ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785
(ISRA), Journal Impact Factor: 2.114

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

GOVERNINIG EQUATIONS FOR COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF THE PERFORMANCE OF LOW HEAD BULB TURBINE

Jayendrakumar Ambalal Parmar*, Prof. Sohil Bux
M.Tech*, Professor, R.K.D.F University, Bhopal, M.P., India

ABSTRACT
Turbine is a rotary engine that converts energy from a fluid flow by transferring the potential energy to mechanical energy for electricity generation. In hydro turbines the runner blades to rotate depending on head ,water flow rate, variation of pressure and momentum .Here we intend to study the governing equation used in the computational fluid dynamics analysis of the low head bulb turbine. The governing equations play an important role in the performance analysis.

KEYWORDS: Low head buble turbine.

INTRODUCTION
India face severe power crisis due to the shortfall of fossil fuel supply .It depends on the nuclear fuel on the other countries which supply the same on their conditions .Our country has tremendous amount of hydraulic energy potential which is if converted in to power would largely satisfy our power demands. The bulb turbine which are normally employed for high head application can be modified for low head applications.

LOW HEAD HYDRO TURBINE [1]
The hydro turbine which uses the head of water from 2 to 35m is classified as low head hydro turbine. Its flow rate is approximately 0.3 to 100 m^3/s. Generally they are axial or radial type.
They are often installed on flat landscapes. They are running river plants needing dam or weir. They may use polluted water .They utilize low speed turbines often needing expensive generators. It also involves certain ecological aspect. e.g. fish friendliness or fish ladder or fish friendly turbine concept.

Type of turbine
There are several types of turbines .
1 Steam turbines are used for the generation of electricity in thermal power plants, such as using coal, fuel oil or nuclear energy. These turbines are used to generate electricity.

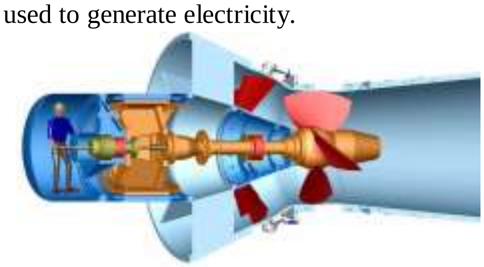

Fig. 1 .Bulb turbine cross section [1]

2 Gas turbines Such turbine has an inlet, fan, compressor, combustor and nozzle etc.
3 Contra-rotating turbines.
In an axial turbines an efficiency can be increased if the downstream turbine rotates in the opposite direction to an upstream unit. The design is essentially a multi-stage radial turbine (or 'nested' turbine rotors) giving more great efficiency as in the reaction (Parsons) turbine.

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785
(ISRA), Journal Impact Factor: 2.114

4 Stator less turbine. Multi-stage turbines have a set of static inlet guide vanes that direct the gas/fluid flow to the rotor blades. In a stator less turbine the fluid flow coming from an upstream rotor impinges onto a downstream rotor without an intermediate set of stator vanes .
5 Ceramic turbine. Conventional high pressure turbine blades (and vanes) are made from nickel based alloys and often utilize intricate internal air-cooling passages to prevent the metal from overheating. In recent years, experimental ceramic blades have been manufactured and tested in gas turbines, with an intension of increasing the rotor Inlet Temperatures and eliminating an air cooling.
6 Shrouded turbines. Many turbine rotor blades have shrouding at the top, which interlocks with that of adjacent blades, to increase damping and thereby reduce blade flutter.
7 Shroud less turbine. These turbines do not have shroud thereby reducing the centrifugal load on the blade and the Cooling requirements.
8 Wind turbines.
These normally operate as a single stage without nozzle and inter stage guide vanes.

Classification of Hydraulic Turbines:
Based on pressure change
1. Impulse Turbine: The pressure of liquid does not change while passing through the rotor . It changes in the nozzles . e.g Pelton Wheel.
2. Reaction Turbine: The pressure changes while passing through the rotor. The change in fluid velocity and reduction in pressure causes a reaction on the blades. Hence they are named as Reaction Turbine. E.g. Francis and Kaplan Turbines
Based on flow path
1. Axial Flow Hydraulic Turbines: Turbine has the flow path parallel to the axis of rotation. Kaplan Turbines has liquid flow in axial direction.
2. Radial Flow Hydraulic Turbines: Such Hydraulic Turbine has the liquid flowing mainly in a plane perpendicular to the axis of rotation.
3. Mixed Flow Hydraulic Turbines: Hydraulic Turbines uses both axial and radial flows. Hence these hydraulic turbines are called as Mixed Flow Turbines. Francis Turbine is an example in which water enters in radial direction and exits in axial direction.
Hydraulic Turbines are neither purely axial flow nor purely radial flow.
3. Mixed Flow Hydraulic Turbines: Hydraulic Turbines use both axial and radial flows. Francis Turbine is an example of mixed flow type.
Based on head at the inlet of turbine:
1. High head turbine (100m<H< above)
2. Medium head turbine (30m<H<100m)
3. Low head turbine (2m<H<30m)
Based on specific speed of the turbine:
1. Low specific speed turbine
2. Medium specific speed turbine
3. High specific speed turbine

INTRODUCTION TO CFD

Computational Fluid Dynamics (CFD) is a useful tool to optimize the design of hydro turbines and improve in their efficiencies; CFD is used to analyze the fluid flow helping the design part to be cost and time saving. The CFD simulation of fluid flow pass hydro turbine is to analyze the effect of blade angle, runner blades and inlet guide vanes on pressure and velocity distributions of hydro bulb turbine. And the results would be useful as the guideline for the design of hydro bulb turbine.

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785
(ISRA), Journal Impact Factor: 2.114

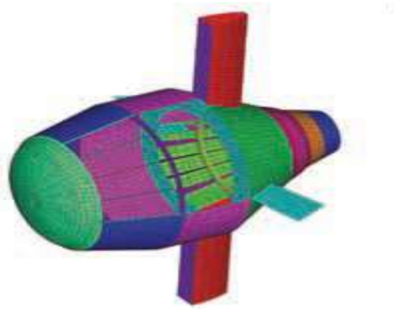

Fig.2 Finite Element Dynamic Analysis

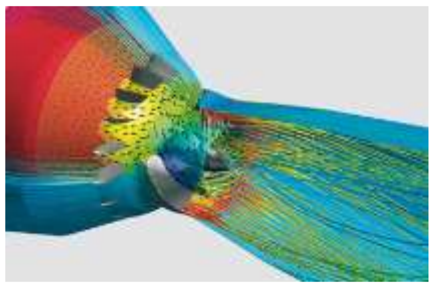

Fig.3 CFD: pressure and velocity distribution of bulb turbine

FINITE ELEMENT ANALYSYS

Introduction to FEM

It is the tool for the numerical solution of various engineering problems using computer. In this method the complex problem is divided into simple problems which can be easily solved by present mathematical tools. The engineering designs are very often complex involving tedious calculations so to make it easy the engineer need FEA .Meshing is the process by which we divide the complex shape into small pieces. The common points of the elements are termed as the nodes. It is a computational technique used to obtain approximate solutions of boundary value problems in engineering. The boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation within a known domain of independent variables and satisfying specific conditions on the boundary of the domain. Boundary value problems are often called field problems. The field is the domain of interest and represent a physical structure. The field variables are the dependent variables of interest governed by the differential equation. The boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few.

GOVERNING EQUATIONS

In fluid dynamics the governing equations are classified into three categories :

(1)elliptical (2)parabolic and (3)hyperbolic.

These equations represents the flow velocity The elliptical equation represent the subsonic flow, the parabolic represent the transonic where as the hyperbolic equation represent the supersonic flow. These equations are second order partial differential equations. The general form of the equation in 2D is as follows:

$$A\frac{\partial^2 u}{\partial x^2} + B\frac{\partial^2 u}{\partial x \partial y} + C\frac{\partial^2 u}{\partial y^2} + D\frac{\partial u}{\partial x} + E\frac{\partial u \frac{\partial u}{\partial x}}{\partial y} + Fu + G = 0 \quad (1)$$

ecThey may depend on the variables. When we check the continuity of the first order derivative of u ,$u_x = \frac{\partial u}{\partial x}$ and $u_y = \frac{\partial u}{\partial y}$.

$$du_x = \frac{\partial u_x}{\partial x}\, dx + \frac{\partial u_x}{\partial y} = \frac{\partial^2 u}{\partial x^2}\, dx + \frac{\partial^2 u}{\partial x \partial y} dy \qquad (2)$$

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785
(ISRA), Journal Impact Factor: 2.114

And $du_y = \frac{\partial u_y}{\partial x} dx + \frac{\partial u_x}{\partial y} du = \frac{\partial^2 u}{\partial x \partial y} dx + \frac{\partial^2 u}{\partial x^2} dy$ (3)
u form the solution space above or below x-y plane and slope dy/dx representing the solution surface space i defined as the characteristic curve.

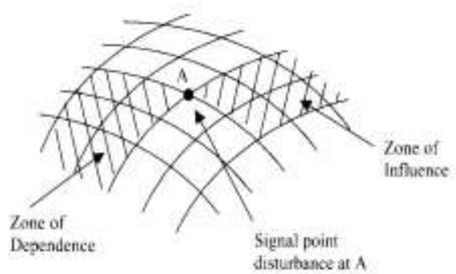

Fig4 Propagation of Disturbance and characteristic[1]

The equation 1,2 and 3 can be combined into matrix form as follows :

$$\begin{bmatrix} A & B & C \\ dx & dy & 0 \\ 0 & dx & dy \end{bmatrix} \begin{bmatrix} u_{xx} \\ u_{xy} \\ u_{yy} \end{bmatrix} = \begin{bmatrix} H \\ du_x \\ du_y \end{bmatrix} \quad (4)$$

Where $H = -\left(D\frac{\partial u}{\partial x} + E\frac{\partial u}{\partial y} + Fu + G\right)$ (5)

These derivatives are indeterminate due to the possible discontinuity in the second order derivative of the dependent variable along the characteristics. This is seen when the coefficient matrix determinant is zero.
Fig.2 Propagation of disturbance and charecteristics[3]

$$\begin{bmatrix} A & B & C \\ dx & dy & 0 \\ 0 & dx & dy \end{bmatrix} = 0 \quad (6)$$

This gives

$$\left(\frac{dy}{dx}\right)^2 - B\left(\frac{dy}{dx}\right) + C = 0 \quad (7)$$

On solving above equation we get

$$\frac{dy}{dx} = \frac{-B \pm \sqrt{B^2 - 4AC}}{2A} \quad (8)$$

The general equation of a conic section can be given by the following equation:

$$AX^2 + BXY + CY^2 + DX + EY + F = 0 \quad (9)$$

Thi s equation represents the curve which may be real or imaginary depending on the value of $B^2 - 4AC$.
$B^2 - 4AC < 0$ represent the Elliptical equation.
$B^2 - 4AC = 0$ represent the parabolic equation and
$B^2 - 4AC > 0$ represent the Hyperbolic equation.
In CFD the PDE's play an important role as they decide the computation scheme and specification of boundary conditions. They may be of mixed type in many situations. Hence the selection of computational schemes and boundary conditions application method are important in CFD.

NAVIER –STOKES SYSTEM EQUATIONS

The non conservative form of governing equation can be derived from first law of thermodynamics.

$$\frac{DK}{Dt} + \frac{DU}{Dt} = M + Q \quad (10)$$

Where K,U,M and Q represent the kinetic energy, internal energy, mechanical power and heat energy respectively.

$K = \int_\Omega \frac{1}{2}\rho_i v_i v_i \, d\Omega$ (11) Where Ω = flow field, ρ = density per unit mass, v_i = velocity vector component

$U = \int_\Omega \rho \epsilon \, d\Omega$ (12) where ϵ = internal energy per unit mass

$M = \int_\Omega \rho F_i v_i \, d\Omega + \int_\Omega \rho_{ij} v_j \, d\Gamma$ (13) where F_i = body force tensor component

 ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785
(ISRA), Journal Impact Factor: 2.114

$Q = \int \rho r\, d\Omega - \int_\Gamma q_i n_i\, d\Gamma$ (14) T= temperature, k = coefficient of thermal conductivity, r = heat supply per unit mass
Where get
$\epsilon = c_p T - \frac{p}{\rho}$(15)μ = coefficient of dynamic viscosity
$\sigma_{ij} = -p_{ij} + \tau_{ij}(16)\tau_{ij}$ = viscous stress tensor,σ_{ij} = total stress tensor
$\tau_{ij} = \mu(v_{i,j} + v_{j,i}) - \frac{2\mu}{3} v_{k,k}\delta_{ij}$ (17)
$q_i = -kT_i$(18)
Whereδ_{ij} = kronecker delt, $\delta_{ij} = 0$ when I and j are not equal and $\delta_{ij} = 1$ when i and j are equal.
On substituting eqn (11) ,(12), (13) and (14) into eqn (10)and using Green- Gauss theorem we get the following equations.
The continuity equation is
$\frac{\partial\rho}{\partial t} + (\rho v_i)i = 0$ (19)
The momentum equation is
$\rho\frac{\partial v_j}{\partial t} + \rho v_i, jv_i + P_i$-τ_ij,i-$\rho F_j$= 0 (20)
The energy equation
$\rho\frac{\partial\epsilon}{\partial t} + \rho\, \epsilon_i v_i + pv_{i,j} - \rho\, \tau_{i,j} vi, j + q_{i,j} - \rho r = 0$ (21)
These equations (19) (20) and (21) are the NAVIER – STOKES equations for the compressible viscous flow in the non conservation form.
The above equations may be written in the conservation form as follows
$\frac{\partial u}{\partial t} + \frac{\partial F_j}{\partial x_j} + \frac{\partial F_i}{\partial x_i} = B$ (22)
U = conservation flow variable, F_i = convection flux variable , G_i = diffusion flux variable and B = source terms.
$$U=\begin{bmatrix}\rho\\ \rho v_j\\ \rho E\end{bmatrix},\ F = \begin{bmatrix}\rho v_i\\ \rho v_i v_j + p\delta_{ij}\\ \rho E v_i + p\delta_{ij}\end{bmatrix}\ G_i = \begin{bmatrix}0\\ -\tau_{ij}\\ -\tau_{ij} v_{j+q_j}\end{bmatrix}\quad B = \begin{bmatrix}0\\ \rho F_j\\ \rho F_{jVj}\end{bmatrix} \quad (23)$$
E=total stagnation energy
= ϵ+1/2 $v_i v_j$ (24)
This is related to pressure and temperature by the following relations
$p = (\gamma - 1)\rho\left(E - \frac{1}{2}v_i v_j\right)$ (25)
$T = \frac{1}{c_v}\left(E - \frac{1}{2}v_i v_j\right)$ (26)
Where $c_v$is the specific heat at constant volume.
The NAVIER – STOKES equations can be simplified to the Euler equation by neglecting the diffusion flux variables G_i .We can obtain the equation (22)which is the NAVIER – STOKES equations conservation form of the by differentiating the equation (21).Where as integrating equation 22 spatially over the volume of the domain gives
$\int_\Omega \left(\frac{\partial U}{\partial t} + \frac{\partial F_i}{\partial x_i} + \frac{\partial G_i}{\partial x_i} - B\right) d\Omega = 0$ (27)
We obtain another form of the governing equations
$\int_\Omega \left(\frac{\partial U}{\partial t} - B\right) d\Omega + \int_\Gamma (F_i + G_i)\, n_i\, d\Gamma$ =0 (28)
The surface integral in the above equation represent the convection and diffusion fluxes through the control surfaces . The surface integral in the above equation serves dual purpose firstly it lay the foundation for the finite volume method and secondly it gives proper solution for the high gradient flows like the shock wave.
The First Law of Thermodynamics equations can be conveniently used for solving the primitive variables like ρ,p,T and v_i where as conservation form of NAVIER – STOKES equations or the Constant Volume surface equations can be preferred for obtaining the discontinuities such as in shock wave
The NAVIER–STOKES equations in the conservation form and non conservation form give rise to various types of fluid flow. The computational schemes are dependent on the flow characterized by the situation.
Navier-Stokes system of equations become the Euler equations when all viscous terms are removed from it. The momentum equations without the pressure gradients are called the Burgers' equation. The Burgers' equation can be classified into inviscid linear, inviscid nonlinear, linear viscous, and nonlinear viscous.

The Navier–Stokes equations are nonlinear partial differential equations in the general case and so remain in almost every real situation. In some cases, such as one-dimensional flow and Stokes flow (or creeping flow), the equations can be simplified to linear equations. The nonlinearity makes most problems difficult or impossible to solve and is the main contributor to the turbulence that the equations model.

The nonlinearity is due to convective acceleration, which is an acceleration associated with the change in velocity over position. Hence, any convective flow, whether turbulent or not, will involve nonlinearity. An example of convective but laminar (non turbulent) flow would be the passage of a viscous fluid (for example, oil) through a small converging nozzle. Such flows, whether exactly solvable or not, can often be thoroughly studied and understood.[17]

PROPERTIES OF NAVIER–STOKES EQUATION

Nonlinearity

The N–S equations are nonlinear PDE in the general case., The equations can be simplified to linear equations in N–S equation1D flow and Stokes flow (or creeping flow).Most problems difficult to be solved. The nonlinearity is due to convective acceleration, which is an acceleration associated with the change in velocity over position. Hence, any convective flow, whether turbulent or not, will involve nonlinearity. An example of convective but laminar (non turbulent) flow would be the passage of a viscous fluid e.g. oil through flow a small converging nozzle.

Turbulence

Turbulence is the time-dependen behavior seen in a fluid flows. It is generally believed that it is due to the inertia of the fluid . It is believed, that the Navier–Stokes equations describe turbulence properly.

For turbulent flow is extremely difficult to have the numerical solution of the N–S equations due to the significantly different mixing-length scales that are involved in turbulent flow. However the stable solution requires such a fine mesh resolution that the computational time becomes significantly large for calculation or direct numerical simulation. Attempts to solve turbulent flow using a laminar solver typically result in a time-unsteady solution, which fails to converge appropriately. Reynolds-averaged N–S equations (RANS) supplemented with turbulence models, are used in the practical computational fluid dynamics (CFD) applications. Some of the models are k-ω , k-ε , and SST models. Large eddy simulation (LES) can also be used to solve these equations. It is more expensive than RANS, but produces better results as it explicitly resolves the larger turbulent scales.

Applicability

Along with supplemental equations and well formulated boundary conditions, the N–S equations model fluid motion accurately .Even turbulent flows seem on average to agree with real world observations.

The N–S equations assume that the fluid is a continuum and is moving at relativistic velocities. At very small scales or under extreme conditions, real fluids made out of discrete molecules will produce results different from the continuous fluids modeled by the N–S equations. It has complicated nature of the equations. The application of the N–S equations to less common families tends to result in very complicated formulations and often to open research problems. The existence and smoothness problem concerns the mathematical properties of solutions to the N–S equations, one of the pillars of fluid mechanics (such as with turbulence). These equations describe the motion of a fluid (that is, a liquid or a gas) in space. The N–S equations are used in many practical applications. In particular solutions of the N–S equations often include turbulence, which remains one of the greatest unsolved problems in physics. The solutions to Navier–Stokes have never been proven for the 3D system of equations, and mathematicians have not yet proved that smooth solutions exist, or that if they do exist, they have bounded energy per unit mass. This is called the existence and smoothness problem of N-S equation.

CONCLUSION

The governing equations play very important role in the CFD analysis of the performance of the low head bulb turbine. Though N- S equation are having a smoothness problem and the particular solutions but it can be efficiently used for the large approximation of the results.

REFERENCES

[1]ADRITZ TECH HYDRO LOW HEAD HYDRO TURBINE
[2] COMPUTATIONAL FLUID DYNAMICS THE BASICS WITH APPLICATION BY JOHN D ANDERSON Jr.
[3] COMPUTATIONAL FLUID DYNAMICS BY T.J.CHUNG
[4] BULB UNITS BY ALSTOM
[5] Prof.Dr. SUMAN CHAKRAVARTY ,I.I.T. KHARAGPUR

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785
(ISRA), Journal Impact Factor: 2.114

[6] NAVIER-STOKE EQUATIONS THEORY AND NUMERICAL ANALYSIS, ROGER TEMAM ,UNIVERSITY DE PARIS,FR