International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue IX, September 2017- Available at www.ijraset.com

Fluid Structure Interaction (FSI) Analysis of Francis Turbine for Low Head and High Specific Speed Appilication

P.Swathi[1], N.Abhishek[2], Dr.S.Adhinarayan3
P.G student[1], Assistant Professor[2], Head of the Department[3] M.V.G.R COLLEGE OF ENGG (A), VIZIANAGARAM[1]

Abstract: In this paper analysis on Francis turbine runner, the growth of computational mechanics. The virtual hydraulic machines are becoming more and more realistic to get minor details of flow, which is not possible in model testing. In the present work, 3D TURBULENT FLOW analysis in hydraulic Francis turbine runner with constant speed at different head using ANSYS CFX (CFD). The pressure and velocity distribution of the runner, which is effects on the runner performance such as power output and mean efficiency point and discharge of the turbine runner. the flow turbulence model is k-ε used .The flow induced stress filed in Francis turbine runner blade using FSI, the model reduces to one blade due to the periodical symmetry of the runner. the pressure obtained from (CFD) is applied as a mechanical load on blade surface in the static structural analysis. The load acting on blade with a different operating conditions, which cause stress distribution occurs between the blade and crown it is near to the trail edge of blade are presented. And to conduct modal analysis on a Francis runner blade, to accurate predication of the natural frequency and mode shapes of runner blade is carried out two different materials like stainless and copper alloy. The highest natural frequencies occurred in stainless steel.
For this project design is done in creo-2 and analysis is done in CFD, FSI done in ANSYS workbench 17.0
Key words: Francis turbine runner, CFD analysis k-ε model

I. INTRODUCTION

To increasing efficiency of hydraulic machines and turbine performance has been an essential part for the industry. In this to develop the runner performance of turbine runner with different design consideration have been used inward radial flow of reaction turbine were inverted by James Francis in 1855 [1].

For simulation of Francis turbine runner by using CFD. CFD is a one of the flow fluid simulation for the turbines to predicate accurate results and to improvement of turbine performance. Although their methods very useful for analysis with design configuration [2].

In many hydraulic turbines are experienced crack in the runner blade, that the crack results from static and dynamic stress on the runner, the static stress are calculated by using FSI analysis method on the runner blade. However, the runner pressure loading pulsation acting on the runner blade by using FSI analysis [3].

In this paper, a computational fluid dynamics simulation of the flow passage acting on a single set of Francis turbine runner blades are performed. And to obtain pressure and velocity distribution on the runner blade due to effect on power output, best efficiency point and discharge variable on the turbine by using CFD.

The K-ε turbulence model used for accurate result. From the CFD pressure imported on the runner blade, a pressure is acting as a mechanical load to calculated stress distribution on the runner blade with different operating conditions by using FSI.

II. DESIGN AND METHODOLOGY

Pro-E software is used for the generation of model which is further imported into ANSYS turbo grid for a mesh generation. After properly meshing a model in ANSYS CFX 17.0. The unstructured tetrahedral meshing is generated in ANSYS CFD software.

Francis turbine runner consist of 18 blades; rotor reference diameter is 2.54m, rotational speed of the runner is specified R- 25.13 RPS, head 40,50 respectively based on the parameters simulated a runner to obtain mean efficiency point, output, rated discharge of the runner, K- ε turbulence model used for accurate results.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue IX, September 2017- Available at www.ijraset.com

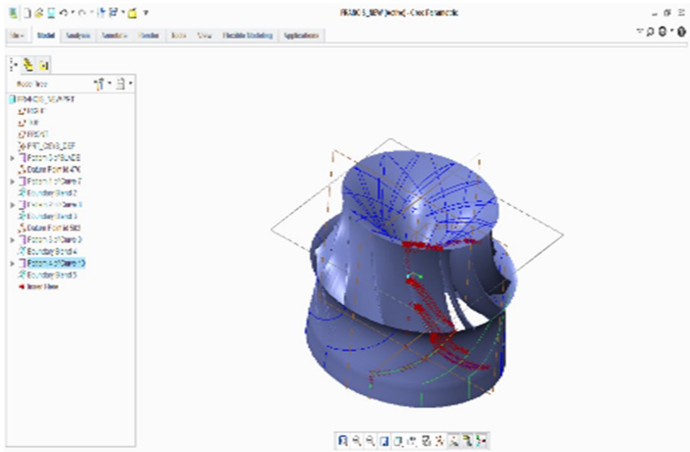

Fig 1. Modelling of Francis turbine runner

A. Boundary conditions

The inlet and outlet boundary conditions are to be specified for each path and the accuracy of solution depends on the location. Reference pressure is specified at outlet boundary conditions.

In this paper, the mass flow rate is 205.3 m^3/s , head 50 m , rotational speed 240 rpm. The static pressure at 105 Kpa is specified at outlet boundary condition. The reference pressure is taken as 0 Kpa, K- ε turbulence model used for accurate results and the walls f all domain are assumed to be smooth with no slip condition.

1) Flow simulation

The CFD analysis provides a pressure and velocity distribution across the blade passage region in the form of pressure, velocity profile. Head, discharge, power and efficiency variations are computed for the present work. Various formulae used for computation of different parameters.

$$\text{Specific speed} \qquad N_s = \text{n}\,\frac{\sqrt{P}}{H^{5/4}}$$

$$\text{Power of the runner} = \rho Q g H$$

B. Flow simulation

Runner model was imported in turbo grid CFX tool with 248384 tetrahedral elements and 265155 nodes. To determined flow rates as a boundary conditions and input, the pressure distribution in flow analysis in CFX tool shown in fig2. The maximum pressure is at the top of leading edge.

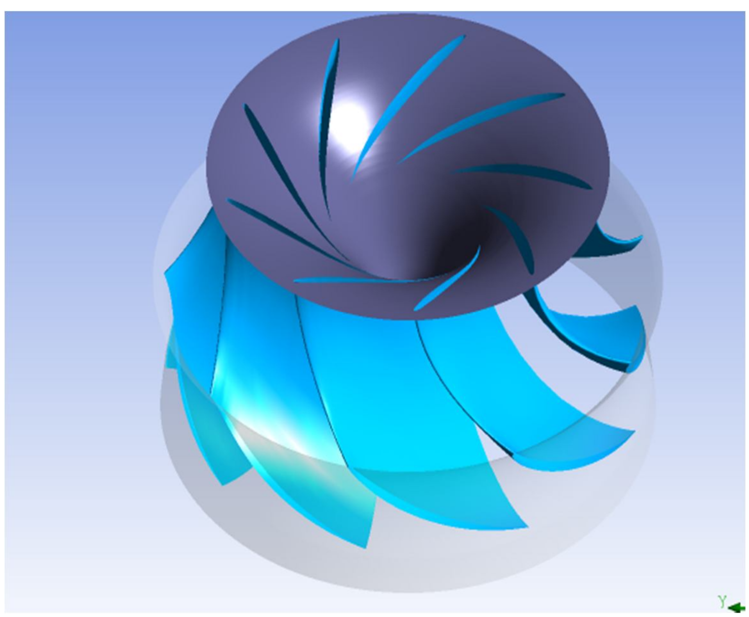

Fig.2 Final topology

III. RESULT AND DISCUSSION

A. Flow simulation

The turbine runner simulated in ANSYS CFX at different speeds, raising from 215 to 240 rpm at different water levels. Namely 40,50 m viz. The performance characteristics viz. Discharge, efficiency, power developed generated by the CFX solver are given in fig 3 to 5

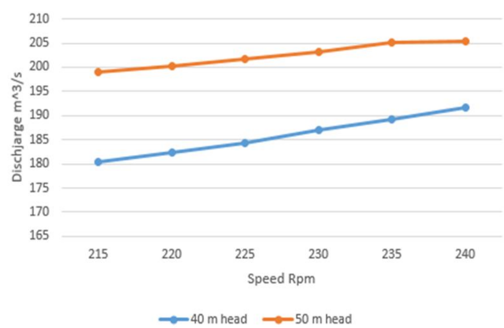

Fig .3 Speed Vs Discharge

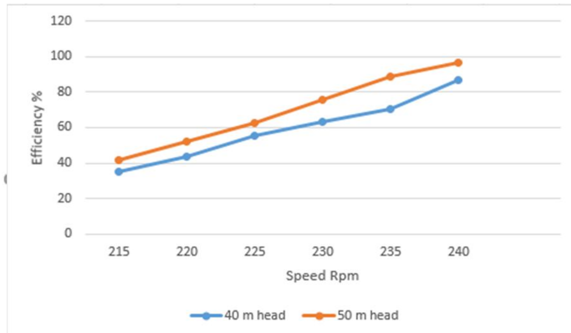

Fig.4 Speed Vs Efficiency

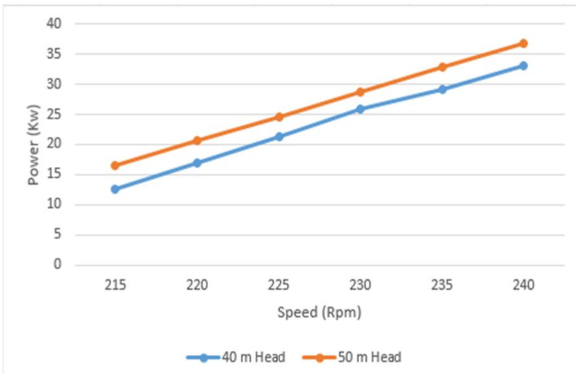

Fig.5 Speed Vs power

The CFD analysis on a single set of blade passage due to time consideration and the runner consist of a thirteen blades. For a rotating frame of the boundary conditions are set to be rotating wall conditions and the rotating velocity is maximum value is 240 m/s, at inlet stationary wall condition is default. The frame is rotating at the shroud and the boundary condition set as a wall. The flow considers as turbulent and incompressible.

The flow turbulence model is K-ε , its useful for a validation process and to predicate accurate results. The value of ε is 10^{-4}.

From this results, the mean efficiency point occurs 96.4% and power is 38.06 Mw at 240 Rpm 50 m of Head. While 87 % and power is 33.08Mw at 240 Rpm 40 m of Head respectively.

Comparison of constant speed at different heads of turbine runner the best operating point, high efficiency and power occurs at 50m of head and also pressure and velocity drops high than the 40 m of head

B. Pressure distribution at constant speed with different heads

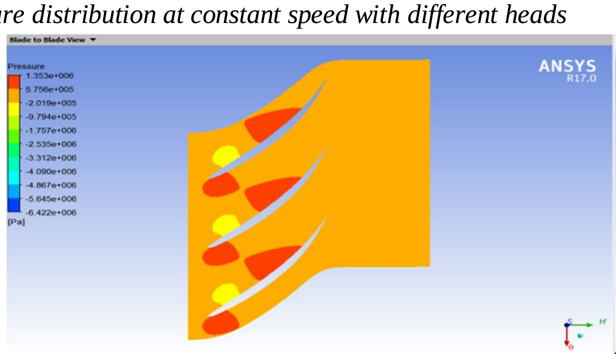

Fig.6 240- Rpm

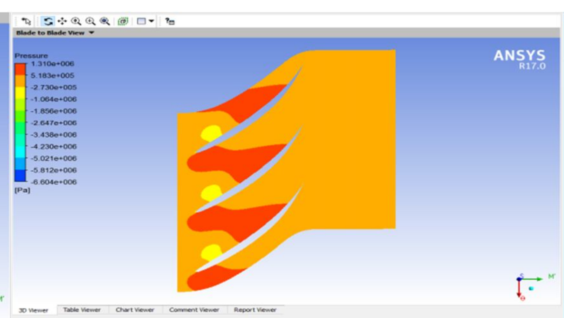

Fig.7 235- Rpm

Fig.8 230-Rpm

Fig.9 225- Rpm

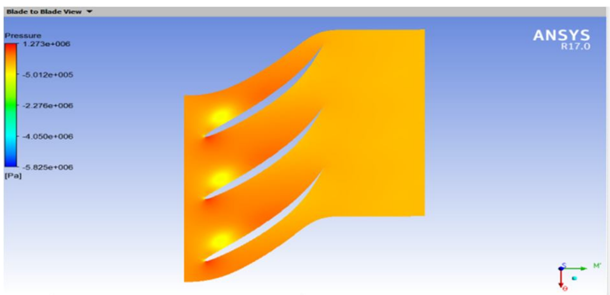

Fig.10 220- Rpm

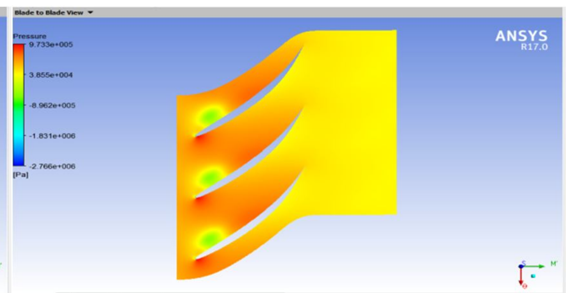

Fig.11 215- Rpm

Referring to figures from 1 to 10, it is observed that a single set of pressure drop across the blade passage from inlet to outlet is highest at (3.8) bar at 240 Rpm (from 9.6 bar to 5.75 bar) which corresponds to maximum power and efficiency while at lower speeds is 235 and 230 Rpm, 225Rpm, 220 Rpm, 215 Rpm the pressure drops in 3.7 bar resulting in total power output drop from 37 Kw at 240 Rpm to 33 Kw, 29 Kw, 24.56 Kw, 20.56 Kw,16.6 Kw.

The velocity from inlet to outlet follows similar trend i.e. velocity increase from (67.62 to 39.96) 27.66m/s at 240 rpm and highest for other low speeds 235 to 215 Rpm (70.39 to 45.6) (24.79).

Comparison the pressure and velocity distribution of constant speeds and different water head of the Francis turbine runner, the high pressure and velocity distribution occurs at 240 Rpm 50 of Head which is effect on the power output, mean efficiency point and discharge of turbine runner.

IV. CONCLUSIONS

In the present study, pressure and velocity distribution flow simulation on runner, which is effect on efficiency, output, discharge of the runner. The simulation carried out only one single blade passage different head levels with constant speeds and highest pressure and velocity distribution occurs on 50 m head 215 to 240 Rpm. Pressure is 3.8 bar at 240 Rpm, efficiency 96.34%, discharge 205.3 m^3/s, output 36.5 Kw.

Pressure distribution on top trailing edge of the turbine due to static loading conditions, the stress distribution occurs at transition between blade and crown of the turbine runner.

REFERENCES

[1] Manioj kumar Shukla et.al “CFD analysis of 3D flow for Francis turbine”Volume 1 no-2, Aug 2011.
[2] chirag Trivedi “state of the art in numerical simulation of high head francis turbine” EDF science 2016.
[3] Xiao Ruofo ,WANG zhengwei, LUO yongyao “ Dynamic stresses in a Francis turbine runner based on FSI nalysis” tsinghuva science and technology volume 13, oct 2008.
[4] Priyono sutikno and Ibrahim Khalil adam “ Design, simulation and experimental of the very low head turbine with minimum pressure and free vortex criterions” IJMME-IJENS volume 11, 2011.
[5] Professor Dr Hameed ulah mugha: Muhammad awis Hamza Mughal; Muhammad ibtsam talha “fluid structure interaction analysis of francis turbine for high head operations” volume 6, issue 4, April-2015.
[6] Leonel Alveyro Teran a, Francisco Jose Larrahondo b, Sara Aida Rodríguez a, “Performance improvement of a 500-kW Francis turbine based on CFD”. In 2015.
[7] Negrua,⇑, S. Munteanb, L. Marsavinaa, R. Susan-Resigaa, N. Pascaa “ Computation of stress distribution in a Francis turbine runner induced by fluid flow” in 2011.