HYDRO DYNAMIC PERFORMANCE OF FRANCIS RUNNER

V. RAMUDU[1], U. SUDHAKAR[2] & P. SADANANDAM[3]

[1,2]Assistant Professor, Marri Laxman Reddy Institute of Technology & Management, Dundigal, Hyderabad, India

[3]Assistant Professor, CVR College of engineering, Hyderabad, India

ABSTRACT

This Analysis describes a methodology which predicts the performance and behavior of Francis turbine runner in terms of the absolute pressure variation from inlet to outlet of the runner in the meridional view. Hydro turbines are generally tailor-made to suit different site conditions for various sites. A Francis type hydro turbine consists of five components which are volute, stay vanes, guide vanes, runner and draft tube. The performance of the turbine depends on the geometry of the components; as the runner is the main component involved in energy conversion. The flow phenomenon in hydro-turbines is simple incompressible, single phase, turbulent except that the geometry is complex and the flow is three-dimensional. The Runner geometry is more complex than the other parts of the turbine. Hence model test in laboratory is a must. Model manufacturing and testing is quite expensive and time consuming. Therefore, it is necessary to reduce the number of variants of the turbines for laboratory tests. CFD approach may be helpful in improvement of the existing efficiency measuring techniques and evaluation of the performance of hydro turbines. CFD analysis helps in reducing the number of variants with the help of computers having high speed computing ability and CFD software tools, which come under affordable prices.

KEYWORDS: Francis Turbine, Runner, CFD, Hydrodynamic Performance

INTRODUCTION

The word turbine was coined in 1828 by Claude Burdin (1790-1873) to describe the subject of an engineering competition for a waterpower source. It comes from Latin turbo, turbines, meaning a "whirling" or a "vortex," and by extension a child's top or a spindle. The precise definition is a machine in which the water moves relatively to the surfaces of the machine, as distinguished from machines in which such motion is secondary, as with a cylinder and piston. The common water wheel is a rotating machine, but not a turbine. We shall discuss many types of water-driven prime movers in this article, but mainly turbines, for which will explain the fundamental theory.

Water in nature is a useful source of energy. It comes directly in mechanical form, without the losses involved in heat engines and fuel cells, and no fuels are necessary. Solar heat evaporates water, mostly from the oceans, where it is mixed into the lower atmosphere by turbulence, and moved by the winds. Through meteorological processes, it falls on the earth as precipitation, on the oceans, but also on high ground, where it makes its way downhill to the sea, with evaporative and other losses. A cubic meter of water can give 9800 J of mechanical energy for every meter it descends, and a flow of a cubic meter per second in a fall of 1 m can provide 9800 W, or 13 hp. The efficiency of hydraulic machines can be made close to 1, so that all this energy is available, and it can be converted to electrical energy with an efficiency of over 95%.

Turbo machines are devices in which energy is transferred either to or from, a continuously flowing fluid by the dynamic action of moving blades on the runner. The word turbo or turbines is of Latin origin and implies that which spins or whirls around. A hydraulic machine is a general term used for all devices/machines handling liquids. Hydraulic machines consist of:

- Turbo machines e.g., pumps and hydraulic turbines generally known as roto dynamic machines.

- Reciprocating machines e.g., reciprocating pumps. These are known as positive displacement pumps.

- Various water-lifting devices e.g., jet pump, airlift pump, pulsometer pump and hydraulic ram.

- Pumps transmitting oil under pressure to operate hydraulic controls and systems e.g., gear pumps, constant delivery and variable delivery pumps.

Classification of Hydraulic Turbines

Turbines can be classified as high head, medium head or low head machines. Turbines are also divided by their principle way of operating and can be either impulse or reaction turbines.

Table 1

Turbine	High Head	Medium Head	Low Head
Impulse Turbine	Pelton Turbine	Cross-flow Multi-jet Pelton Turbo	Cross-flow
Reaction Turbine		Francis	Propeller Kaplan

Table 2: Operating Ranges of Hydraulic Turbines

	Pelton Turbine	Francis Turbine	Kaplan Turbine
Specific speed(rad)	0.05-0.4	0.4-2.2	1.8-5.0
Head (m)	100-1770	20-900	6-70
Maximum power(MW)	500	800	300
Optimum efficiency, per cent	90	95	94
Regulation Method	Needle value and deflector Plate	Stagger angle of guide vanes	Stagger angle of rotor bades

Performance of Turbines

Turbines are often required to work under varied conditions of head, speed, output and gate opening. As such, in order to predict their behavior it is essential to study the performance of the turbines under the varying conditions.

- The head and hence the output of the turbine may change, the speed being correspondingly adjusted so that no appreciable change in efficiency occurs, the gate opening remaining constant.

- The output may be varied by the movements of the gates or the head and speed remaining constant. These are normal operating conditions for most of the turbines.

- The head and speed may vary. Such variations are common particularly in low head units. It may, however be

stated that although the speed is permitted to vary within very narrow limits, the head may vary by even 50% or more.

- The speed may be allowed to vary by adjusting the load on the turbine, the head and gate opening remaining constant. These conditions can be developed only for laboratory turbines or those in the test plant otherwise uncommon.

- In order to predict the behavior of the turbine working under varying conditions and to facilitate comparison between the performances of the turbines of the same type but having different outputs and speeds and working under different heads, it is often convenient to express the test results in terms of certain unit quantities.

BASIC EQUATIONS IN TURBINE DESIGN

When the water passes through the runner blades, a reciprocal action takes place between them. The stream deviates from its initial direction, and its pressure on the blades causes their rotation, thereby creating the torque upon the turbine shaft.

The runner reaction on the stream at steady operational conditions of the turbine may be determined. After a time interval Δt, a quantity of water having the mass m reaches the runner blades. According to the law of mass conservation, the same quantity is discharged from the runner. By v_1 we denote the mean velocity of the water particle before entering the blade and by v_2 the mean velocity after its exit. At the entrance of the blade the mass m, having the velocity v_1 carries the momentum mv_1 and after leaving the runner it carries the momentum mv_2 away into the draft tube. It is known that the variation in the momentum of a mass in a unit of time is proportional to the force exerted on it.

This force is the runner reaction. Let the velocity have radial, axial, and tangential components v_r, v_z, and v_u respectively. The moment of the Components with respect to the turbine axis is zero. Hence, the moment of the velocity v is determined only by the second component v_u. The variation in the moment of momentum of the mass during time equals the product of the driving force (M) and there action time

$$M\Delta t = mr_2 v_{u_2} - mr_1 v_{u_1} \, or \, M = \frac{m}{\Delta t}\left(r_2 v_{u_2} - r_1 v_{u_1}\right)$$

Where

r_1 = radius of the center of gravity of the water mass at the entrance to the runner.

r_2 = radius of the center of gravity of the water mass at the exit from the runner.

The moment exerted upon the blade by the stream, i.e., the moment driving the turbine calculated in terms of absolute values, equals this moment but is of opposite sign, i.e.,

$$M_T = -M = \frac{m}{\Delta t}\left(r_1 v_{u_1} - r_2 v_{u_2}\right)$$

$$m = \frac{Q\gamma}{g}\Delta t$$

Where

Q = Discharge, γ = Specific weight, g = acceleration due to gravity

Thus,

$$M_T = \frac{Q\,\gamma}{g}\left(r_1 v_{u_1} - r_2 v_{u_2}\right)$$

The power developed at the turbine shaft is

$$N_T = M_T\,\omega = \frac{Q\,\gamma}{g}\left(r_1 v_{u_1} - r_2 v_{u_2}\right)\omega$$

Where ω = Angular velocity.

Since the peripheral velocities at the entrance and the exit are $r_1\omega = u_1$ and $r_2\omega = u_2$, respectively, then

$$N_T = \frac{Q\gamma}{g}\left(v_{u_1} u_1 - v_{u_2} u_2\right)$$

The turbine power is also $N_T = \gamma QH\eta$

By inserting N_T into the foregoing equation, we obtain

$$\gamma QH\eta = Q\gamma/g\left(r_1 v_{u_1} - r_2 v_{u_2}\right)$$

This equation may also be written as

$$\eta gH = \frac{\Gamma_1 - \Gamma_2}{2\pi}\,\omega$$

$\Gamma_1 = 2\pi v_{u1} r_1$ −circulation before the runner

$\Gamma_2 = 2\pi v_{u2} r_2$ − circulation after the runner

These equations show that the moment transmitted to the runner equals the difference between the moments of momentum, at the inlet and the outlet of the runner, of a mass Q flowing through the turbine during the unit time.

COMPUTATIONAL PROCESS

A CFD based software Numeca has been used for hydrodynamic performance of Francis runner blade. This report presents various steps .Numeca software does mesh generation of both static and rotating components of Francis turbine. As the runner is the main component involved in energy conversion.

- Creation of Geomturbo file

- Mesh generation by using auto grid

- Initial computation

- CF Views

Creation of geomTurbo File

The "geomTurbo" file format is structured in three main blocks: the header, the channel and the definition of the rows

Mesh Generation

In order to model the runner blade it is necessary to generate mesh. Mesh generation is done in AutoGrid5.

Input Data for Initial Computation

Access to FINE graphical user interface from the Auto Grid window is proposed through the Modules menu.

- Link mesh and project

- Define grid units

- Manage computation

- Set flow configuration

    o   Select water as a working fluid

    o   Select Steady to indicate that a steady solution is of interest

    o   Select Turbulent Navier-Stokes to activate the resolution of RANS equations

    o   Select Spalart-Allmaras as the turbulence model of choice

- Set boundary conditions

    o   Select Mass Flow Imposed (related patches must be grouped)

    o   Select Velocity Direction

    o   mass flow = <310.4736kg/sec><Enter>

    o   static temperature = <293k><Enter>

    o   Turbulence Viscosity = <0.0001><Enter>

- The outlet boundary condition is covered with one patch only, named as row_1_flux_1_Main_Blade_downStream_outlet.

    o   Select Pressure Imposed

    o   Select Static Pressure (related patches must be grouped).

    o   Set Static Pressure <Enter>

- Select periodicity

    o   Activate periodicity matching

    o   Select adiabatic

    o   Activate constant rotational speed

- o   Activate Pressure from extrapolation

- o   Activate computing force and torque

- Keep CFL = <3>

The CFL number drives the pseudo-time step selected to reach a steady state solution using the time-matching method integrated in the flow solver. In combination with the 4-steps Runge-Kutta integration scheme included in the flow solver, it has a theoretical stability limit close to 2.8. However, the additional implicit smoothing scheme can increase this value slightly, which justifies the default value of 3. The default value for CFL number is adequate in most cases

- Set Current Grid Level to <1 1 1><Enter>

- Set Number of Grid(s) = <2><Enter>

- Check that the Coarse Grid Initialization is activated

In addition to the multi-grid acceleration technique, the number of iterations to convergence can be significantly reduced activating the coarse grid initialization process. In that case, the simulation starts on the coarsest grid level available, leading to fast convergence. Once done, the flow solver automatically interpolates the current flow solution on the intermediate levels before the iterative process is restarted. A similar process occurs up as long as the expected finest grid level is reached. The iterative process is then pursued up as far as the convergence criterion is not reached or the maximum number of iterations is exceeded. [31]

- The Maximum number of cycles per grid level to <400>

- Set initial condition

  - o   Estimated static pressure <200000Pa>

- Control computation steering

  - o   Set Maximum Number Of Iterations = <3000><Enter>

  - o   Set Save Solution Every = <100><Enter>

  - o   Set Convergence Criteria = <-6><Enter>

- Save setting and follow the convergence

CFD analysis is carried out using Euranus solver module. Efficiency convergence history for initial runner obtained as a curve is shown as a variation with no. of iterations while the solver iterates to convergence. It is seen that when the CFD run converges there is no change in efficiency value. Thus for the original blade efficiency, based on total pressure drop and power output is 90.049%

RESULTS AND DISCUSSIONS

The following results were obtained for hydrodynamic performance of Francis Runner is as follows.

Table 3: Hydrodynamic Performance of Francis Runner

Total Efficiency (Turbine)	0.827702
Static Efficiency(Turbine)	1.81801
Mass Flow	310.34 kg/sec
Torque	-1035.4 N m
Pressure Loss (Turbine)	10.416754 Pa
Static Pressure Rise	-124240 Pa
Absolute Total Pressure Rise	-240880 Pa
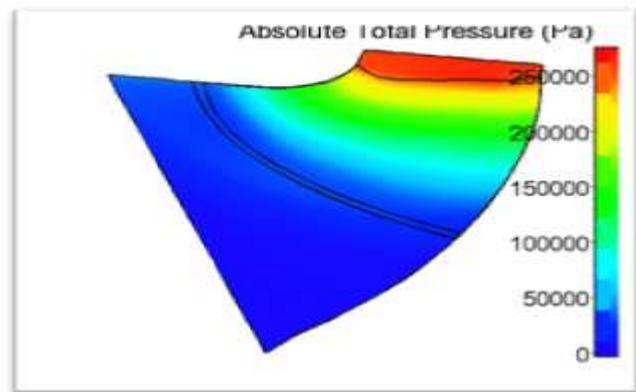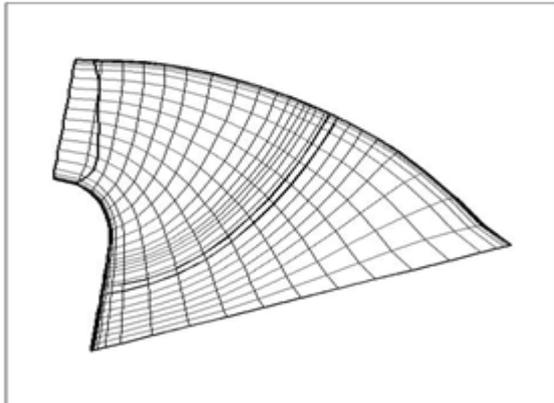
Figure 1: Approximate Model Mesh Generation in Meridional View

Figure 2: Absolute Pressure Contour Variation from Inlet to Outlet in Meridional View

The above figure 1& 2 shows the mesh generation in database and absolute total pressure variation from inlet to outlet of the runner in the meridional view. The absolute pressure rise in the approximate model is -240880Pa and the efficiency is 82.7702%
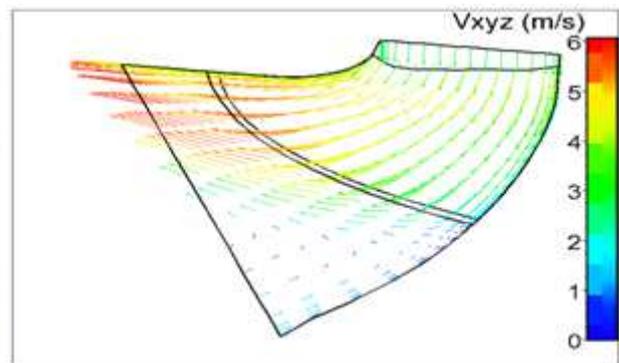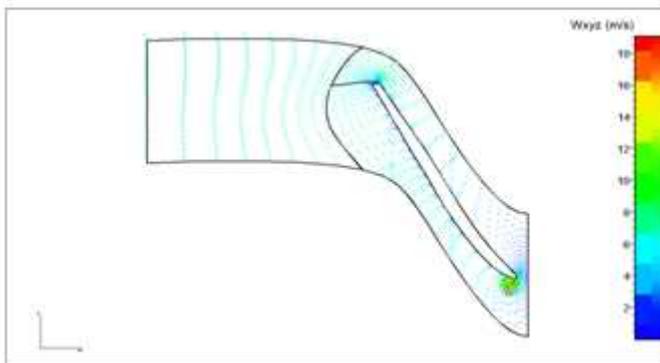
Figure 3: Relative Velocity Variation in the Blade to Blade View

Figure 4: Velocity Vector Distribution over the Runner
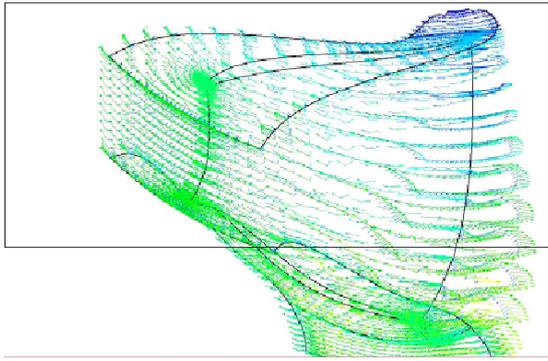
| **Figure 5: Vector Plot of Vxyz-Wxyz through the Runner Blade** | **Figure 6: Static Pressure Distribution through the Runner Blade** |
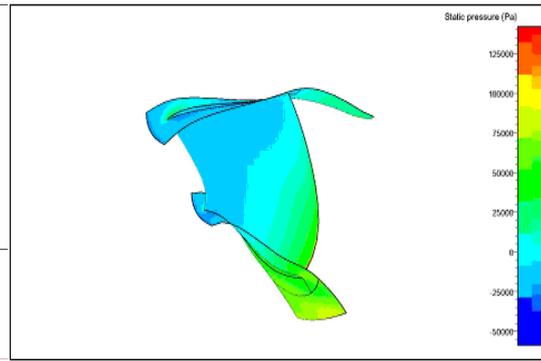
Below Figure shows the variation in the static pressure. The static pressure rise is -124240Pa with efficiency at part load conditions is 82.7702%
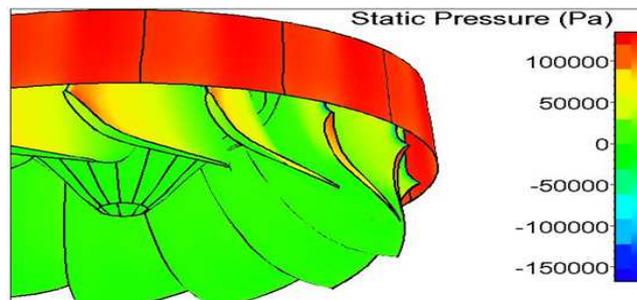
Figure 7: Static Pressure Distribution through the Runner

CONCLUSIONS

A CFD based software Numeca has been used for hydrodynamic performance of Francs runner blade. This report presents various steps .Numeca software does mesh generation of both static and rotating components of Francis turbine. The initial boundary conditions used in this report as mass flow, static temperature and Turbulence Viscosity. Static pressure defined as the outlet boundary condition. CFD approach may be helpful in improvement of the existing efficiency measuring techniques and evaluation of the performance of hydro turbines. A new runner with improved hydrodynamic performances can be carried out by CFD analysis and using Euranus solver module in conjunction with data available about operating conditions. Also A CFD based software Numeca has been used for hydrodynamic performance of Francs runner blade. This report presents various steps .Numeca software does mesh generation of both static and rotating components of Francis turbine.

REFERENCES

1. Richardson L F (1910) Proceeding of Royal Society of London, Set-A pp 210, 307-357.

2. Liepmman H (1918), "Mathematics and Physics", Klasae, Sitzungsber, Academic press 388

3. Howarth L (1938), Proceeding of Royal Society of London, part A pp 164,547.

4. Southwell R V (1940), "Relaxation Methods in Engineering Science", Oxford University Press.

5. Courant R (1943), Bulletin of the American Mathematical Society, vol 49, pp 1-23.

6. Lax P D (1954), "Computational and Pure Applied Mathematics", vol 7, pp 159-193.

7. Evans M E and Harlow F H (1957), Los Alamos Scientific Laboratory Report LA-2139.

8. Von Mises R (1958), Mathematical Theory of Compressible Fluid Flow, Academic Press.

9. Clough R W (1961), Proceedings of the Second ASCE Conference, Pittsburgh.

10. Fromm J E (1963), Los Alamos Scientific Laboratory Report, LA-2910.

11. Fromm J E and Harlow F H (1963), Journal of Physics and Fluids, vol 6, pp 975-982.

12. Van Dyke M (1964), Perturbation Methods In Fluid Mechanics, Academic Press.

13. Harlow F H and Welch (1965), Journal of Physics and Fluids, vol 8, pp 2182-2188.

14. Lng.L.Grimm (1969), Lectures Notes at the Technical University of Brno.

15. Harlow F H and Amsden A A (1970), Los Alamos Scientific Laboratory Report, LA-4370.

16. R a Novak & R M Hearsey (1977), "A nearly three dimensional intrablade computing system for turbo machinery", Transaction of ASME, pp 153-157.

17. MiroslavNechleba (1980), "Hydraulic Turbine", Their Design and Equipment, ARTIA Prague, Czechoslovakia.

18. Schlichting H (1987), "Boundary Layer Theory", McGraw Hill, New York.

19. R EisingerRupracht A (1990), "Automatic shape optimization of hydro turbine components based on CFD" Proceeding of 15th Symposium of Hydro Machinery Belgrade, Tugaslavia.

20. C A J Fletcher (1991), "Computational Techniques for Fluid Dynamics" Vol. 1, Springer Pub.

21. Ruprecht A, Eissinger R, Gode (2000), "Innovative design Environment for Hydroturbine components". HIS Report, Univ. of Stuttgart.

22. Andrej Lipej and Carlo Poloni (2000), "Design of Kaplan Runner using multi objective genetic algorithm optimization", Journal of Hydraulic research,Vol.38.

23. J Li and N Satofuka (2001), "Optimization, design of a compressor cascade aerofoil using a Navier- Strokes solver and genetic algorithm". Journal of Power and Energy IME, Vo. 216 Part A.

24. N M White, A Tourlidakis and R L Elder (2001), "Axial compressor performance modeling with a quasi 1-D approach", Journal of Power and Energy IME, Vol. 216 Part a pp 181-193.

25. Sttern F, Wilson R V, Coleman H W and Paterson E G (2001), "Comprehensive Approach to Vreification and Validation of CFD Simulations – Part 1: Methodology and Procedures", ASME Journal of Fluids Engg. 123, pp 793-802.

26. GlanlucaLaccarino (2001), "Prediction of a turbulent separated flow causing commercial codes". Journal of Fluids Engg. Vol. 123, pp 819-827, Dec 2001.

27. M J Cervantes and T S Engstrom (2004), "Factorial Design Applied to CFD", ASME Journal of Fluids Engg. Vol. 126, pp 791.

28. CFX 5.7 user manual, ANSYS Inc., 2004.

29. Csanady (1964)," Theory of Turbo machines", McGraw Hill, New York.

30. Anderson D. J Jr. (1995). "CFD basics with applications". McGraw Hill, New York

AUTHOR'S DETAILS

V. Ramudu working as a Assistant professor in Marri Laxman Reddy Institute of Technology and

Management, Hyderabad. Having 4+ years teaching experience and 4+Years in industrial. He completed Bachelor degree in Mechanical engineering, Presently pursuing M.Tech in CAD/CAM in Marri Laxman Reddy College of engineering and technology& management, Dundigal

U. Sudhakar working as a Assistant professor in Marri Laxman Reddy Institute of Technology and

Management, Hyderabad. Having 5 years teaching experience. He completed Master's degree in Computational Fluid Dynamics.

P. Sadanandam Working as Assistant professor in Mechanical Department, CVR College of engineering, Ibrahimpatnam, Telangana.