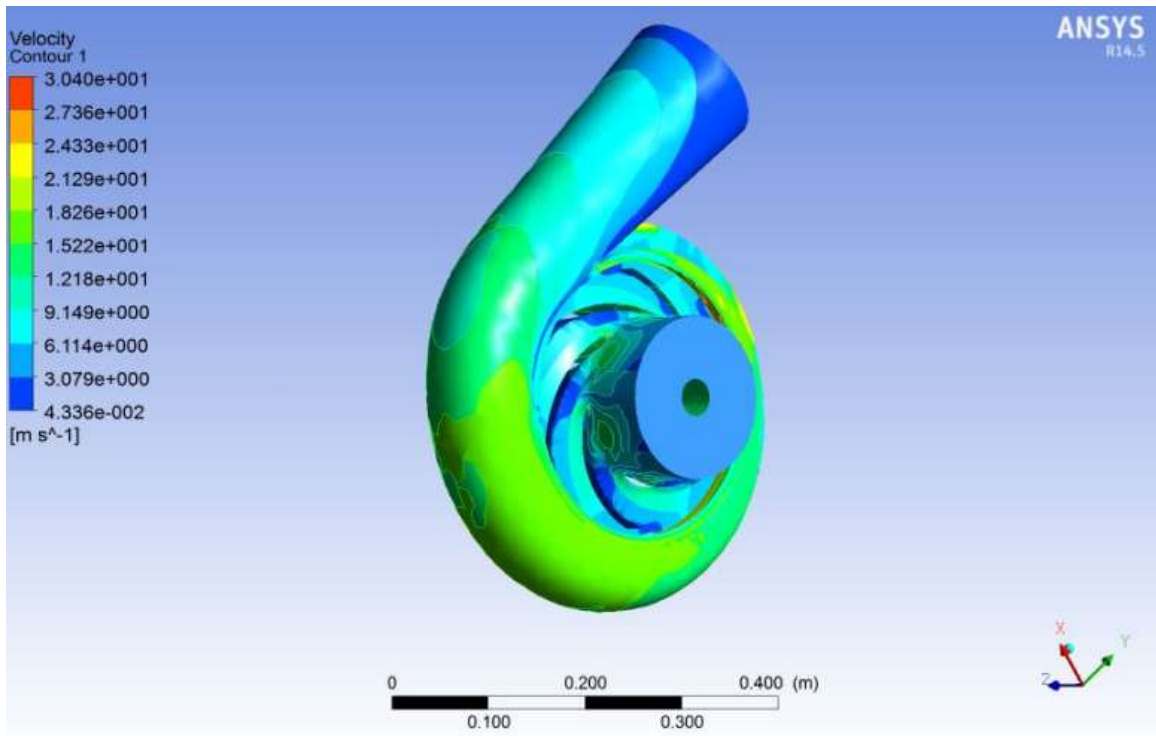


Fig.13: Velocity counters flow through the pump in the 3D view at design value of the flow rate.

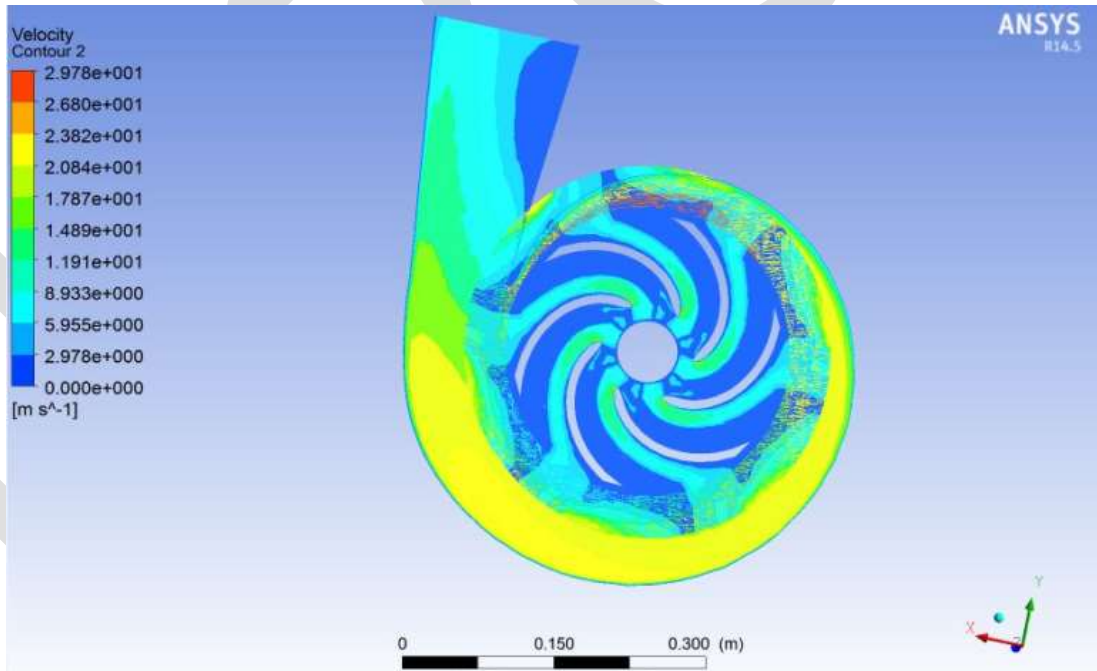


Fig.14: Velocity counters flow through the pump in the 3D mid plane view at design value of the flow rate.

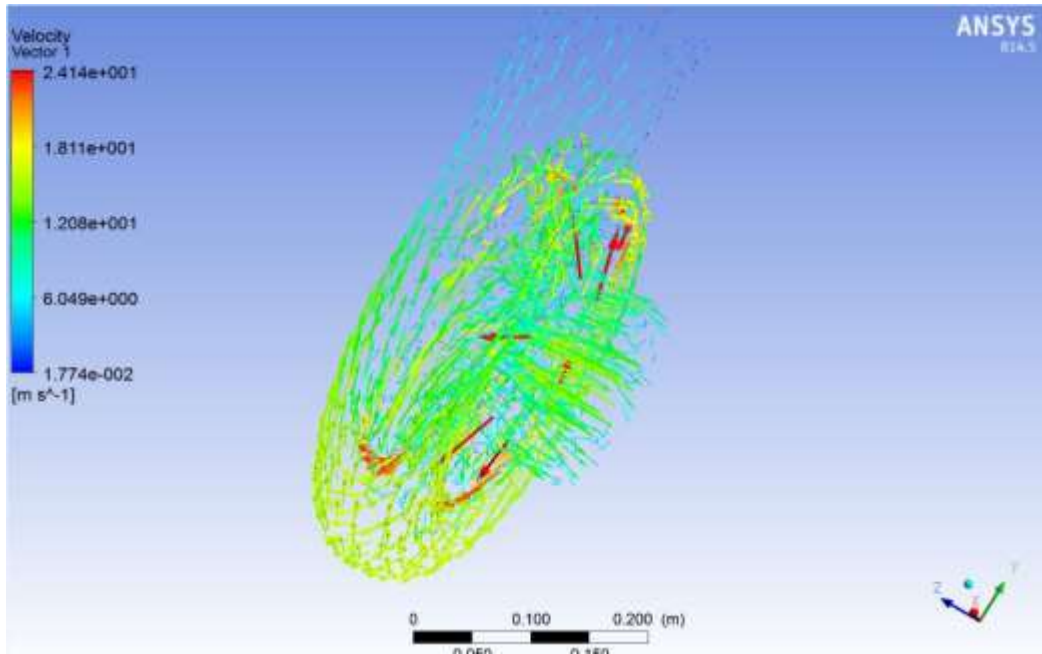


Fig.15: Velocity vectors in the 3D view at design value of the flow rate.

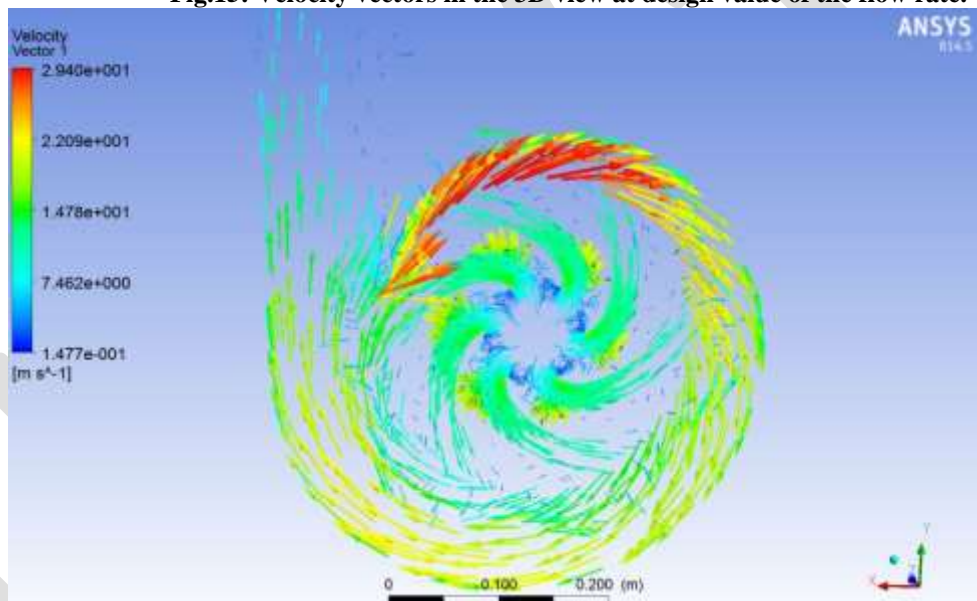


Fig.16: Velocity vectors in the 3D mid plane view at design value of the flow rate.

CONCLUSION

Experimental methods and past experience are undoubtedly important, but the most effective way to study pump performance is through Computational Fluid Dynamics (CFD).

The CFD-code (**ANSYS Turbo system – R 14.5** (Ansys CFX), version 14.5), has been used in this paper for the flow analysis of pump with end-suction volute type: The impeller and volute geometry was designed by Vista TF CPD V14.5 software by assuming the required duty parameters by the pump to be design as a case study are: Head = 20 m, Flow rate = 280 m³/hr, RPM = 1450, Density = 1000 Kg/ m³, and the model prepared has been analyzed in CFD tool CFX and its performance is analyzed at different flow rates. It is found that the design and analysis methods lead to completely very good flow field predictions. This makes the

methods useful for general performance prediction. In this way, the design can be optimized to give reduced energy consumption, lower head loss, prolonged component life and better flexibility of the system, before the prototype is even built.

REFERENCES:

- [1]. Suthep kaewnai, Manuspong Charmaoot and Somchai Wongwiset “prediction performance of radial flow type impeller centrifugal pump using CFD”. *journal of mechanical science and Technology* 23(2009) 1620-162
- [2]. Takemura T and Goto A 1996 *Journal of Turbomachinery*
- [3]. Goto A 1997 *Prediction of diffuser pump performance using 3D viscous stage calculation* FEDSM97-3340 ASME Fluids Engineering Division Summer Meeting
- [4]. ANSYS CFX basic information (<http://www.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/Fluid+Dynamics+Products/ANSYS+CFX>)
ANSYS CFX technical specification (<http://www.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/brochure/ansys-cfx-tech-specs.pdf>) WIKIPEDIA (<http://en.wikipedia.org/wiki/CFX>)