African Research Review

An International *Multidisciplinary Journal, Ethiopia*
Vol. 7 (3), Serial No. 30, July, 2013:178-195
ISSN 1994-9057 (Print) ISSN 2070--0083 (Online)
DOI: http://dx.doi.org/10.4314/afrrev.v7i3.14

Development of a 5kw Francis Turbine Runner Using Computation Fluid Dynamics

Odesola, I. F. - Department of Mechanical Engineering, University of Ibadan, Ibadan, Nigeria
E-mail: *ifodesola@yahoo.com*, *joe_idise@yahoo.com*
Tel: +2347032743197

&

Oriabre, J. I. - Department of Mechanical Engineering, University of Ibadan, Ibadan, Nigeria

Abstract

A small scale Francis turbine runner for a turbine located at Awba dam in the University of Ibadan with designed head and flow rate of 6m and 0.244m^3/s is designed. The basic design of the Francis turbine runner is completed based on basic fluid dynamics turbo machinery principles. A 2-D and 3-D steady state, single-phase CFD analysis is conducted for the runner, stay vanes, guide vanes, spiral case and draft tube of Francis turbine using two commercial CFD codes (ANSYS FLUENT and Solidworks Flow Simulation). The dimension of the runner is obtained from empirical formulas using the available head and flow rate. The runner is then optimized using CFD analysis to get required shape and performance.

Performance of turbine is predicted at different guide vane openings. Cavitation and flow separation analyses are also conducted using a 2-D, steady state and two-phase (water vapour and liquid water) in order to reduce their occurrence in the designed runner. A hydraulic efficiency of 68.2% was obtained.

Key words: *Fluid dynamics, hydro-turbines, cavitations, CFD, flow separation*

Introduction

Small scale hydropower is one of the main renewable energy sources for which Nigeria has great potentials. If properly harnessed, it will help Nigeria to meet its power demand and maintain her economic growth for the next decade. Hydraulic Turbines convert hydraulic energy of water into mechanical energy which is further converted into electrical energy. This energy obtained is known as hydro-electric power which is one of the cheapest forms of energy generation. The water enters the turbine through the guide vanes which are aligned such as to give the flow a suitable degree of swirl. The flow from the guide vanes passes through the curved passage. This will force the radial flow to axial direction. The axial flow of water with a component of swirl applies force on the blades of the rotor and loses its momentum, both linear and angular, producing torque and rotation in the shaft. There are four basic types of turbine (Francis, Kaplan, Pelton and Turgo) used for hydropower generation (Sabourin et al., 1996).

The Awba dam situated near the Faculty of Technology in the University of Ibadan, was hitherto used for flood and erosion control. The available head and flow rate at the dam have been estimated to have the capacity to generate about 10kW of electricity during the raining season(Ajuwape et al.,2011). The aim of this work therefore, is to design a Francis turbine runner capable of producing 5kW of electricity for a small- hydropower plant situated at the dam using Computational Fluid Dynamics (CFD) for the runner design.

CFD is the latest state of the art technological tool which is being used by various designers of hydraulic turbine (Schilling et al.,2002). This paper describes how a low head Francis turbine runner is designed and optimized using CFD. It starts with basic design of the runner (runner diameter, rotational speed, specific speed, guide vanes, runner blade angle, etc.) and optimization & performance evolution using CFD.

General description of the Francis turbine

The runner

The runner is the most important component of Francis turbine. Water in high pressure is fed in through the spiral case and guided by the stay and guide vanes to the runner blades. The high-pressure water enters the runner inlet with partial kinetic energy and flows through the area between two blades of the runner. By reaction principle, water pressure energy is converted into kinetic energy, which produces pressure difference between two sides of blade. This pressure difference between the two sides of blade produces torque and finally mechanical energy in the form of rotation of runner.

The guide vane

The guide vane converts partial pressure energy from available total pressure energy into kinetic energy. This develops the needed swirl which is required at the inlet of turbine runner. It also distributes the flow discharge at inlet of turbine runner equally to the runner blades. The positioning of guide vane is based on inlet velocity triangle, while height of guide vane is based on runner height.

Spiral Casing

Function of spiral casing is to equally distribute discharge at the inlet of guide vane. At the exit of spiral casing, stay vane ring is fitted. The purpose of stay vane is to give mechanical strength to spiral casing as well as provide required flow direction at inlet of guide vane (Kiran et al..2011)

Draft tube

Function of draft tube is to convert unused kinetic energy at the exit of runner into pressure energy and hence to increase the effective net head on turbine unit.

Methods of predicting hydropower turbine performance

There are basically two methods for predicting the performance of hydropower turbines: experimental and use of CFD. Prior to the advent of CFD, hydraulic turbine shape was determined through experimental research using scaled-down prototype of turbine models. The experimental approach for predicting the performance of hydro-turbine is costly and time consuming. It gives a value but it is unable to ascertain the root cause of the poor performance of the hydropower. CFD on the other hand has become today's state-of-the-art technique in fluid flow analysis. It is an effectual tool which provides detailed insight into flow characteristics in hydropower plants. It largely reduces design cost and time necessary for the completion of hydropower projects. It also offers detailed insight into and helps in resolving different fluid flow related issues such as cavitation, flow separation, etc (Jingchun et al., 2007)

Optimization of turbine runner by computational fluid dynamics

Computational Fluid Dynamics is the science of predicting fluid flow (Turbine, Pump), heat transfer, mass transfer (Perspiration, Dissolution), phase change (Freezing, Boiling, cavitation), chemical reaction(Combustion) and related phenomenon by solving mathematical equations that govern these processes using a numerical algorithm by means of computer based simulation (Kiran et al., 2011). It can provide detailed information about what is happening in a process where fluid is in motion. The technique is very powerful and spans a wide range of industrial and non-industrial applications. Recent advances in computing power, advanced computer graphics, and robust solvers make CFD a cost effective engineering tool. Four basic steps involved in any CFD analysis (modeling, meshing, problem definition and post processing of result) are described as follows:

Geometric modelling

Geometric modelling is a branch of computational geometry that studies methods and algorithms for the mathematical description of shapes. The shapes studied in geometric modelling are mostly two- or three-dimensional, although many of its tools and principles can be applied to sets of any finite dimension (Gizem, 2010). In this work, two commercial softwares (ANSYS Gambit and Solidworks) are used for making the entire model in 2-D and 3-D respectively.

Meshing

Numerical discretization or Meshing is the technique to discretize or divide the whole flow domain in a fluid flow volume into small elements. These elements consist of nodes at which unknown variables are calculated. The accuracy of the CFD solution is greatly affected by the size of elements of various components (Guoyi,2005). In this work, a triangular and tetrahedral elemental mesh is used in meshing the runner blade and other components.

Problem definition

Although the flow inside the various components of Francis turbine is unsteady and two-phase (water and vapour) in nature, steady state single-phase (only water) CFD analysis was computed due to complexity and large completion time required for such simulation. All the components have been used in analysis and complete system CFD analysis accomplished. Total discharge is specified as inlet boundary condition for the spiral casing while atmospheric pressure is specified as boundary condition for the outlet via draft tube.

Post processing

Post-processing is defined as the art of results representation. The result obtained in the analysis is presented in different formats for proper interpretation.

Materials and methods

Hydropower projects are designed on the basis of resources allocated to the project and other project requirements. Available resources like flow duration data and geological conditions determine the design parameters. Two design parameters which play leading role in turbine design are: operational head range and discharge variation. Turbine design and selection are then possible based on the design operation condition. The design operation condition implies the design head (H_d) and the design discharge (Q_d) available for each turbine.

Each hydropower project necessitates a different turbine design. In this work, a methodology is developed, as shown in figure 1 below to obtain a Francis turbine design specific for this project.

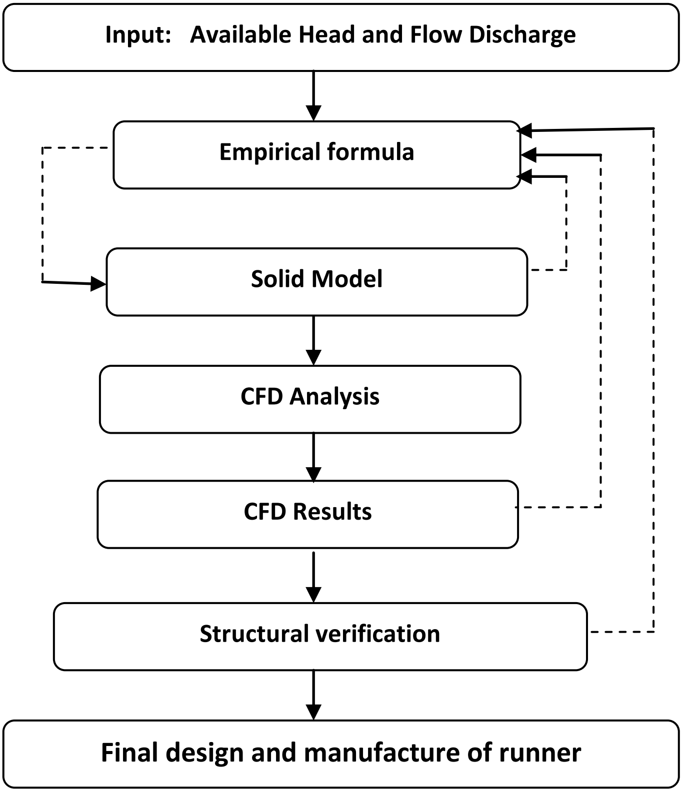

Figure 1: Design Methodology Chart

The first step of the methodology is the preliminary design of the turbine. The dimensions of the preliminary design are based on the net available head at inlet and the discharge in the turbine system. Empirical formulae are used in sizing the different components of the runner. The design loop which exists between the empirical formulae and the Solid Model is used to check if the solid model obtained from the empirical formulae is suitable. If the model obtained is suitable, CFD analysis is done otherwise the empirical formulae are further worked on in order to obtain a suitable solid model.

A design loop exists between solid model design parameters and CFD simulation results. Preliminary design is enhanced by the interaction of successive CFD results with design parameters of the turbine. This is an iterative process leading to an optimum working condition of the turbine, especially of the turbine runner. A large portion of hydraulic losses (approximately 96%) occur in the runner of a Francis turbine. Therefore, the main design work will focus on the runner blades.

Basic design of a Francis turbine runner

The following equations were used in sizing the different components of the runner.

$$P_d = \eta \rho g Q_d H_d \qquad (1)$$

$$n = n_q \frac{H_d^{1.25}}{P_d^{0.5}} \qquad (2)$$

$$n_q = \frac{C_{nq}}{H_d^{0.535}} \qquad (3)$$

$$C_{nq} = \min\left(2600; 2600 - \frac{(200000 - P_d)}{365}\right) \qquad (4)$$

$$n_{sync} = \frac{120f}{2(number\ of\ poles)} \qquad (5)$$

$$n_s = n_{sync} \frac{P_d^{0.5}}{H_d^{1.25}} \qquad (6)$$

$$D_2 = \sqrt{\frac{4Q_d}{\pi C_{m2}}} \qquad (7)$$

$$C_{m2} = u_2 \tan \beta_2 \qquad (8)$$

$$D_1 = 1.5 D_2 \qquad (9)$$

$$D_s = 0.21 D_1 \quad (10)$$

$$\tan(\beta_1 - 90^0) = \frac{U_1 - V_{w1}}{V_{r1}} \quad (11)$$

$$\tan(\beta_2 - 90^0) = \frac{U_2 - V_{w2}}{V_{f2}} \quad (12)$$

$$H = \frac{P}{\rho g Q} = \frac{U_1 V_{w1} - U_2 V_{w2}}{g} \quad (13)$$

Computational Models

Conservation equations

The turbo-machinery industry has experienced tremendous advances through the direct approach applying Euler 3D and viscous 3D Navier–Stokes methods to solve the flow fields for a given blade geometry(Sabourin et al.,1996, Song et al.,1996). Applying the Reynolds averaging approach for turbulence modeling to the Navier-Stokes, the mass and momentum conservation equation can be written in Cartesian tensor form as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho . u_i) = 0 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots..(14)$$

$$\frac{\partial}{\partial t}(pu_i) + \frac{\partial}{\partial x_j}(pu_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_j}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_j}{\partial x_j}\right)\right] + \frac{\partial}{\partial x_j}\left(-p\overline{u_i u_j}\right) \quad \ldots (15)$$

Solution algorithm

The above set of equations is solved in ANSYS FLUENT using standard finite volume techniques. Equations are integrated over the individual computational cells. A pressure based solver is used with a standard k-ε turbulence model and enhanced wall treatment for near wall conditions. The

Indexed African Journals Online: www.ajol.info

second upwind scheme is used for all discretization domain (momentum, turbulent kinetic energy, and turbulent dissipation energy) to calculate convective flux on the boundary surfaces of control volumes. The SIMPLE algorithm, which uses a relationship between velocity and pressure corrections, is used to enforce mass conservation and to obtain the pressure field.

Boundary conditions

Pressure inlet and mass flow outlet conditions are used for runner simulations for rotating components of turbomachinery simulations. Solid boundaries are given non-slip condition. CFD analysis is performed for spiral case with mass flow rate given as inlet boundary condition.

Convergence criteria

The convergence criteria are the maximum residual value of less than 10^{-5} while the inlet volumetric flow rate and runner blade torque remain unchanged despite further iteration. Unchanging values for volumetric flow rate result in stable boundary conditions and allow the calculation of hydraulic characteristic for the selected operating point when the convergence criteria are met.

Results and discussions

The runner solid model obtained is shown in figure 2, while figure 3 shows the runner blade while figure 4 shows an assembly of the runner, guide vane and stay vane.

Different simulations were carried out in order to achieve specific results. Simulation 1 which was done in Solidworks flow simulation is focused on the accurate description of the flow inside the spiral case and it is aimed at providing a good inlet conditions to simulation 2. The result of simulation 1 is shown in fig. 5. The flow in the spiral case must be accelerated from inlet to outlet. The result obtained was as expected.

Simulation 2 gives detailed flow analysis of the complete turbine. The setup for this simulation is as shown in figure 6. It was done also in Solidworks flow simulation. It is expected that minimum velocities should be recorded at the exit of the draft tube. From the result, minimum velocities were recorded at the exit of the draft tube as expected. Simulation 2 is repeated for different conditions of flow, blade shape, and blade angle in order to meet overall turbine requirements.

The result of the von Mises stress distribution and ultimate displacement of the blade is shown in figure 7 and figure 8. From the results, the maximum stress on the blade was 42568380N/m^2 while yield strength for the material was 292000000N/m^2. The result showed that the calculated maximum stress on blade can be tolerated.

Figure 9 shows the variation of factor of safety for different materials used in fabricating turbine runner. Aluminum alloys had the least values, while stainless steel had the highest values. This showed that stainless steel will withstand more pressure force than aluminum alloys as such our choice of material will be stainless steel.

Due to computational limitation, a 2-D analysis was carried in other to design against certain flow related issues such as cavitation and flow separation. Figure 10 shows a meshed turbine setup in ANSYS Gambit, while figure 11 shows the turbine setup read into ANSYS FLUENT. Figure 12, shows the convergence result of simulation in FLUENT. If a peak pressure occurs at the blade leading edge, the blade angle is changed at the blade leading edge, this peak pressure is eliminated and a smooth distribution is obtained as shown in Figure 13.

To numerically determine the hill chart of the Francis Turbine the simulations for different opening percentages and different volumetric flows for each opening was performed. For 80% opening, five simulations were run with volumetric flows of 0.244, 0.24, 0.2, 0.15 and 0.1m^3/s. The same volumetric flows were considered for the openings of 60%, 40%, 30% and 25%. Twenty five simulations for the same head of 6m were carried out. Input pressure of 160160N/m^3 was used for the simulations. This pressure was reduced in steps of 100N/m^3 to 158260N/m^2. A trial and error approach was used to modify blade geometric shapes in order to increase blade efficiency. The overall efficiency of the turbine was determined based on the fundamental equations.

Conclusions

The flow through a 5kW Francis turbine runner for Awba dam in the University of Ibadan-Nigeria was modeled according to available technical specifications enabling fluid flow simulations. Certain fluid flow related issues such as flow separation and cavitation were largely reduced since these cannot be totally eradicated. An efficiency of 68.2% was obtained. This efficiency though low is accepted due to very large friction losses associated with low head. The runner is fabricated using locally available materials and technology in other to encourage local manufacture of the turbine runner.

References

Ajuwape T. & Ismail O. S. (2011). "Design and Construction of a 5kW Turbine for a Proposed Micro Hydroelectric Power Plant Installation at Awba Dam University of Ibadan". *International Journal of Electrical and Power Engineering.* Vol. 5, Issue: 3, Pg. 131-138.

Kiran P., Jaymin D., Vishal C. & Shahil C. (2011). "Development of Francis Turbine using Computational Fluid Dynamics. The 11th Asian International Conference on Fluid Machinery and Paper number AICFM_TM_015. The 3rd Fluid Power Technology Exhibition November 21-23; IIT Madras, Chennai, India.

Gizem O. (2010). 'Utilization of CFD Tools in the Design Process of a Francis Turbine.' A Thesis submitted to The Graduate School of Natural and Applied Sciences of Middle East Technical University, Turkey.

Guoyi P (2005). A practical combined computation method of mean through-flow for 3D inverse design of hydraulic turbomachinery blades, *ASME Journal of fluid Engineering.*

Jingchun W, Katsumasa S, Kiyohito T, Kazuo N, and Joushirou S (2007), "CFD-Based Design Optimization for Hydro Turbines". *Journal of Fluids Engineering*, Vol. 129/159. Pg. 159 – 168.

Ruprecht A, Maihöfer M, Heitele M and Helmrich T (2002). "Massively parallel computation of the flow in hydro turbines. 21st IAHR Symposium on Hydraulic Machinery and Systems, September 9–12; in Lausanne, CH.

Sabourin M, Labrecque Y, & Henau V, (1996). "From Components to Complete Turbine Numerical Simulation," Proceedings of 18th IAHR Symposium on Hydraulic Machinery and Cavitation, Iberdrola, pp. 248–256.

Schilling R, Thum S, Müller N, Krämer S, Riedel N, and Moser W, (2002). "Design Optimization of Hydraulic Machinery Bladings by Multi Level CFD-Technique," Proceedings of the 21st IAHR Symposium on Hydraulic Machinery and Systems, Lausanne, Switzerland.

Song, C. C. S., Chen X, Ikohagi T, Sato J, Shinmei K, & Tani, K. (1996). "Simulation of Flow through Francis Turbine by LES Method," Proceedings of 18th IAHR Symposium on Hydraulic Machinery and Cavitation, Iberdrola, pp. 267–276.

Timo F. (2007). "Design of the runner of a Kaplan turbine for small hydroelectric power plants". Doctoral Thesis. Department of Environmental Engineering, Tampere University of Applied Sciences, Finland. Supervised by Jaakko Mattila.

APPENDICES

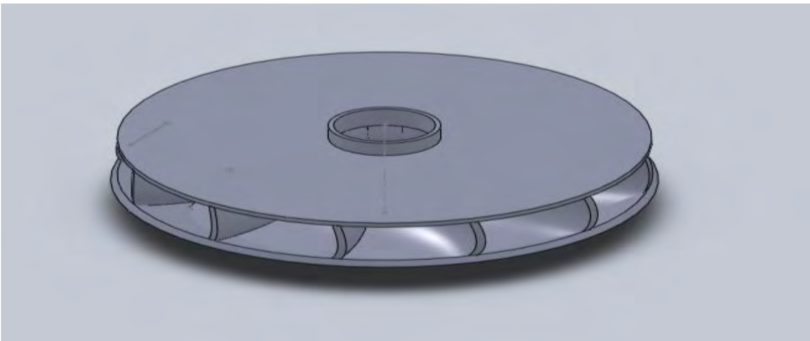

Figure 2: Solid Model of Runner in SolidWorks

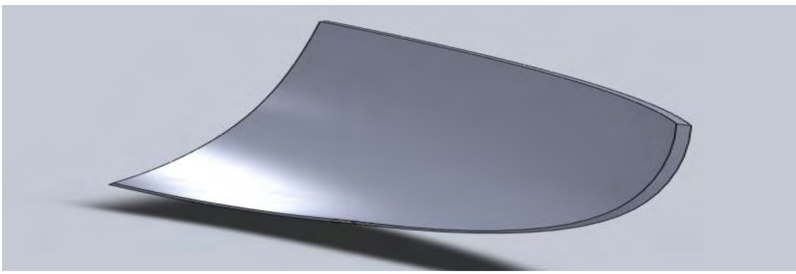

Figure 3: Solid Model of Runner blade in SolidWork

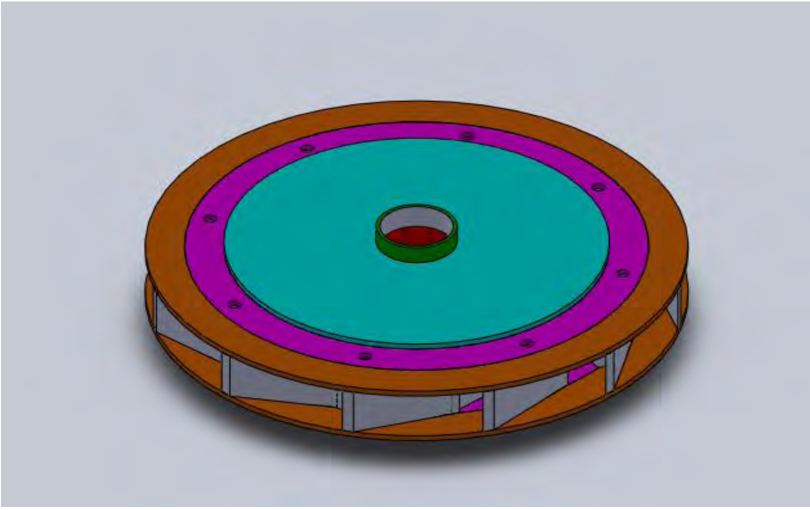

Figure 4: Solid Model of Runner, guide and stay vane in SolidWorks

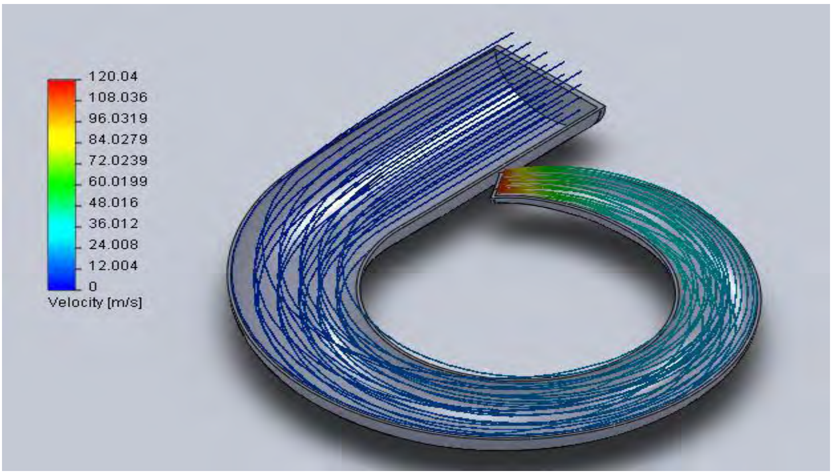

Figure 5: Showing spiral case simulation results

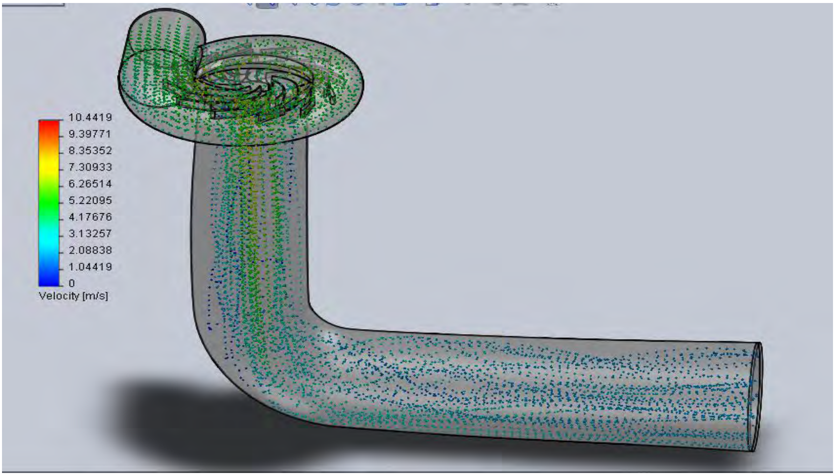

Figure 6: Showing simulation result for complete setup

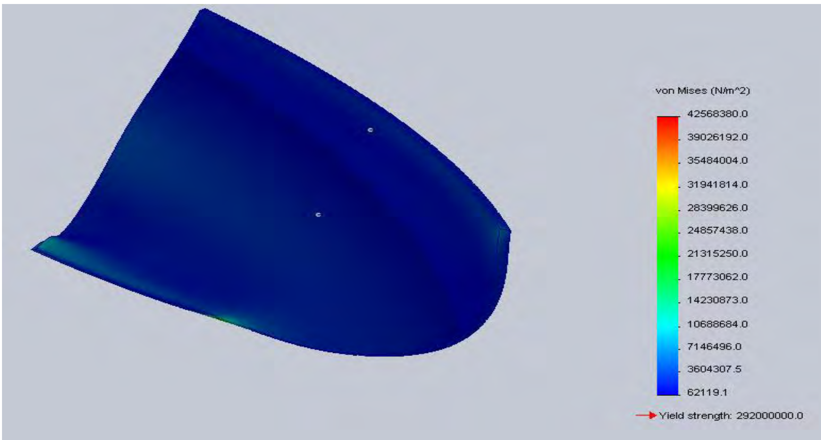

Figure 7: Vector plot showing von Mises Stress Distribution on blade surface

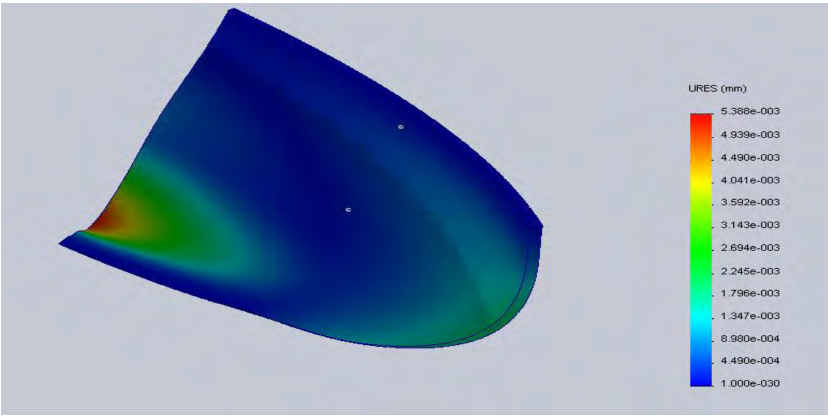

Figure 8: Vector plot showing maximum displacement of blade due to pressure force

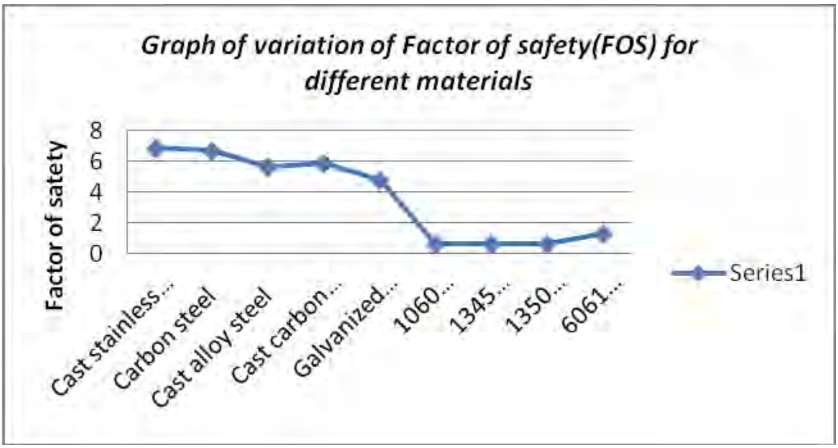

Figure 9: Graph of FOS for different materials

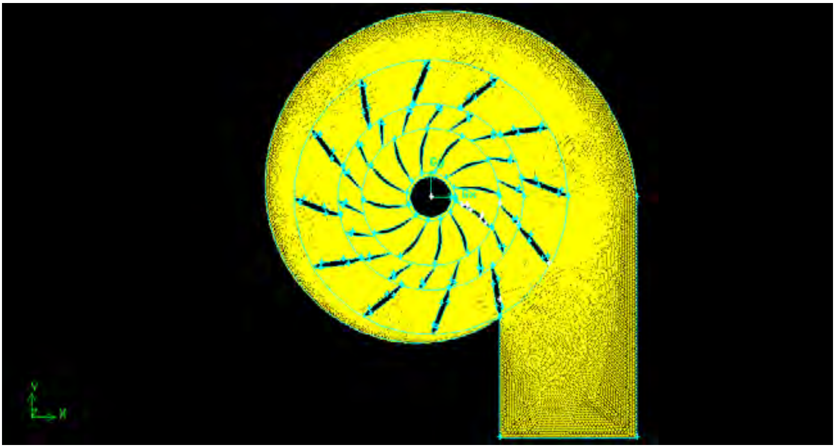

Figure 10: 2-D Geometry in ANSYS Gambit

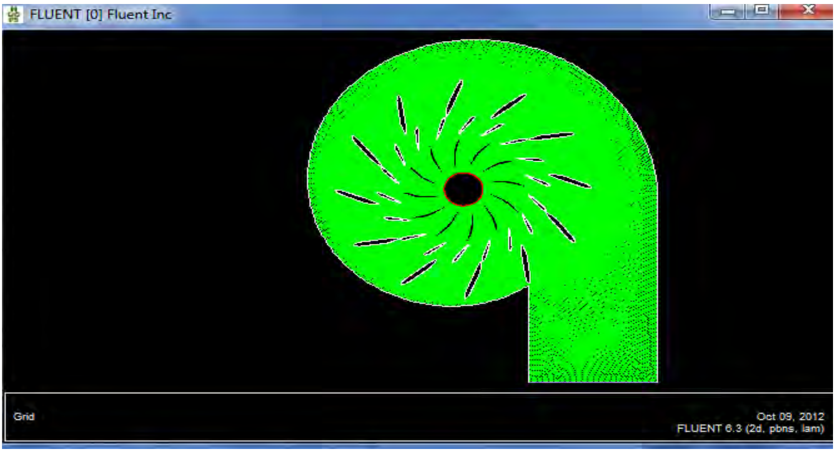

Figure 11: 2-D Geometry in ANSYS FLUENT

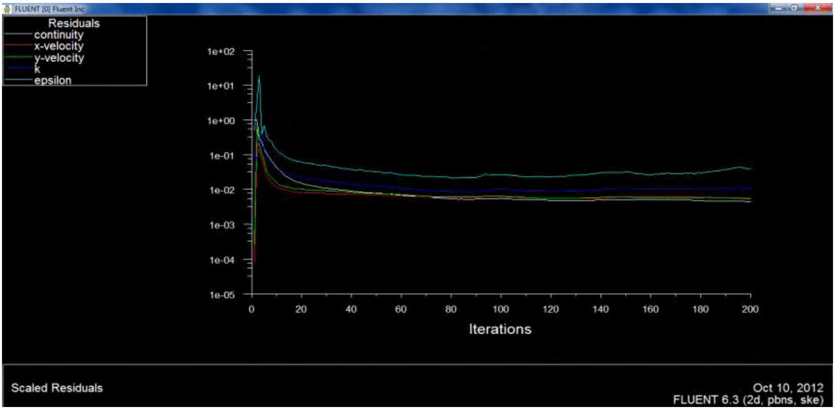

Figure 12: Convergence result of simulation in ANSYS FLUENT

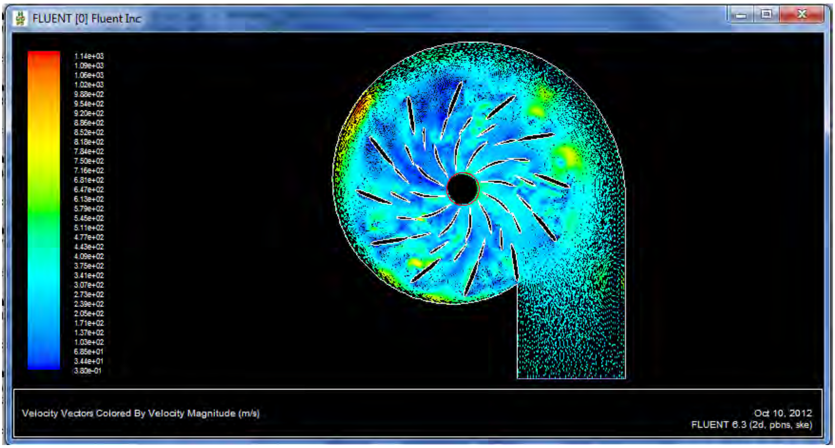

Figure 13: Result of simulation in ANSYS FLUENT showing velocity vector

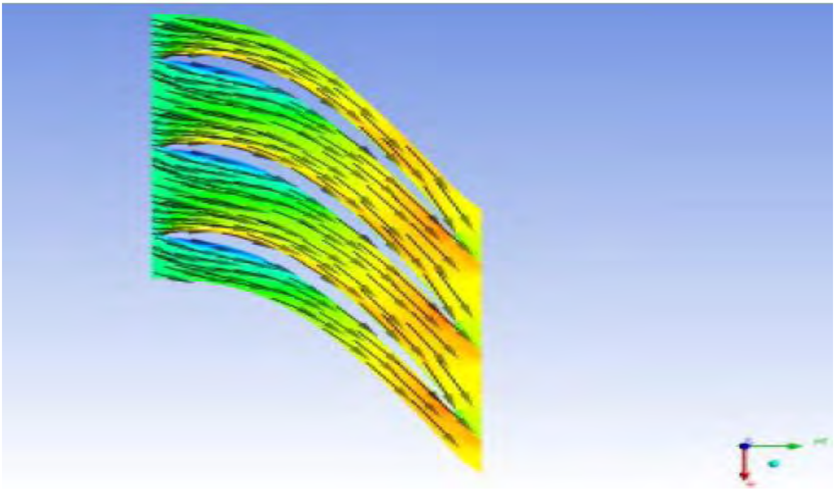

Figure 14: Velocity vectors for meridional section

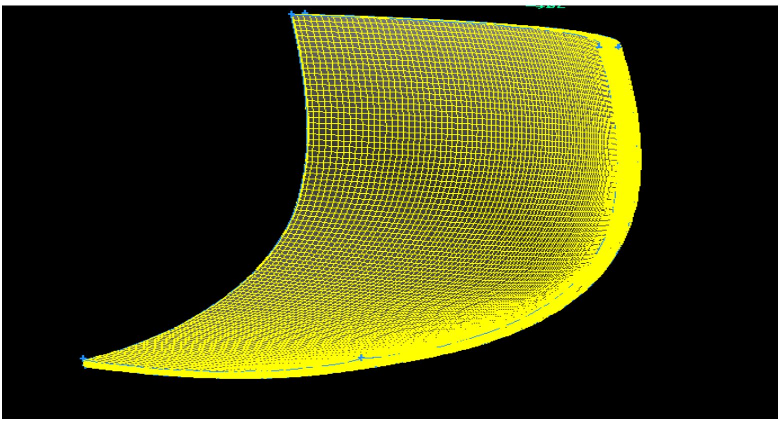

Fig.15: Solid Model of Runner, guide and stay vane in SolidWorks