FLOW ANALYSIS OF A SHP STATION

A DISSERTATION

Submitted in partial fulfillment of the requirements for the award of the degree

of

MASTER OF TECHNOLOGY

in

ALTERNATE HYDRO ENERGY SYSTEMS

By

SRIDHAR REDDY KUNAM

ALTERNATE HYDRO ENERGY CENTRE
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE - 247 667 (INDIA)

JUNE, 2006

ACKNOWLEDGEMENT

I would like to express my sense of gratitude to my guides **Dr. R.P.Saini,** Senior Scientific Officer, Alternate Hydro Energy Centre and **Shri Arun Kumar,** Head, Alternate Hydro Energy Centre, Indian Institute of Technology Roorkee, for guiding me to undertake this dissertation as well as providing me all the necessary guidance and support throughout this dissertation work. I can never forget their caring words and support in the difficult times. They have displayed unique tolerance and understanding at every step of progress, without which this dissertation work would not have been in the present shape. I feel it is my privilege to have carried out the dissertation work under their valuable guidance.

I owe a great deal of appreciation to Faculty of Alternate Hydro Energy Centre for imparting knowledge during my M.Tech course.

I would also like to thank all my friends, for their help and encouragement at the hour of need. Finally I would like to mention my parents who are with me all the times to assist me with their caring and affection right from my childhood.

Date: **June 2006** **(Sridhar Reddy Kunam)**

ABSTRACT

Quest for energy is unending due to the rapid depletion of the fossils fuels we need to look for the alternate sources of energy, which is also clean and renewable. Among the renewable energy sources available, small hydropower is considered as most the promising and economical source of renewable energy. India is blessed with many rivers, natural streams, irrigation cannel with barrages, dams network and mountains offering hydro potential of different size like large, small, mini and micro hydropower.

The cost effective and efficient design for the different machines SHP Station is required. This hydropower industry is faced today with challenge of addressing the cost effective innovative concepts of Hydro turbines and station layout. General and economic approach to employ three dimensional computational fluid dynamics (CFD) models to simulate the flow through the various components of hydropower station including the fore bay and tailrace channel is required. The application of CFD has become an essential part of the design, model study, and testing for such development.

Under present study of flow behavior across the water conductor system of a SHP station is considered for analysis. The commercial available *Fluent* software has been used for the analysis. The flow field is calculated numerically by means of color visualizations, Quantitative analysis of the flow field for better understanding of the fluid mechanics was obtained and discussed.

CONTENTS

CHAPTER – 4: ANALYSIS OF FLOW IN SHP POWER CHANNEL BY USING *FLUENT* SOFTWARE

CHAPTER –5: CONCULSIONS AND RECOMMENDATIONS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

Symbol	Description	Unit
V	Velocity	m/s
C	Chezy's resistance factor	-
R_h	Hydraulic radius	m
S_c	Channel bottom slope	-
n	Manning Roughness coefficient	-
S_o	Settling Velocity	m/s
y_1	Conjugate Depth before Jump	m
y_2	Conjugate Depth after Jump	m
F_1	Froude's number	-
h_c	Head loss due to contraction	m
V_1	Velocity at inlet	m
V_2	Velocity at outlet	m
g	Acceleration due to gravity	m^2/s
h_{ex}	Head loss due to Expansion	m
A_1	Area of cross section at inlet	m^2
A_2	Area of cross section at outlet	m^2
d	Diameter before contraction	m
D	Diameter after contraction	m
K_c	Coefficient of contraction joint	-
K_e	Coefficient of the expansion joint	-

α	Diffuser angle	Degrees
h_t	Head loss due to trash rack	m
K_t	Coefficient of trash rack	-
t	Bar thickness	mm
b	Width between bars	mm
V_o	Velocity of approach	m/s
ϕ	Angle of inclination from horizontal	Degrees
K_v	Coefficient of the valve	-
P	Pressure	N/m^2
τij	Shear stress	N/m^2
ρ	Density	kg/m3

INTRODUCTION AND LITERATURE REVIEW

1.1 INTRODUCTION

Energy is critical in developing countries not only for economic growth but also for social development and human welfare. The energy use in the developing world has doubled in the last two decades and further expected to double again in an even shorter time span of the next fifteen years. So in order to meet the desired increase in demand all the sources of the energy are required to exploit. Due to the rapid depletion of the fossils fuels we need to look for the renewable energy sources.

Indian per capita consumption of electricity continues to be around 350 kWh per annum. About 80,000 villages remain yet to be electrified in spite of the highest priority given to rural electrification in India [1]. Most of these villages are located in remote areas, with very low load densities. In remote areas where transmission of grid power is totally uneconomical off grid electrification can be undertaken through renewable energy systems such as Small Hydro schemes besides solar photovoltaic, which is not available throughout the day.

India is blessed with many rivers, natural streams, cannel networks and mountains offering tremendous hydro potential of major, small, mini and micro hydropower. Hydroelectric power was initiated in India in 1897 with a run-of-river unit was located near Darjeeling [2]. The worldwide contribution of small hydropower has grown substantially in the last ten years. Among all the renewable energy sources available, small hydropower is considered as most the promising source of the energy.

1.2 SMALL HYDRO POWER

There is a general tendency in world all over the world to define small hydropower is by the power output. Different countries follow different norms, the upper limit ranges between 5 to 50 MW, as given in the following Table 1.1

TABLE 1.1 Worldwide Definitions for Small Hydropower [2]

UK (NFFO)	<= 5MW
UNIDO	<=10MW
INDIA	**<=25MW**
SWEDEN	<=15MW
COLOMBIA	<=20MW
AUSTRALIA	<=20MW
CHINA	<=25MW
PHILIPPINES	<=50MW
NEW ZEALAND	<=50MW

In India Small hydropower the Central Electricity Authority (CEA) classifies schemes as follows in the table 1.2 [2]

Table 1.2 Various Capacity of Small hydropower [2]

Type	Station capacity	Unit rating
Micro	Up to 100 kW	Up to 100 kW
Mini	101 kW to 2000 kW	101 kW to 1000 kW
Small	2001 kW to 25000 kW	1001 kW to 5000 kW

1.3 DIFFERENT TYPES OF SHP SCHEMES

Small hydropower can also be broadly categorized in three types as follows

1.3.1 Run-of-River Schemes

Run-of-River hydroelectric schemes are those, in which water is diverted from stream without creating any storage in the river. In such schemes, power is generated from flowing water and available head. The output of a run-of-river plant is subjected to the instantaneous flow of the stream.

1.3.2 Canal Based Schemes

Canal based small hydropower scheme is planned to generate power by utilizing the fall in the canal. These schemes may be planned in the canal itself or in the bye pass channel.

1.3.3 Dam Toe Based Schemes

Dam Toe Based schemes are those in which water is stored in a large reservoir. Level of the water in the dam decides the head available for power generation.

The Hydropower industry is faced today with challenge of addressing, in a cost effective manner of development, evaluation and implementation of innovative concepts for hydro turbines and design of the powerhouse components. A more general and economic approach is to employ three dimensional computational fluid dynamics (CFD) models to simulate the flow through the various components of hydropower installations [6].

1.4 COMPUTATIONAL FLUID DYNAMICS MODELLING

1.4.1 Scope of CFD

Computational Fluid Dynamics is a branch of Fluid Mechanics that resolves fluid flow problems numerically. The physical laws governing a fluid flow problem are represented by a system of partial differential equations including the continuity equation, the Navier-Stokes equations [3]. The numerical analysis resolves the equations by accurate and complex numerical schemes.

It nicely and synergistically complements the other two approaches of pure theory and pure experiment, but it will never replace either of these approaches. There will always be a need for theory and experiment. The future advancement of fluid dynamics will rest upon a proper balance of all three approaches, with computational fluid dynamics helping to interpret and understand the results of theory and experiment, and vice versa [8]. The Fig.1.1 shows the three dimensions in fluid dynamics.

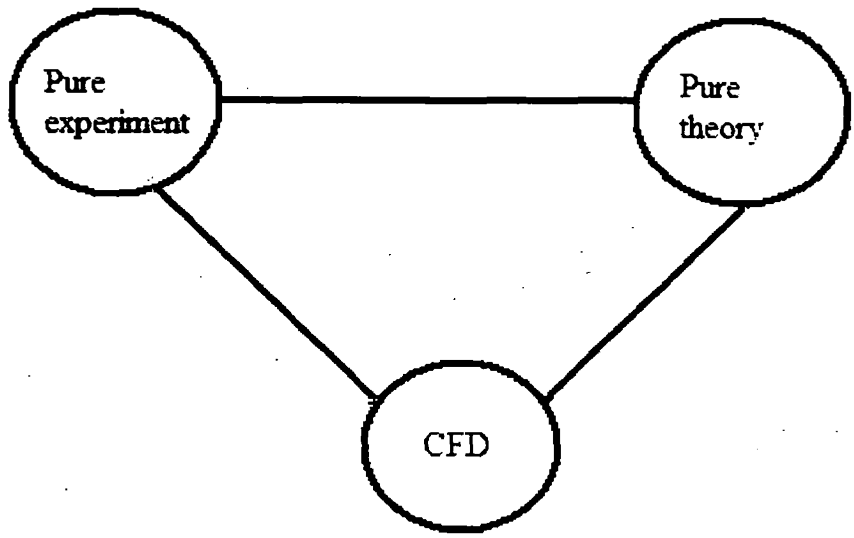

Fig.1.1 Three dimensions of fluid mechanics [8]

As the performance-to-cost ratio of computers has increased at a spectacular rate in the last decade and shows no sign of slowing down, CFD is considered more often as a key industrial tool. The main attraction in using Computational Fluid Dynamics, resides in its ability to overcome the difficulties encountered within physical models in site field experiments to measure flow quantities which are inaccessible flow regions and to avoid disturbances associated with intrusive instrumentation by the experimental methods.

In CFD flow field is calculated numerically by means of color visualizations, quantitative analysis a view of the flow field and better understanding of the fluid mechanics can be obtained. This technique also enables a detailed comparison to experiments to attain an even better understanding of the flow field. The method has become an essential part of design, testing and optimizing process.

1.4.2 The Benefits of CFD

i.Prediction

CFD may be applied to predict how a design will perform and test many variations to reach an optimal result. Prior to the use of CFD modeling physical prototyping and testing is performed in order to predict the performance of the system.

ii. Design Insight

Parts of the system can be visualized by CFD analysis which are some times not possible by any other mean to know what phenomena is happening within the system.

iii. Efficiency

Time and money are saved with shorter design cycles due to better and faster analysis. Products get to market faster. Provide insights into flow conditions by extending physical measurements and modeling, thus saving time and money.

1.5 LITERATURE REVIEW

Rajesh Bhaskaran et al, [4] illustrated the basic concepts of CFD by applying them to a simple 1D example and application of the CFD techniques to the most of engineering problems. The discretization using the finite difference method, finite volume method, assembly of discrete system and application of boundary conditions and the solution of the discrete has been explained. Dealing with the non-linearity occurred in the simulation.

I.Gunnar J Hellstrom [5] has applied the CFD techniques to redesign of an existing hydropower Draft tube discussed the scope and application of the standard k-ε and Statistical turbulence model (SST) to study the flow behavior of the turbulence modeling. The flow field and global engineering quantities, especially the pressure recovery factor, in an existing hydro power draft tube are effected by both shape of the draft tube and the CFD flow assumption, particularly the turbulence modeling

Suhas V. Patankar [6] presented various numerical methods to develop numerical equation for the fluid flow and heat transfer. He also discussed various methods of discretization equations and presented the algorithm to solve pressure correction equations like SIMPLER which is revised one SIMPLE from which force acting at each grid point can be derived.

John D.Anderson Jr [7] explained the basic equations required to apply CFD for various problem incurred in the common practice. Like flow behavior in the turbo machines, flow over the aeroplane wings and heat transfer. He discussed the step by step procedure on developing the mathematical equations and mentioned different methods used for solving the mathematical equations applied in CFD like SIMPLE and SIMPLER.

Subhash C. Jain [8] mentioned the flow in open channel by using various numerical equations, which are essential for the planning and the design of systems to manage water resources. Flow behavior over the spillways, hydraulic jump which usually occur after the roll waves phenomenon which was generally occur in the open channel flow and the surges created in the open channel due to operational conditions were explained.

Layman's Guide Book [9] on "how to develop a small hydro site", states that hydraulic engineering is based on the principles of fluid mechanics. Based on the large amount of accumulated experience there exists many empirical relationships to achieve practical engineering solutions with the movement of the water. The failures like seepage under the weir, open channel slides occurred through a lack of proper geological studies of the site. Turbines transform the potential energy of water to mechanical rotational energy, which in turn is transformed into electrical energy in the generators.

Albert Ruprecht [10] studied unsteady simulations presented applications with self-excited unsteadiness, vortex shedding or vortex rope in the draft tube, as well as applications with externally forced unsteadiness by changing or moving geometries, rotor-stator interactions. The requirements, potential and limitations of unsteady flows analysis assessed. Particularly the demands on the turbulence models and the necessary computational efforts are discussed.

K.L.Kumar [11] explained different types of the Hydraulic jumps occur in an open channel flow. The flow trough pipes discussed along with energy gradient line. The purpose of the draft tube and different used in common practice has been illustrated. He also discussed the regimes of the flow in the open channel.

Filip Sadlo et al, [12] presented the Vorticity Based Flow Analysis and Visualization for Pelton Turbine Design Optimization. They described the creation, transformation and extinction of vortices. For a better understanding of a flow it is therefore of interest to examine vorticity in all of its different roles. The Jet quality is affected mostly by vortices originating in the distributor ring. And also mentioned understanding of this interrelation, it is crucial to not only visualize these vortices but also to analyze the mechanisms of their creation.

Balint D et al, [13] mentioned the numerical simulation of three-dimensional flows in hydraulic turbines is established as one of the main tools in design, analysis and optimization of turbo machines. A methodology for computing the 3D inviscid flow in

Kaplan turbines has been presented. Both velocity and pressure field results are presented, with a particular focus on the runner blades pressure distribution.

Helena Ramos et al, [14] have discussed different components in SHP Station and flow behavior across them. The hydraulic design of the Small hydro plants with various configurations has been discussed. The various hydraulic transients and dynamic effects, which occur in open channel, penstock and surge tanks, have been illustrated.

E.A.Meselhe et al, [15] presented a numerical model to calculate three dimensional turbulent flow in open channels and an arbitrary cross section has developed and validated. The model solve the compressible, Reynolds averaged Navier stokes (RANS) equations, formulated in generalized curvilinear coordinates, in conjunction with the k-ε turbulence closure. The numerical model was presented by applying it to simulate the flow through a meandering open channel of trapezoidal cross section, for which experimental measurements are available.

Literature Survey reveals that now a days CFD plays an important role in cost effective design of the system in every area. However, a very limited study on SHP plants has been reported. Keeping this in view, the present study is focused to work the application of Computation Fluid Dynamic techniques to study the flow behavior for an SHP Power channel has been done.

Commercial available Fluent 6.2 version of CFD has been used to study the flow in power channel. The Velocity profile has been discussed along the Power channel in order to investigate the flow conditions, which could be used during design of similar structures.

The flow field is also calculated numerically and by means of color visualizations and quantitative analysis a view of the flow field has been observed for better understanding of the fluid mechanics.

DIFFERENT COMPONENTS OF SHP PLANT

The typical layout of the Small hydropower plant is as shown in Fig. 2.1. It consists of the Weir to divert the flow from the main stream, Intake to water from the main stream to forebay tank. Penstock conducts water from the forebay tank to Powerhouse and Tailrace to conduct water from powerhouse to the stream back [22].

2.1 POWER CHANNEL

The channel conducts the water from the intake to the Forebay tank. The length of a channel can be a few kilometers to create a required head. The length of the channel depends on local conditions. The combination of penstock and channel will be used depending on the conditions [22].

2.1.1 Water Flow in Open Channels

Flow in the open channels always associated with free surface. The free water surface is subject to the atmospheric pressure, referred as the zero pressure reference and considered as constant along the full length of the canal. The depth of water changes with the flow conditions [9].

The canal even a straight one, has a three-dimensional distribution of velocities. Fig2.2 illustrates the Iso-velocity lines in channels of Triangular, Trapezoidal, Shallow and Natural profile.

2.1.2 Steady and Unsteady Flow

According to the time criterion, channel flow is considered steady when the discharge and the water depth at any section of the stretch do not change with time. Unsteady when one or both of them changes with time.

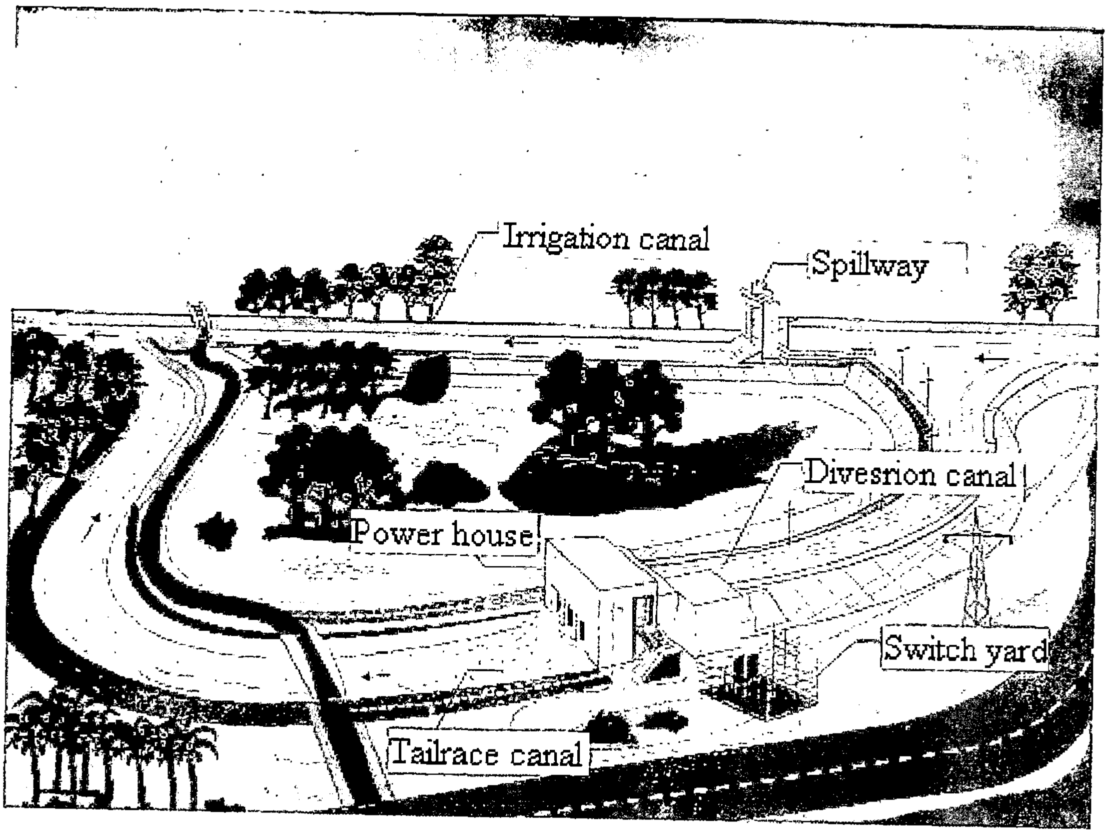

Fig.2.1 Typical layout of the canal based small hydro power plant [22]

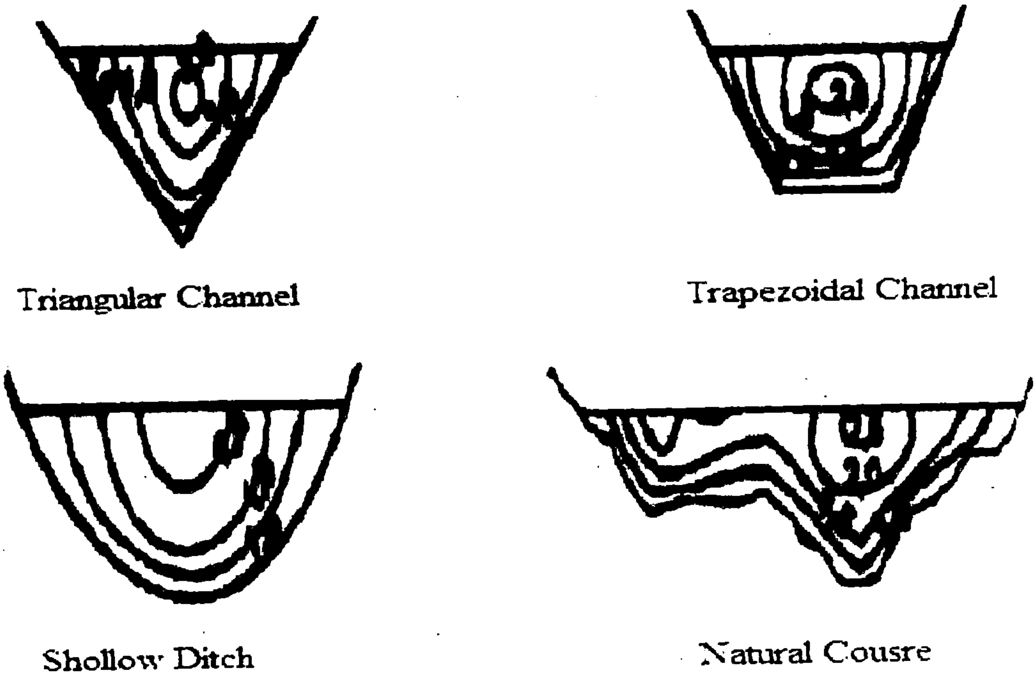

Fig.2.2 Iso-velocity lines in channels of different profile [9]

Based on the space criterion, an open channel flow is said to be uniform if the discharge and the water depth at any section of the stretch do not change with time, and is said to be varied when the discharge and the water depth change along its length.

2.1.3 Governing Equations

The following are the various governing equations, which are used to study flow in an open channel flow. Based on these concepts Chezy found that the velocity in the open channel is given by Eq. (2.1),

$$v = c\sqrt{R_h S_e} \tag{2.1}$$

Where C = Chezy's resistance factor

R_h = Hydraulic radius of the channel cross-section

S_e = Channel bottom line slope

Manning derived the following empirical relation to calculate the 'C'

$$C = \frac{1}{n} R_h^{\frac{1}{6}} \tag{2.2}$$

Where 'n' is the well-known Manning's roughness coefficient.

Substituting C from (2.1) into (2.2) we has the Manning formula for uniform flows

$$V = \frac{1}{n} R_h^{\frac{2}{3}} S_e^{\frac{1}{2}} \tag{2.3}$$

$$Q = \frac{1}{n} A R_h^{\frac{2}{3}} S_e^{\frac{1}{2}} \tag{2.4}$$

Where, $AR_h^{2/3}$ has been defined as the section factor

The formula is entirely empirical and the 'n' coefficient is not dimensionless, so the formulae given here are only valid in S.I. units and formulae are applicable to channels with a flat bottom.

From Eq. (2.4) it may be deduced that for a channel with a certain cross-section area A and a given slope S, the discharge increases by increasing the hydraulic radius. That means the hydraulic radius is an efficiency index. As the hydraulic radius is the quotient of the area 'A' and the wetted perimeter 'P', the most efficient section will be the one with the minimum wetted perimeter.

Among all cross-sectional areas, the semicircle is the one, which has the minimum wetted perimeter for a given area. A channel with a semicircular cross section is expensive to build and difficult to maintain, used only in small section channels built with prefabricated elements. The next most efficient shape of the open channel cross section is trapezoidal section which cross section of a half hexagon.

Most channels are excavated, in practice the structures like aqueducts are necessary prevailing the conditions. To reduce friction and prevent leakages channels are often sealed with cement, clay or polythene sheet.

2.2 WEIR AND INTAKE

The water flowing in the channel must be regulated during high river flow and low flow conditions. A weir can be used to raise the water level and ensure a constant supply to the intake. Sometimes it is possible to avoid building a weir by using natural features of the river [22].

The principle of the temporary weir is to construct a simple structure at low cost using local labour, skills and materials. The intake of a hydro scheme is designed to divert a certain part of the river flow. This part can go up to 100 % as the total flow of the river is diverted via the hydro installation.

The following points are required to be considered while designing an intake

i). The desired flow must be diverted.

ii). The peak flow of the river must be able to pass the intake and weir.

iii). As less as possible maintenance and repairs.

iv). It must prevent large quantities of loose material from entering the channel.

v). It must have the possibility to remove piled up sediment.

Different types of intakes are characterized by the method used to divert the water into the intake. These are discussed as follows;

2.2.1 Side Intake with Weir

The following are the features of the Side Intake with Weir used in SHP structures,

i). Control water level

ii). Little maintenance is required

iii). Low flow cannot be diverted properly

iv). Modern materials like concrete necessary

2.2.2 Side Intake without Weir

The following are the features of the Side Intake without Weir used in SHP structures,

i). Relatively cheap

ii). No complex machinery required for construction

iii). Asks for regular maintenance and repairs

iv). At low flows very little water will be diverted and therefore this type of intake is not suitable for rivers with great fluctuations in flow

2.2.3 Bottom Intake

The following are the features of the Bottom Intake Weir used in SHP structures,

i). Very useful at fluctuating flows. Even the lowest flow can be diverted

ii). No maintenance required

iii). Expensive

iv). Local materials not useable

v). Good design required to prevent blockage by sediment

At a bottom intake the whole weir is submerged into the water. Excess water will pass the intake by flowing over the weir. This type is weir is suitable in the region where boulder movement is high. They are many unconventional approaches used in the weir like Conda weir, Rubber weir depending upon on the suitable conditions.

The floating debris can be removed by using a steel or wooden bar ('skimmer'), can be positioned on the water surface at an angle to the flow as to stop the debris and protect the intake.

2.3 SETTLING BASIN

The water drawn from the river is feed to the turbine, which will usually carry a suspension of small particles. This sediment will be composed of hard abrasive materials such as sand, which can cause expensive damage and rapid wear to turbine runners. To remove suspension of small particles from the water flow must be slowed down in settling basins so that the silt particles will settle on the basin floor. The deposit formed is then periodically flushed away.

As shown in the Fig. 2.3 water enters will slow downs to settling speed S_o slit presented will be settled down. The size of the smallest particle allowed into the penstock the maximum speed of the water in the settling basin could be calculated as the slower the water flows the lower the carrying capacity of the water for particles. Increasing the cross section area of the channel the silt can be settling basin down. The silt settled at the bottom is periodically flushed out from the bottom as shown in the Fig.2.3.

2.4 SPILLWAYS

Spillways are designed to permit controlled overflow at certain points along the channel. Flood flows through the intake can be twice the normal channel flow, so the spillway must be large enough for diverting excess flow trough the channel. Floods are

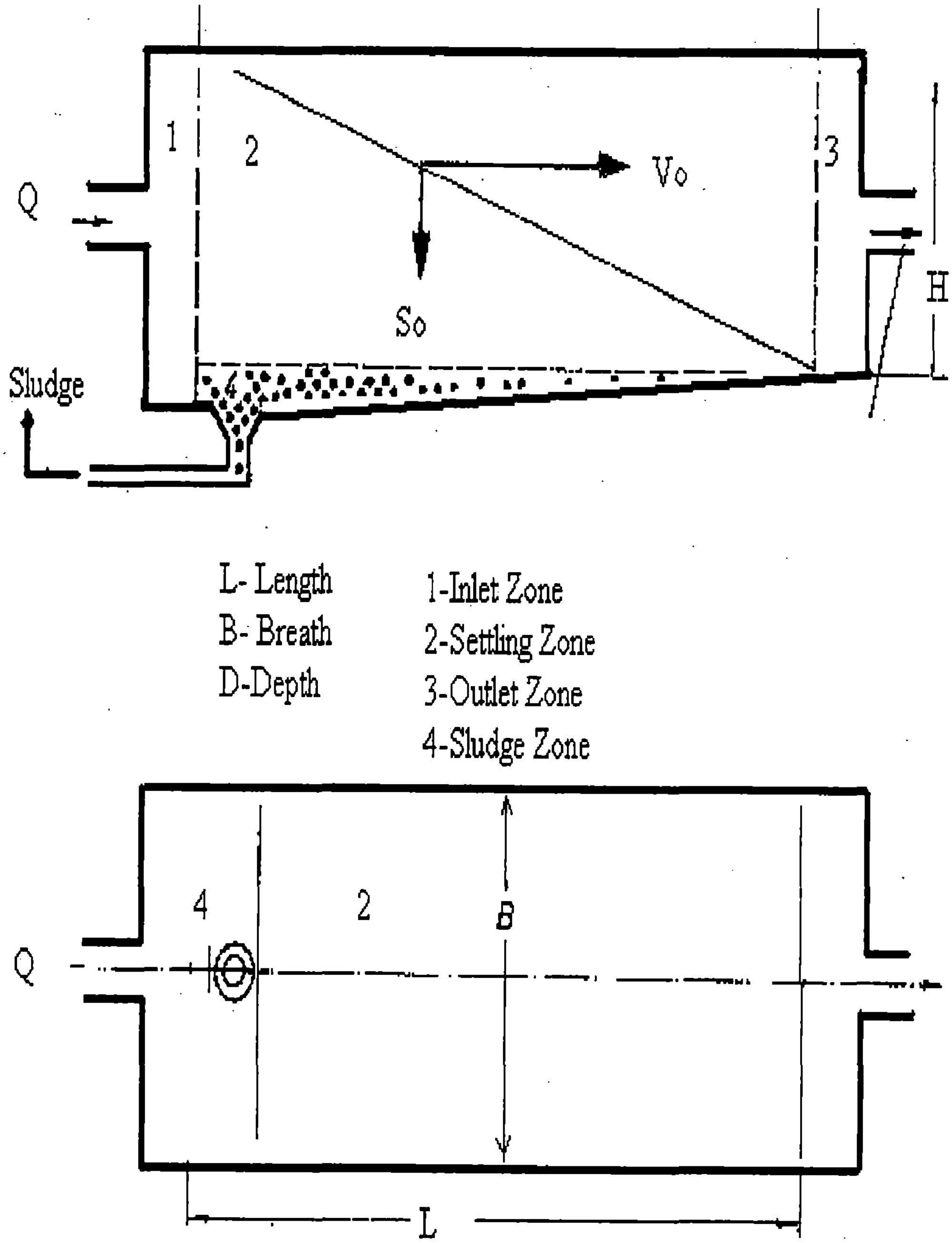

Fig.2.3 Settling Basin [23]

released by spillways so that the water does not overtop and damage, or even destroy, the dam by overflowing the structure [22].The spillway is a flow regulator for the channel. In addition it can be combined with control gates to provide a means of emptying the channel. The following are the different type of the spillways used in small hydro.

2.4.1 Uncontrolled Spillways

It does not have gates when the water rises above the lip or crest of the spillway it begins to be released from the reservoir. The rate of discharge is controlled only by the depth of water within the reservoir. All of the storage volume in the reservoir above the spillway crest can be used only for the temporary storage of floodwater.

2.4.2 Controlled Spillway

This has mechanical structures such as gates to regulate the rate of flow. This allows nearly the full height of the dam to be used for water storage year-around, and flood waters can be released as required by opening one or more gates.

Some spillways are designed like an inverted bell-mouth so that water can enter all around the perimeter (also termed a morning-glory design). In areas where the surface of the reservoir may freeze, bell-mouth spillways are normally fitted with some arrangements for breaking the ice to prevent the spillway from becoming ice-bound. The spill flow must be fed back to the river in a controlled way so that it does not damage the foundations of the channel.

2.5 FOREBAY TANK

The forebay tank forms the connection between the channel and the penstock. Fig.2.4 shows the sectional view of the Fore bay tank. The main purpose is to allow the last silt particles to settle down before the water enters the penstock. Depending on its size it can also serve as a reservoir to store water. A sluice will make it possible to close the entrance to the penstock. In front of the penstock a trash rack need to be installed to prevent large particles to enter the penstock. An under sluice present at the bottom to flush way the silted water.

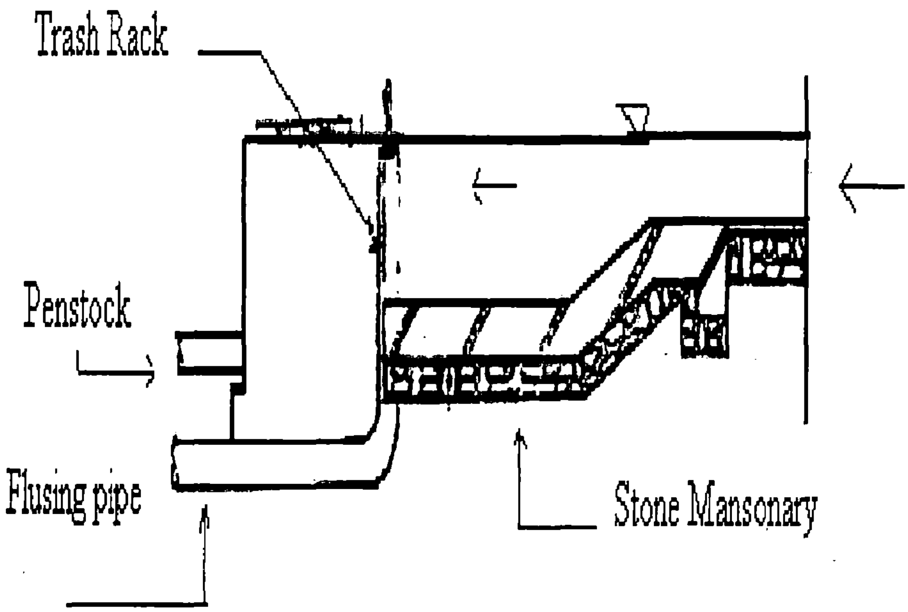

Fig.2.4 Sectional View of the Fore bay Tank [23]

2.7 PENSTOCK

The penstock is the pipe, which conveys water under pressure from the forebay tank to the turbine inlet. The penstock often constitutes a major expense in the total micro hydro budget, as much as 40 % is not uncommon in high head installations, and it is therefore worthwhile optimizing the design. The trade-off is between head loss and capital cost. Head loss due to friction in the pipe decreases with increasing pipe diameter. Conversely, pipe costs increase steeply with diameter. Therefore a compromise between cost and performance is required [9].

The design philosophy is first to identify available pipe options, then to select a target head loss, 5 % of the gross head being a good starting point. The details of the pipes with losses close to this target are then tabulated and compared for cost effectiveness. A smaller penstock may save on capital costs, but the extra head loss may account for lost revenue from generated electricity each year.

Penstocks are generally supplied in standard lengths and have to be joined together on site. Penstock pipelines can either be surface mounted or buried underground. The decision will depend on the pipe material, the nature of the terrain and environmental considerations. Buried pipelines should be ideally being at least 750 mm below ground level, especially when heavy vehicle are likely to cross it. Burying a pipeline removes the biggest eyesore of a hydro scheme and greatly reduces its visual impact.

Penstock is impossible to burry sometimes no option but to run the line above the ground, in which case piers, anchors and thrust blocks will be needed to counteract the forces which can cause undesired pipeline movement.

The three types of forces that need to be designed against are:

i). Weight of the pipes plus water,

ii). Expansion and contraction of the pipe,

iii). Fluid pressure (both static and dynamic).

Support piers are used primarily to carry the weight of the pipes and enclosed water. Anchors are large structures, which represent the fixed points along a penstock, restraining all movements by anchoring the penstock to the ground. A thrust block is used to oppose a specific force, like force at a bend or contraction.

The different support structures can usually be built of rubble masonry or plain concrete. Anchor blocks may need steel reinforcement and triangulated steel frames are sometimes used for support piers.

The size and cost of support structures for a given penstock are minimized by

i). Keeping the penstock closer to the ground ii). Avoiding tight joints

iii). Avoiding soft and unstable ground

2.8 FLOW CONDITIONS

2.8.1 Hydraulic Jump

A hydraulic jump forms when a supercritical flow changes into a sub critical flow. The change in the flow regime occurs with a sudden rise in water surface. Considerable turbulence, energy loss, and air entrainment are produced in the hydraulic jump. A hydraulic jump is used for mixing chemical in water supply systems, for dissipating energy below the artificial channel controls, and as an aeration device to increase the dissolved oxygen in water.

The loss of energy in jump is given by

$$E_l = (y_2 - y_1)^3/(4y_1y_2) \qquad (2.5)$$

Where y_1-conjuate depth before jump

y_2- conjugate depth after jump

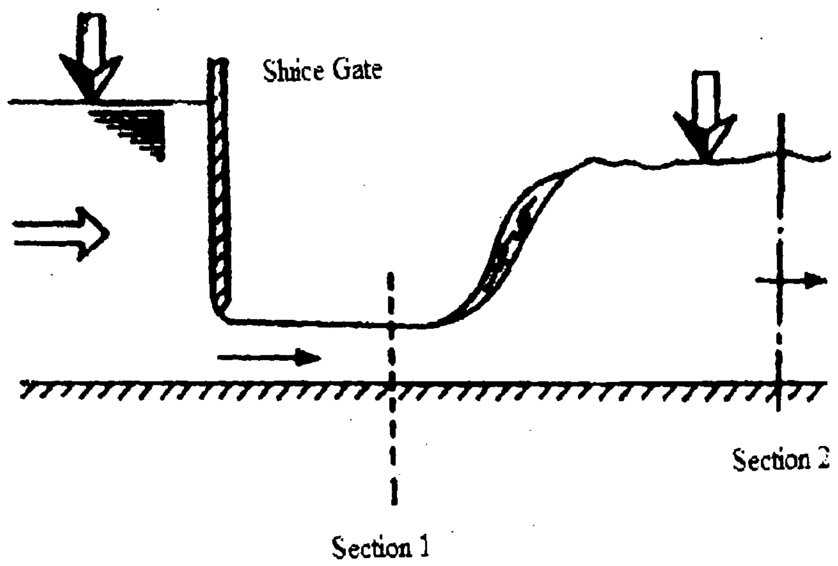

Fig.2.5 Hydraulic Jump [11]

2.8.1 Types of jumps

Hydraulic jumps in horizontal rectangular channels can be classified into five categories as,

2.8.1.1 Undular Jump ($1<F_1<1.7$)

Fig.2.7 shows the flow in Undular jump. The water surface shows undulations. The energy loss in the Undular jump is insignificant.

2.8.1.2 Weak Jump ($1.7<F_1<2.5$)

A series of surface rollers develops on the surface of the hydraulic jump. The water surface down stream of the jump is smooth. Fig. 2.8 shows the flow lines in the Weak jump occurring in the flow. The energy loss in weak jump is small.

2.8.1.3 Oscillating Jump ($2.5<F_1<4.5$)

Fig.2.8 shows the flow lines in the Oscillating jump. The incoming jet oscillates in a random manner between the bed and the surface producing surface waves that persist a considerable distance downstream and can bank erosion.

2.8.1.4 Steady Jump ($4.5<F_1<9.0$)

The jump is well stabilized. Fig. 2.9 shows the flow in the Steady jump. The down stream extremity of the surface roller and the point at which the high velocity jet tends to leave the floor occur in practically the same vertical plane. The energy dissipation in the steady jump ranges from 45 percent at $F_1=4.5$ to 70 percent at $F_1=9.0$.

2.8.1.5 Strong Jump ($9.0<F_1$)

Fig.2.10 shows the flow lines in the strong Jump. The water surface is very rough. The point at which the high- velocity jet tends to leave the floor lies upstream of the down stream extremity of the surface roller. The jump action is rough but effective.

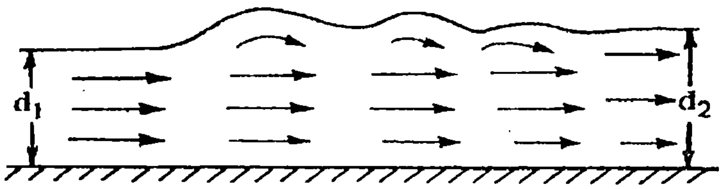

Fig.2.6 Undular Jump [11]

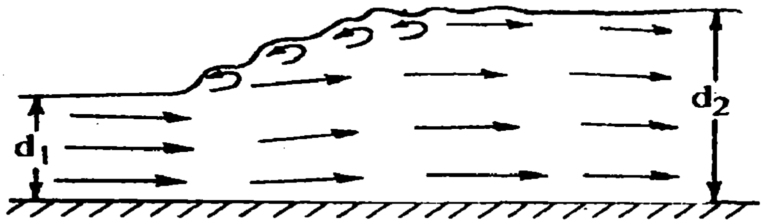

Fig.2.7 Weak Jump [11]

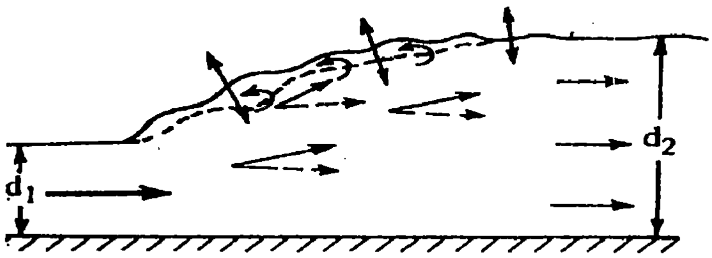

Fig.2.8 Oscillating Jump [11]

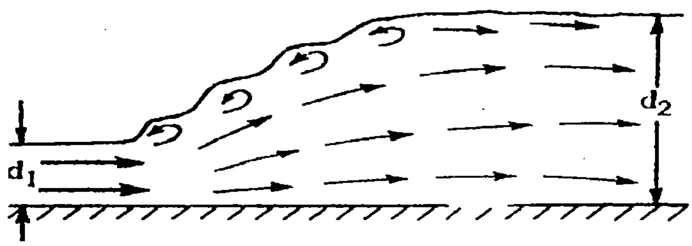

Fig.2.9 Steady Jump [11]

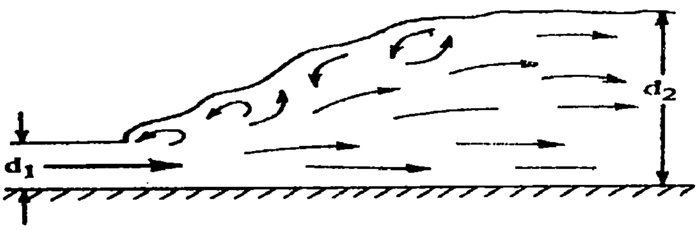

Fig.2.10 Strong Jump [11]

2.8.1.6 Length of the Hydraulic Jump

The length of the Hydraulic Jump is an important parameter that governs the length of the stilling basin. The length is defined as the distance between the front of the jump to a vertical section passing through the down stream extremity of the surface roller. The length is practically constant at about $6y_2$ over the range $4.5<F_1<13$, where y_2 is the depth down stream of the jump.

2.8.2 Flow Trough Sudden Contraction and Expansion

When the pipe has a sudden contraction there is a loss of head due to the increase in velocity of the water flow and to the turbulence [9].

The flow path is so complex that, at least for the time being, it is impossible to provide a mathematical analysis of the phenomenon. The head loss is estimated multiplying the kinetic energy in the smaller pipe, by a coefficient K_c that varies with the index of contraction d/D

$$h_c = K_c\left(\frac{V_2^2}{2g}\right)$$

(2.6)

For an index up to d/D = 0.76, K_c approximately follows the formula

$$K_c = 0.42\left(1 - d^2/_{D^2}\right) \tag{2.7}$$

Over this ratio, K_C is substituted by K_{ex}, coefficient used for sudden expansion. In sudden expansion the loss of head can be derived from the momentum consideration, and is given by

$$h_{ex} = \frac{(V_1 - V_2)^2}{2g} = \left(1 - \frac{V_2}{V_1}\right)^2 \frac{V_1^2}{2g} = \left(1 - \frac{A_1}{A_2}\right)^2 \frac{V_1^2}{2g} = \left(1 - \frac{d^2}{D^2}\right)^2 \frac{V_1^2}{2g} \tag{2.8}$$

Where V_1 is the water velocity in the smaller pipe.

The head loss can be reduced by using a gradual pipe transition, known as confuser.

In the confuser the head loss varies with the confuser angle as it is shown in

Table 2.1 K_c values for different confuser angles [9]

Angle	K'
30^0	0.02
45^0	0.04
60^0	0.07

In the diffuser the analysis of the phenomenon is more complex. Fig. 2.12 shows the experimentally found values of K_{ex} for different diffuser angles. The head loss is given by equation (2.9):

$$h'_{ex} = K'_{ex}\left(\frac{V_1^2 - V_2^2}{2g}\right) \tag{2.9}$$

A submerged pipe discharging in a reservoir is an extreme case of sudden expansion, where V_2, given the size of the reservoir, compared with the pipe, can be considered as zero, and the loss $V_1^2/2g$. An entrance to the pipe is, otherwise, an extreme case of sudden contraction.

2.8.3 Flow Across Bends

Flow across bends in the pipes, experiences an increase of pressure along the outer wall and a decrease of pressure along the inner wall [9]. This pressure unbalance causes a secondary current such as shown in the Fig.2.14. Both movements together the longitudinal flow and the secondary current produces a spiral flow that, at a length of around 100 diameters, is dissipated by viscous friction. The head loss produced in these circumstances depends on the radius of the bend and on the diameter of the pipe. Furthermore, in view of the secondary circulation, there is a secondary friction loss, dependent of the relative roughness, e/d. There is also a general agreement that, in seamless steel pipes, the loss in bends with angles under 90°, is almost proportional to the bend angle.

The problem is extremely complex when successive bends are placed one after another close enough to prevent the flow from becoming stabilized at the end of the

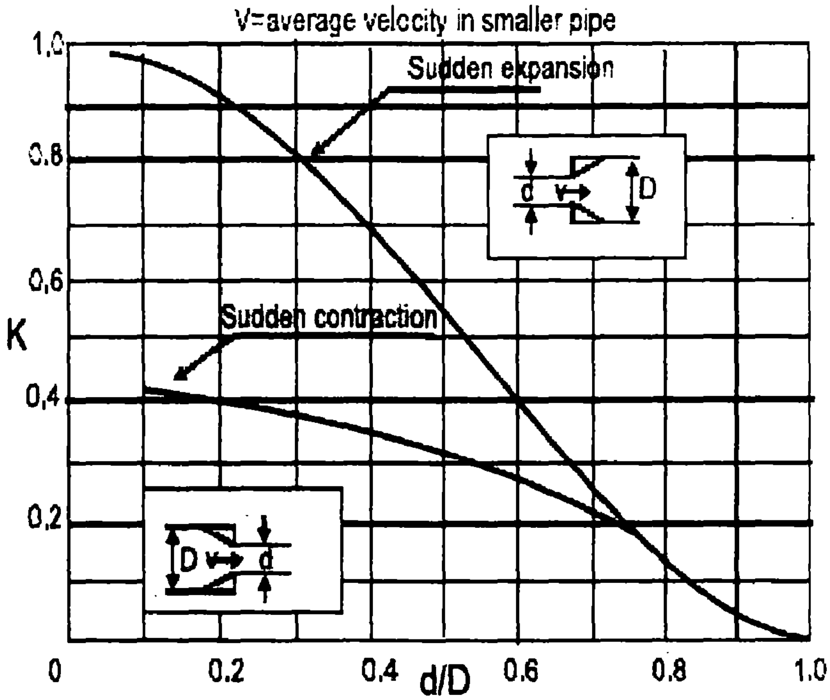

Fig.2.11 Graph between K v/s d/D [9]

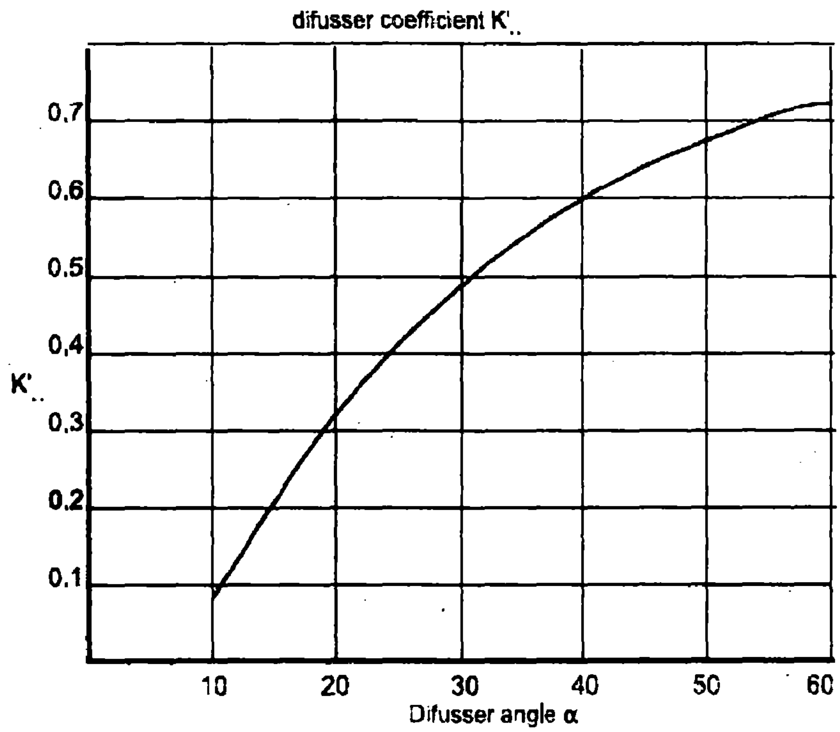

Fig.2.12 Showing Graph between K_c and K_{ex} v/s Difusser angle α [9]

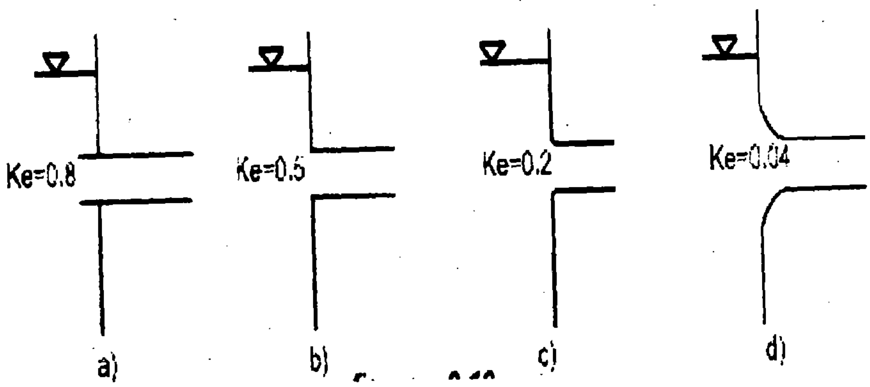

Fig.2.13 The value of the K_e coefficient for different sections [9]

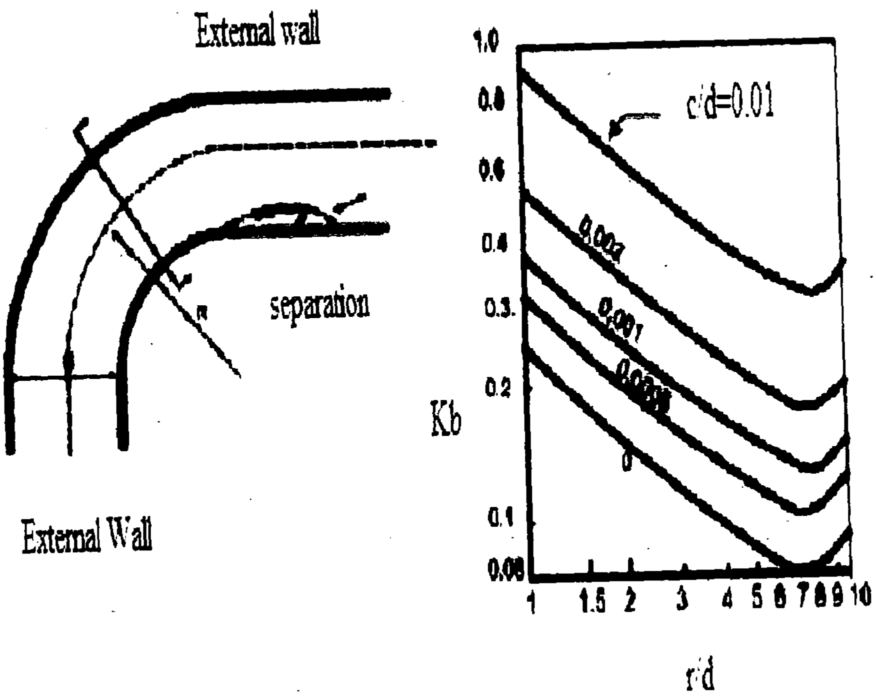

Fig.2.14 Flow Across Bend in a Pipe [9]

bend. This is case hardly ever the case on a small hydro scheme mainly in hilly areas due to site-specific conditions.

2.8.4 Flow Trough Trash Rack

A screen or grill is always required at the entrance of a pressure pipe. The flow of water through the rack also gives rise to a head loss. Though usually small, it can be calculated by a formula due to Kirchmer as given in Eq. (2.10)

$$h_t = K_t \left(\frac{t}{b}\right)^{4/3} \left(\frac{V_o^2}{2g}\right) \sin\phi \tag{2.10}$$

The Fig.2.15 shows the alignment of the trash rack in the forebay tank a head of the penstock. If the grill is not perpendicular but makes an angle b with the water flow (b will have a maximum value of 90° for a grill located in the sidewall of a canal), there will be an extra head loss, as by the equation

$$h_\beta = \frac{V_o^2}{2g} \sin\beta \tag{2.11}$$

2.8.5 Flow Trough Gates and Valves

Valves or gates are used in small hydro scheme to isolate a component from the rest, so they are either entirely closed or entirely open. The Fig 2.16 shows the value of K_v for different kind of valves. Flow regulation is assigned to the distributor vanes or to the needle valves of the turbine. The loss of head produced by the water flowing through an open valve depends on the type and manufacture of the valve.

2.9 TURBINE

A turbine converts energy in the form of falling water into rotating shaft power. The selection of the best turbine for a hydro site depends on the site characteristics [22]. It depends on the speed of the generator, the turbine is expected to produce power under part-flow conditions play a role in the selection. All turbines have a power-speed characteristic. They will tend to run most efficiently at a particular speed, head and flow combination.

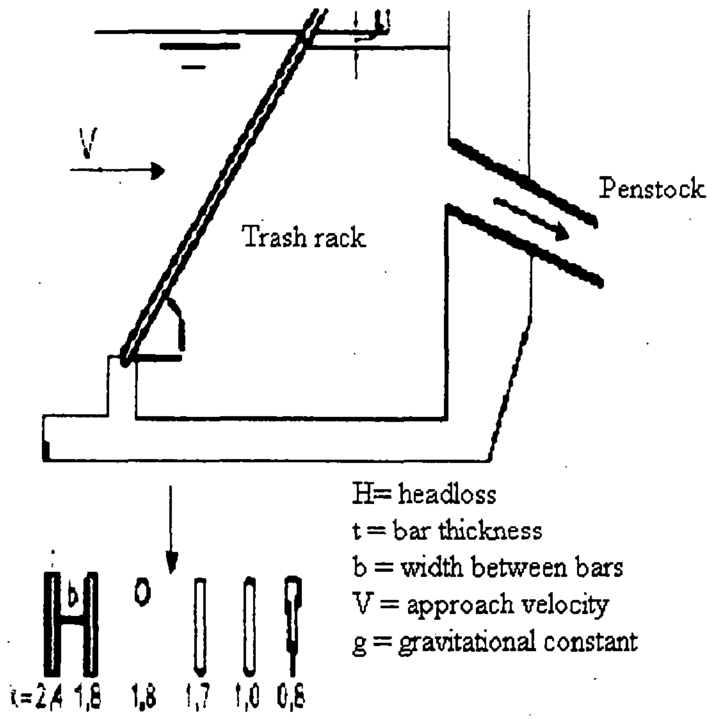

Fig.2.15 Trash Rack Alignment [9]

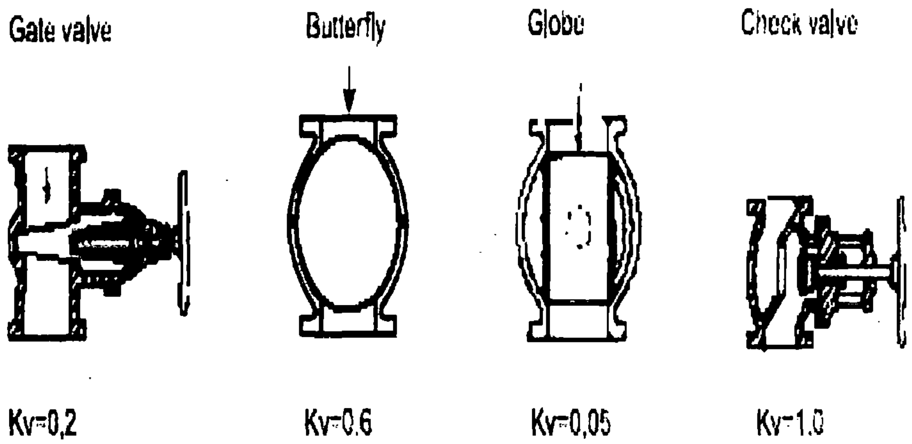

Fig.2.16 Different Types of Valves with K_v Values [9]

Turbines can be classified as high head, medium head or low head machines. Turbines are also divided by their principle way of operating impulse or reaction. The rotating element of a Reaction turbine is fully immersed in water and is enclosed in a pressure casing.

Impulse turbine runner operates at atmospheric pressure, driven by a jet (or jets) of water. The water remains at atmospheric pressure before and after making contact with the runner blades. Nozzle converts the pressurised low velocity water into a high-speed jet. The runner blades deflect the jet so as to maximise the change of momentum of the water and thus maximising the force on the blades.

Impulse turbines are usually cheaper then reaction turbines because there is no need for a specialist pressure casing, carefully engineered clearances.

2.9.1 Pelton Turbines

Pelton turbines are impulse turbines where one or more jets impinge on a wheel carrying on its periphery a large number of buckets. Each jet issues through a nozzle with a needle (or spear) valve to control the flow. They are only used for relatively high heads.

The speed of the turbine under variable load conditions is controlled by using the governor, when turbine approaches runaway speed due to load rejection- the jet may be deflected by a plate so that it does not impinge on the buckets. In this way the needle valve can be closed very slowly, so that overpressure surge in the pipeline is kept to an acceptable minimum. Any kinetic energy leaving the runner is lost and so the buckets are designed to keep exit velocities to a minimum.

The turbine casing only needs to protect the surroundings against water splashing and therefore can be very light. Fig.2.18 shows the sectional view of the Pelton turbine along with the all components.

Table 2.2 Classification of Hydro turbines [22]

Turbines	High Head	Medium Head	Low Head
Impulse turbines	Pelton Turgo	Cross- flow Multi-jet Pelton Turgo	Cross-flow
Reaction turbines		Francis	Propeller Kaplan

2.9.2 Turgo Impulse Turbine

The Turgo impulse turbine can operate under a head in the range of 30-300 m. Like the Pelton it is an impulse turbine, but its buckets are shaped differently and the jet of water strikes the plane of its runner at an angle of 20°. Water enters the runner through one side of the runner disk and emerges from the other as shown in Fig 2.18. The volume of water a Pelton turbine can admit is limited because the water leaving each bucket interferes with the adjacent ones, the Turgo runner does not present this problem.

The resulting higher runner speed of the Turgo makes direct coupling of turbine and generator more likely, improving its overall efficiency and decreasing maintenance cost.

2.9.3 Cross-flow Turbines

This cross flow turbine is used for a wide range of heads overlapping those of Kaplan, Francis and Pelton. It can operate with discharges between 20 litres/sec and 10 m^3/sec and heads between 1 and 200 m [9].

Water enters the turbine, directed by one or more guide-vanes located in a transition piece upstream of the runner, and through the first stage of the runner, which

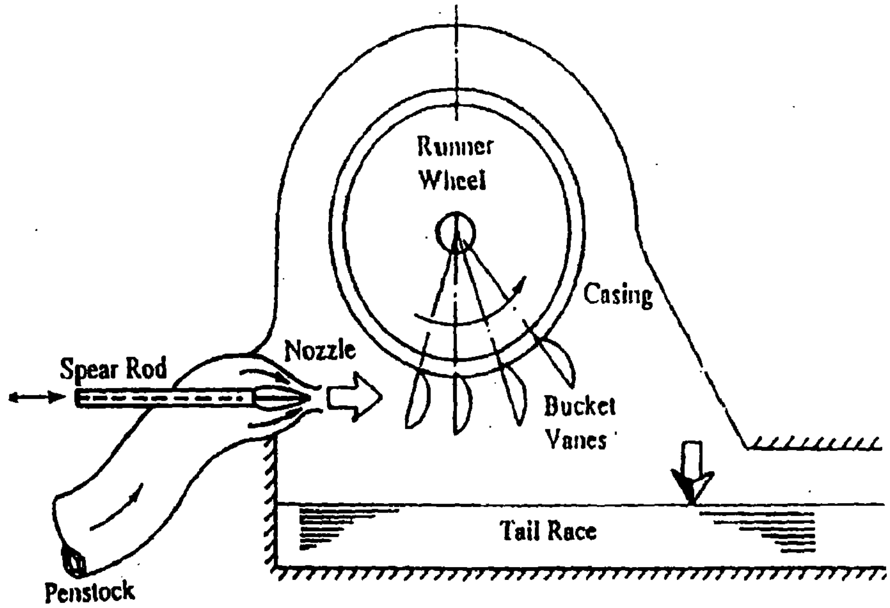

Fig.2.17 Pelton Turbines jet hitting the bucket [11]

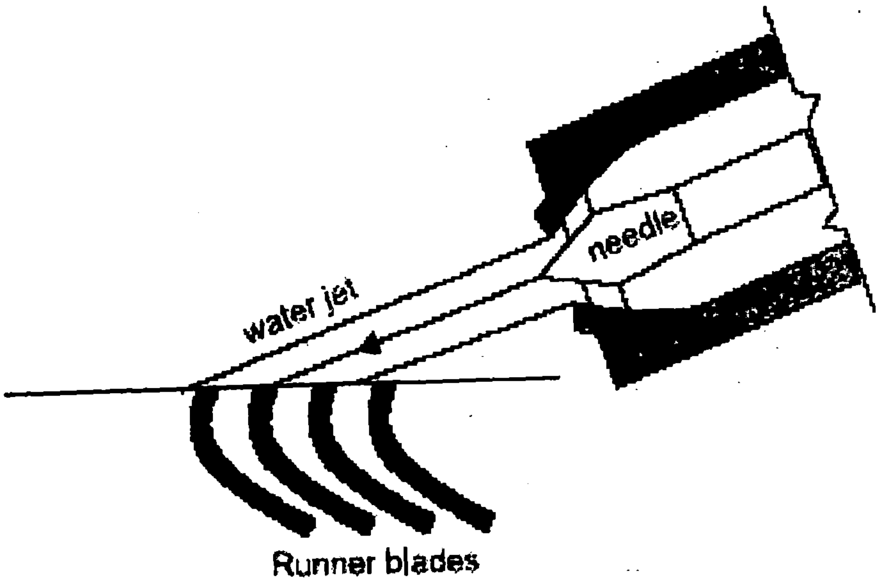

Fig.2.18 Flow across Turgo Impulse Turbine [9]

runs full with a small degree of reaction. Flow leaving the first stage attempt to crosses the open center of the turbine. As the flow enters the second stage, a compromise direction is achieved which causes significant shock losses. The runner is built from two or more parallel disks connected near their rims by a series of curved blades). Their efficiency lowers than conventional turbines, but remains at practically the same level for a wide range of flows and heads (typically about 80%). The Fig.2.19 shows the section of the cross flow turbine.

Impulse turbines are suitable for micro-hydro applications compared with reaction turbines because they have the following advantages

i). Greater tolerance of sand and other particles in the water.

ii). Better access to working parts.

iii). No pressure seals around the shaft.

iv). Easier to fabricate and maintain.

v). Better part-flow efficiency.

The major disadvantage of impulse turbines is that they are mostly unsuitable for low-head sites because of their low specific speeds too great an increase in speed would be required of the transmission to enable coupling to a standard alternator. The Cross-flow, Turgo and multi-jet Pelton are suitable at medium heads.

2.9.4 Francis Turbines

Francis turbines are radial flow reaction turbines, with fixed runner blades and adjustable guide vanes, used for medium heads. In the high speed Francis the admission is always radial but the outlet is axial. The water proceeds through the turbine as if it was enclosed in a closed conduit pipe, moving from a fixed component, the distributor, to a moving one, the runner, without being at any time in contact with the atmosphere. Guide vanes, whose purpose is to control the discharge going into the runner, rotate around in their axes, by connecting rods attached to a large ring that synchronize the movement off all vanes. It should be emphasized that the size of the spiral casing contrasts with the lightness of a Pelton casing. The rotating ring is attached to the links that operate the guide vanes the adjustable vanes and their mechanism, both in open and closed position. The wicket gates can be used to shut off the flow to the turbine in emergency situations, although their use does not avoid the installation of a butterfly valve at the entrance to the turbine. Francis turbines can be set in an open flume or attached to a penstock. For small heads and power open flumes are commonly employed. Steel spiral casings are used for higher heads, designing the casing so that the tangential velocity of the water is constant along the consecutive sections around the circumference. Fig.2.20 shows a Francis across the runner in perspective from the outlet end. Small runners are usually made in aluminum bronze castings. Large runners are fabricated from curved stainless steel plates, welded to a cast steel hub.

In reaction turbines, to reduce the kinetic energy still remaining in the water leaving the runner a draft tube or diffuser stands between the turbine and the tailrace. A well-designed draft tube allows, within certain limits, the turbine to be installed above the tail water elevation without losing any head. As the kinetic energy is proportional to the square of the velocity one of the draft tube objectives is to reduce the outlet velocity. An efficient draft tube would have a conical section but the angle cannot be too large, otherwise flow separation will occur. The optimum angle is 7° but to reduce the draft tube length, and therefore its cost, sometimes angles are increased up to 15°.

Draft tubes are particularly important in high-speed turbines, where water leaves the runner at very high speeds. In horizontal axis machines the spiral casing must be well anchored in the foundation to prevent vibration that would reduce the range of discharges accepted by the turbine.

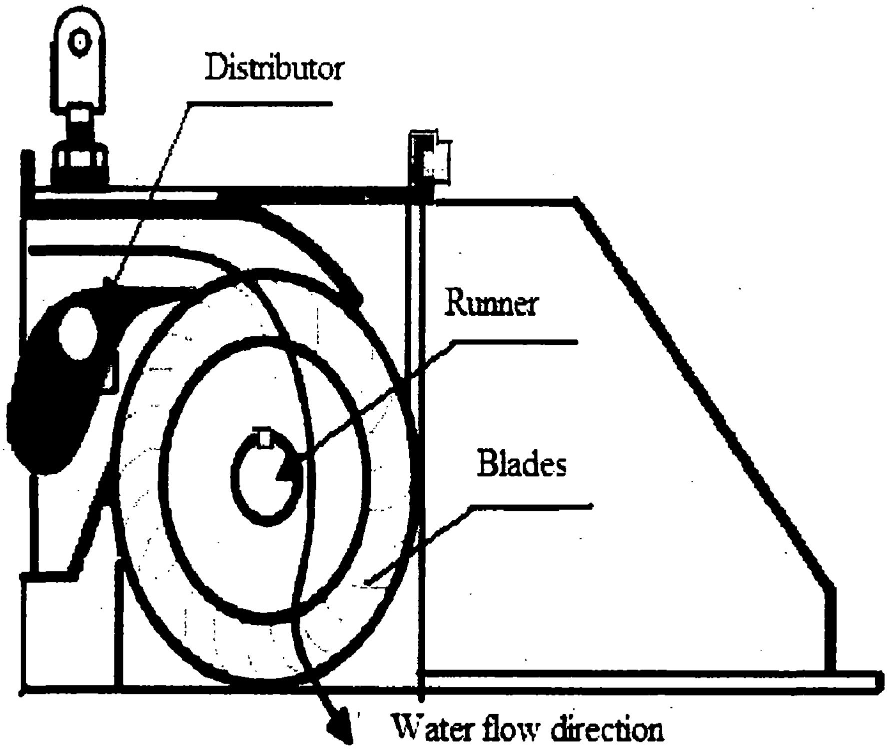

Fig.2.19 Flow across Cross Flow Turbine [9]

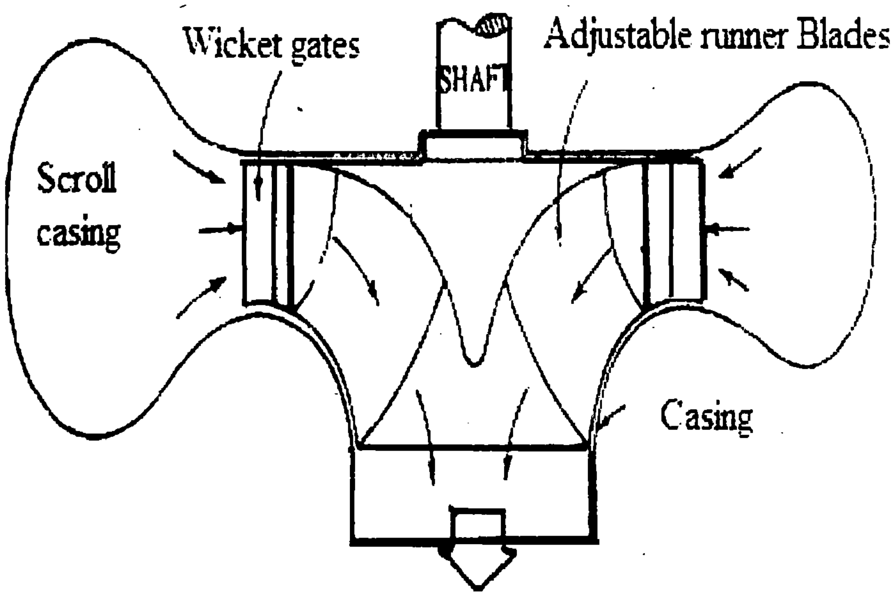

Fig.2.20 Francis Turbine Runner [11]

2.9.5 Kaplan and Propeller Turbines

Kaplan and propeller turbines are axial-flow reaction turbines, generally used for low heads [9]. The Fig 2.21 shows the view the flow in Kaplan turbine. The Kaplan turbine has adjustable runner blades and may or may not have adjustable guide- vanes. If both blades and guide-vanes are adjustable it is described as double regulated. If the guide-vanes are fixed it is single-regulated. Unregulated propeller turbines are used when both flow and head remain practically constant. The double-regulated Kaplan is a vertical axis machine with a scroll case and a radial wicket-gate configuration. The flow enters radially inward and makes a right angle turn before entering the runner in an axial direction. The control system is designed so that the variation in blade angle is coupled with the guide-vanes setting in order to obtain the best efficiency over a wide range of flows. The blades can rotate with the turbine in operation, through links connected to a vertical rod sliding inside the hollow turbine axis.

2.9.6 Bulb Turbines

Bulb units are derived from Kaplan turbines, with the generator contained in a waterproofed bulb submerged in the flow [9]. The generator cooled by pressurized air is lodged in the bulb. Only the electric cables, duly protected, leave the bulb.

The Bulb turbine is used for the lowest heads. It is characterized by having the essential turbine components as well as the generator inside a bulb, from which the name is developed. A main difference from the Kaplan turbine is moreover that the water flows with a mixed axial-radial direction into the guide vane cascade and not through a scroll casing. The guide vane spindles are inclined (normally 60^{0}) in relation to the turbine shaft. Contrary to other turbine types this results in a conical guide vane cascade. The Bulb turbine runner is of the same design as for the Kaplan turbine, and it may also

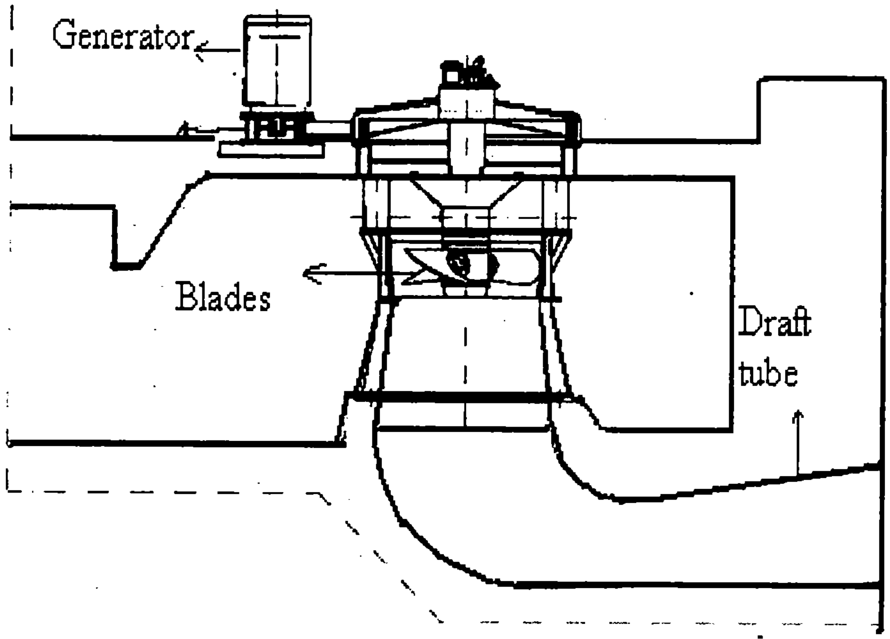

Fig.2.21 Kaplan Turbine [11]

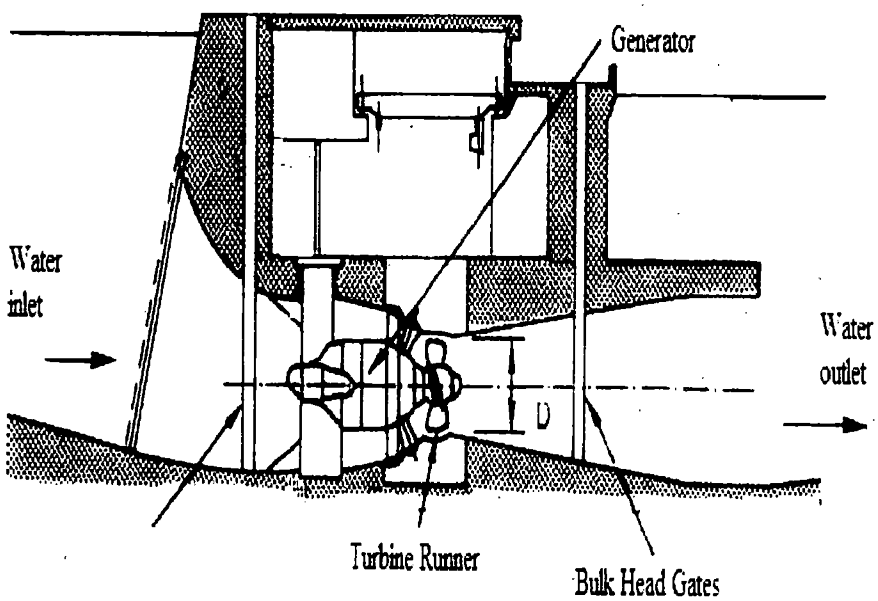

Fig.2.22 Bulb Turbine [11]

have different numbers of blades depending on the head and water flow. Fig.2.22 shows the cross sectional view of the flow across the Bulb turbine.

The reaction turbines are the Francis turbine and the propeller turbine. A special case of the propeller turbine is the Kaplan. Specific speed is high in case of reaction turbines rotate faster than impulse turbines given the same head and flow conditions. This has the very important consequences that a reaction turbine can be coupled directly to an alternator without requiring a speed-increasing drive system. Significant cost savings are made in eliminating the drive and the maintenance of the hydro unit is very much simpler. The Francis turbine is suitable for medium heads, while the propeller is more suitable for low heads.

The reaction turbines require more sophisticated fabrication than impulse turbines because they involve the use of larger and more intricately profiled blades together with carefully profiled casings. The extra expenses involved is offset by high efficiency and the advantages of high running speeds at low heads from relatively compact machines. Most reaction turbines tend to have poor part-flow efficiency characteristics.

2.9.7 Reverse Pump as Turbine

Centrifugal pumps can be used as turbines by passing water through them in reverse. The potential advantages are the low cost due to mass production, the availability of spare parts. The disadvantages are the poorly understood performance characteristics and very poor part-flow efficiency.

2.9.8 DRAFT TUBE

The water after doing work on the runner passes on to the tailrace through a draft tube, which is a riveted steel plate pipe or a concrete tunnel, its cross section gradually increasing towards the outlet. The draft tube is a conduit, which connects the runner exit to the tailrace. The tube should be drowned approximately one meter below the lowest tailrace level [11].

2.9.8.1 Functions of Draft Tube

The following are the functions of the Draft Tube

i. The water is discharged freely from the runner; turbine will work under a head equal to the height of the headrace water level above the runner outlet. An airtight draft tube connects the runner to the tailrace, workable head is increased by an amount equal to the height of runner outlet above tailrace.

ii. The draft tube will thus, permit a negative suction head to be established at the runner outlet thus making it possible to install the turbine above the tail race without loss of head.

iii. The water leaving the runner still possesses a high velocity and this kinetic energy would be lost if it is discharged freely as in a Pelton turbine. With the increase in net working head on the turbine, output will also increases, thus raising the efficiency of turbine.

The following are some types of draft tubes used in the Small Hydropower Station

i. Straight Divergent Tube
ii. Moody Spreading Tube
iii. Simple Elbow Tube
iv. Elbow Type with a Circular Inlet and a Rectangular outlet section

The Fig. 2.23 shows the Straight Divergent, Tube Moody Spreading Tube and Elbow type with a Circular Inlet and a Rectangular outlet section draft tubes.

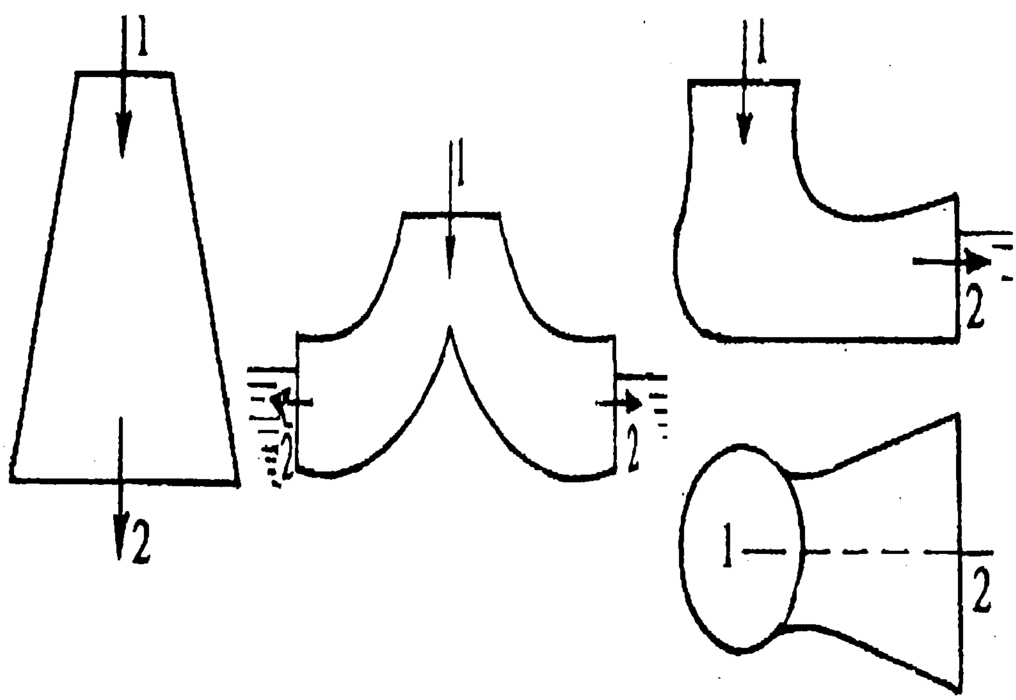

Fig.2.23 Straight Divergent, Tube Moody Spreading Tube and Elbow Type with a Circular Inlet and a Rectangular outlet section [11]

CONCEPTS OF COMPUTATIONAL FLUID DYNAMICS

3.1 GENERAL

The basic concepts and the numerical methods involved in the CFD has been explained under this chapter. The strategy involved in the CFD for the continuous and the discontinuous problem domains are explained. The conditions required for the convergence of the solution and the grid has also been discussed along with the one-dimensional case.

3.2 NEED AND STRATERGY FOR CFD

Applying the fundamental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass equation is

$$\frac{\partial \rho}{\partial t} + \nabla .(\rho V) = 0 \tag{3.1}$$

Conservation of momentum equation is

$$\rho \frac{\partial V}{\partial t} + \rho(V.\nabla)V = -\nabla p + \rho.g + \nabla.\ \tau_{ij} \tag{3.2}$$

These equations along with the conservation of energy equation form a set of coupled, nonlinear partial deferential equations. It is not possible to solve these equations analytically for most engineering problems. It is possible to obtain approximate computer-based solutions to the governing equations for a variety of engineering problems. This is the subject matter of Computational Fluid Dynamics (CFD).

The strategy of CFD is to replace the continuous problem domain with a discrete domain using a grid. In the continuous domain, each flow variable is defined at every point in the domain. For instance, the pressure p in the continuous 1D domain shown in the Fig. 3.1 below would be given as

$$p = p\ (x),\ 0 < x < 1 \tag{3.3}$$

In discrete domain, each flow variable is defined only at the grid points. So, in the discrete domain shown below, the pressure would be defined only at the N grid points.

$$p_i = p\ (x_i),\ i = 1, 2, \ldots, N \tag{3.4}$$

In a CFD solution, one would directly solve for the relevant flow variables only at the grid points. The values at other locations are determined by interpolating the values at the grid points. The governing partial differential equations and boundary conditions are defined in terms of the continuous variables p, V etc. The discrete system is a large set of coupled, algebraic equations in the discrete variables. Setting up the discrete system and solving it (which is a matrix inversion problem) involves a very large number of repetitive calculations and is done by the digital computer. This idea can be extended to any general problem domain.

3.3 DISCRETIZATION USING THE FINITE-DIFFERENCE METHOD

The fundamental ideas underlying CFD by applying them to the following simple 1D equation is as shown in the below,

$$\frac{du}{dx} + u^m = 0\ ; 0 \le x \le 1\ ; u(0) = 1 \tag{3.5}$$

By considering case where $m = 1$ the equation is linear. Then considering the $m =$ 2 case where the equation is nonlinear. Driving the discrete representation of the above equation with $m = 1$ on the following grid,

The grid has four equally spaced grid points with Δx being the spacing between successive points as show in the Fig.3.2. The governing equation is valid at any grid point, was given by

$$\left(\frac{du}{dx}\right)_i + u_i = 0 \tag{3.6}$$

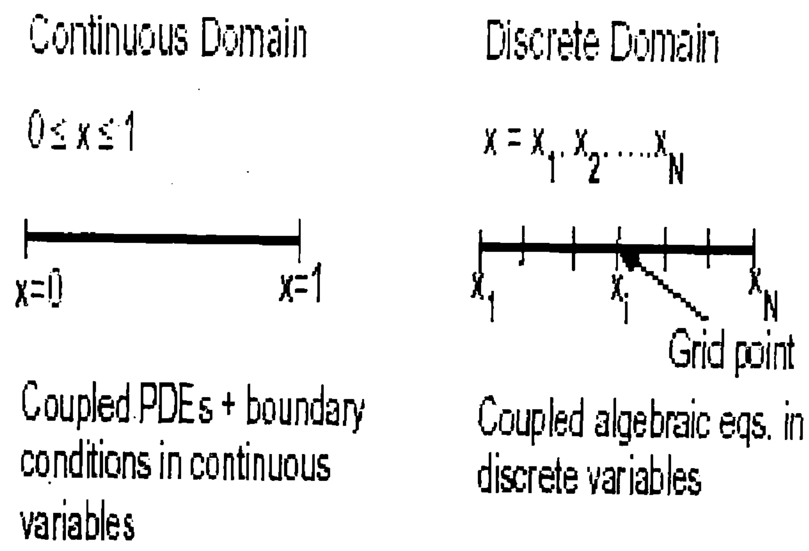

Fig. 3.1 Continuous and Discrete domain [4]

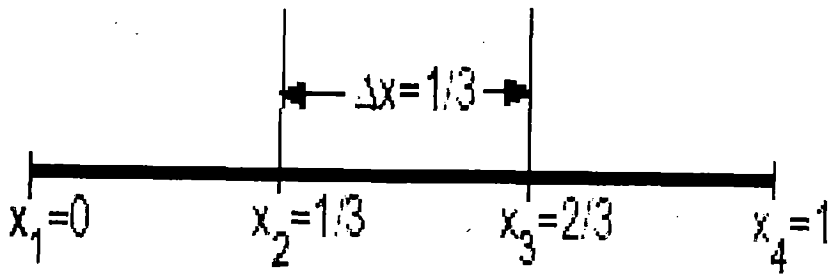

Fig. 3.2 Discrete representation [4]

Subscript i represent the value at grid point x_i. To get an expression for $(du/dx)_i$ in terms of u at the grid points, by expanding u_{i-1} in a Taylor's series

$$u_{i-1} = u_i - \Delta x\,(du/dx)_i + O(\Delta x)^2 \tag{3.7}$$

Rearranging the Eq. 3.7 gives the following Eq. 3.8

$$\left(\frac{du}{dx}\right)_i = \frac{u_i - u_{i-1}}{\Delta x} + O(\Delta x) \tag{3.8}$$

The error in $(du/dx)_i$ due to the neglected terms in the Taylor's series is called the truncation error. The truncation error is of the order $O(\Delta x)$, this discrete representation is termed first order accurate.

Using (3.6) in (3.8) and excluding higher-order terms in the Taylor's series, we can get the following discrete equation:

$$\frac{u_i - u_{i-1}}{\Delta x} + u_i = 0 \tag{3.9}$$

This method of deriving the discrete equation using Taylor's series expansion is called the finite-difference method. The most of commercial CFD codes use the finite-volume or finite-element methods, which are better suited for modeling flow past complex geometries. FLUENT code uses the finite-volume method and ANSYS uses the finite-element method.

3.4 DISCRETIZATION USING THE FINITE-VOLUME METHOD

In the finite-volume method, a quadrilateral is considered as cell and a grid point as a node. In 2D, one could also have triangular cells. In 3D, cells are usually hexahedral, tetrahedral, or prisms. In the finite-volume approach, the integral form of the conservation equations are applied to the control volume defined by a cell to get the discrete equations for the cell. The integral form of the continuity equation for steady, incompressible flow is

$$\int_S \vec{V} \cdot \hat{n}\, dS = 0 \tag{3.10}$$

The integration is over the surface S of the control volume and ^n is the outward normal at the surface. The Eq. (3.10) means that the net volume flow into the control volume is zero.

The velocity at face i is taken to be $\vec{V}_i = u_i\hat{i} + v_i\hat{j}$. Applying the mass conservation equation (3.10) to the control volume defined by the cell gives

$$-u_1\Delta y - v_2\Delta x + u_3\Delta y + v_4\Delta x = 0 \qquad (3.11)$$

This is the discrete form of the continuity equation for the cell. It is equivalent to summing up the net mass flow into the control volume and setting it to zero. It ensures that the net mass flow into the cell is zero that mass is conserved for the cell. The values at the cell centers are stored. The face values u_1, v_2, etc. are obtained by suitably interpolating the cell-center values at adjacent cells.

While using FLUENT or another finite-volume code, it's useful to remind that the code is finding a solution such that mass, momentum, energy and other relevant quantities are being conserved for each cell.

3.5 DISCRETE SYSTEM AND BOUNDARY CONDITIONS

The discrete equation obtained by using the finite-difference method was,

$$\frac{u_i - u_{i-1}}{\Delta x} + u_i = 0 \qquad (3.12)$$

Rearranging Eq. (3.12),

$$-u_{i-1} + (1 + \Delta x)u_i = 0 \qquad (3.13)$$

Applying this equation to the 1D grid shown earlier at grid points $i = 2, 3, 4$ gives

$$\begin{aligned} -u_1 + (1 + \Delta x)u_2 &= 0 \; (i = 2) \\ -u_2 + (1 + \Delta x)u_3 &= 0 \; (i = 3) \\ -u_3 + (1 + \Delta x)u_4 &= 0 \; (i = 4) \end{aligned} \qquad (3.14i, 3.14ii, 3.14iii)$$

The discrete equation cannot be applied at the left boundary (i=1) since u_{i-1} is not defined here. Instead, we use the boundary condition to get u_1=1.

Equations (3.14i) form a system of four simultaneous algebraic equations in the four unknowns u_1, u_2, u_3 and u_4. It's convenient to write this system in matrix form,

$$\begin{bmatrix} 1 & 0 & 0 & 0 \\ 1 & 1+\Delta x & 0 & 0 \\ 0 & 1 & 1+\Delta x & 0 \\ 0 & 0 & 1 & 1+\Delta x \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} 1 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

(3.16)

In a general situation, apply the discrete equations to the grid points (or cells in the finite-volume method) in the interior of the domain. For grid points (or cells) at or near the boundary, apply a combination of the discrete equations and boundary conditions. In the end, obtain a system of simultaneous algebraic equations with the number of equations being equal to the number of independent discrete variables. The process is essentially the same as for the model equation above with the details being much more complex.

FLUENT, like other commercial CFD codes, offers a variety of boundary condition options such as velocity inlet, pressure inlet and pressure outlet. It is very important to specify the proper boundary condition.

3.6 SOLUTION OF DISCRETE SYSTEM

The discrete system represented by Eq. (3.16) form 1D example can be inverted to obtain the unknowns at the grid points. Solving for u_1, u_2, u_3 and u_4 in turn and using $\Delta x = 1/3$, we get

$$u_1 = 1 \; u_2 = 3/4 \; u_3 = 9/16 \; u_4 = 27/64 \quad (3.17)$$

The exact solution for the 1D example is calculated to be $u_{exact} = \exp(-x)$. Fig.3.5 shows the comparison of the discrete solution obtained on the four-point grid with the exact solution. In a practical CFD application, have thousands to millions of unknowns in the discrete system, by using the Gaussian elimination procedure naively to invert the matrix, it will long time. Lot of work goes into optimizing the matrix inversion in order to minimize the CPU time and memory required. The matrix to be inverted is sparse most of the entries in it are zeros since the discrete equation at a grid point or cell will contain only quantities at the neighboring points or cells. A CFD code would store only the non-zero values to minimize memory usage. It would also generally use an iterative procedure to invert the matrix, the longer one iterates, the closer one gets to the true solution for the matrix inversion.

3.7 GRID CONVERGENCE

Developing the finite-difference approximation for the 1D example, there is an truncation error in discrete system is $O(\Delta x)$. By increasing the number of grid points it is expected Δx is reduced, the error in the numerical solution would decrease and the agreement between the numerical and exact solutions. By considering the effect of increasing the number of grid points N on the numerical solution of the 1D problem. Consider $N = 8$ and $N = 16$ in addition to the $N = 4$ case. By repeating the assembly and solution steps for the discrete system on each of these additional grids. The Fig. 3.5 compares the results obtained on the three grids with the exact solution. The numerical error decreases as the number of grid points is increased.

The numerical solutions obtained on different grids agree to within a level of tolerance specified, they are referred to as converged solutions. The concept of grid convergence applies to the finite-volume approach also where the numerical solution, becomes independent of the grid as the cell size is reduced. It is very important to investigate the effect of grid resolution on the solution in every CFD problem.

3.8 ITERATIVE CONVERGENCE

The linearization and matrix inversion error tends to zero. By continuing the iteration process until some selected measure of the difference between u_g and u, referred to as the residual, is small enough. The residual R as the RMS value can be defined as the difference between u and u_g on the grid

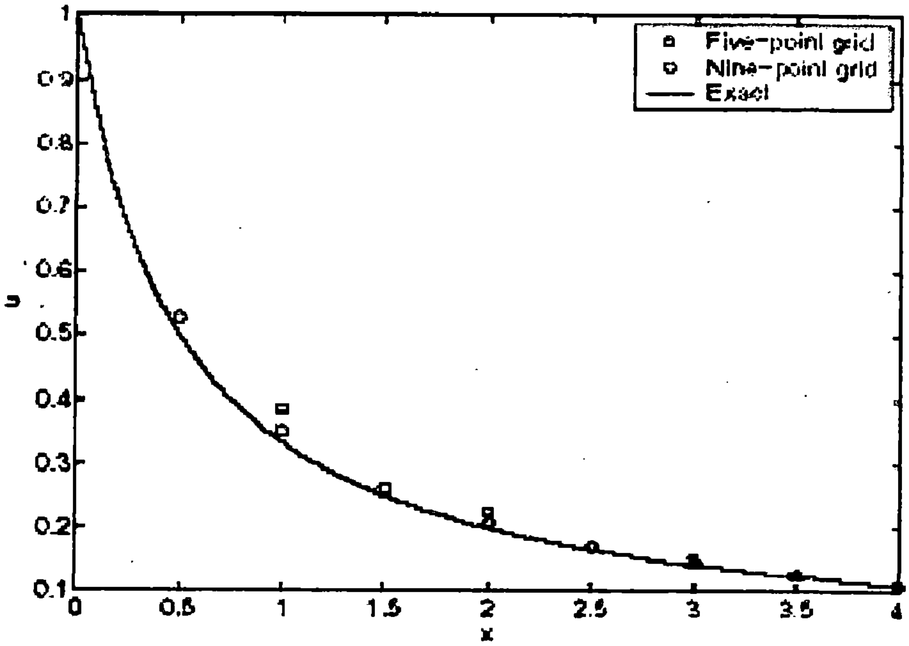

Fig. 3.4 Discrete solution obtained on the four-point grid with the exact solution [4]

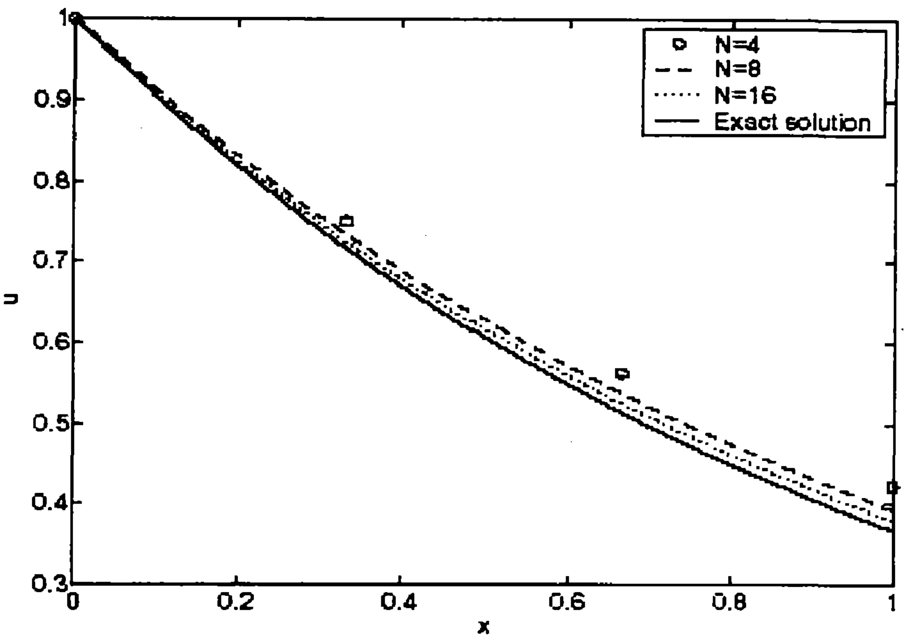

Fig. 3.5 Results obtained on the three grids (N=4, 8, 16) with the exact solution [4]

$$R \equiv \sqrt{\frac{\sum_{i=1}^{N} \left(u_i - u_{gi}\right)^2}{N}} \tag{3.18}$$

It's useful to scale this residual with the average value of u in the domain. An unscaled Residual of, 0.01 would be relatively small if the average value of u in the domain is 5000 but would be relatively large if the average value is 0.1. Scaling ensures that the residual is a relative rather than an absolute measure. Scaling the above residual by dividing by the average value of u gives

$$R = \left(\sqrt{\frac{\sum_{i=1}^{N} \left(u_i - u_{gi}\right)^2}{N}} \right) \left(\frac{N}{\sum_{i=1}^{N} u_i} \right) = \frac{\sqrt{N \sum_{i=1}^{N} \left(u_i - u_{gi}\right)^2}}{\sum_{i=1}^{N} u_i} \tag{3.19}$$

For the nonlinear 1D case, by takeing the initial guess at all grid points to be equal to the value at the left boundary i.e. $U_g^{(1)} = 1$. In each iteration, by update u_g, sweep from right to left on the grid updating, in turn, u_4, u_3 and u_2 using (3.18) and calculating the residual using (3.19). The iterations will terminate when the residual falls below 10^{-9} (which is referred to as the convergence criterion). The iterative process converges to a level smaller than 10^{-9} in just 6 iterations. In more complex problems, a lot more iterations would be necessary for achieving convergence.

The solution after 2, 4 and 6 iterations and the exact solution are shown in the Fig.3.6. It can easily be verified that the exact solution is given by the solutions for iterations 4 and 6 are indistinguishable on the graph. This is another indication that the solution has converged. The converged solution doesn't agree well with the exact solution because we are using a coarse grid for which the truncation error is relatively large. The iterative convergence error, which is of order 10^{-9}, is swamped out by the truncation error of order 10^{-1}. So driving the residual down to 10^{-9} when the truncation error is of order 10^{-1} is a waste of computing resources. In a good calculation, both errors would be of comparable level and less than a tolerance level chosen by the user. The agreement between the numerical and exact solutions should get much better on refining the grid as was the case for $m = 1$. The Fig.3.7 shows the Residual convergence of the equations along with the u and X.

Some points to note

i. Different codes use slightly different definitions for the residual.

ii. In the FLUENT code, residuals are reported for each conservation equation. A discrete Conservation equation at any cell can be written in the form LHS = 0. For any iteration, the current solution to compute the LHS, it won't be exactly equal to zero, with the deviation from zero being a measure of how far from achieving Convergence. So FLUENT calculates the residual as the (scaled) mean of the absolute value of the LHS over all cells.

iii. The convergence criterion for each conservation equation is problem and code-dependent. It's a good to start with the default values in the code.

3.9 NUMERICAL STABILITY

The iterations converged very rapidly with the residual falling below the convergence criterion of 10^{-9} in just 6 iterations. In complex problems, the iterations converge more slowly and in some instances, may even diverge. To know a priori the conditions under which a given numerical scheme converges. This is determined by performing a stability analysis of the numerical scheme. A numerical method is referred to as being stable when the iterative process converges and as being unstable when it diverges. It is not possible to carry out an exact stability analysis for the Euler or Navier-Stokes equations. But a stability analysis of simpler, model equations provides useful insight and approximate conditions for stability.

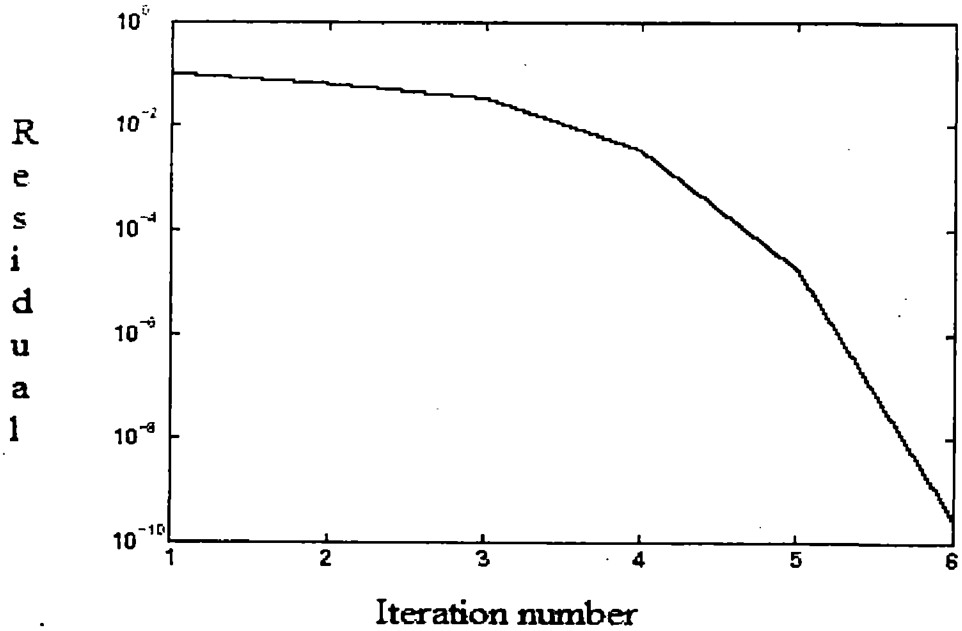

Fig. 3.6 Iteration v/s Residual [4]

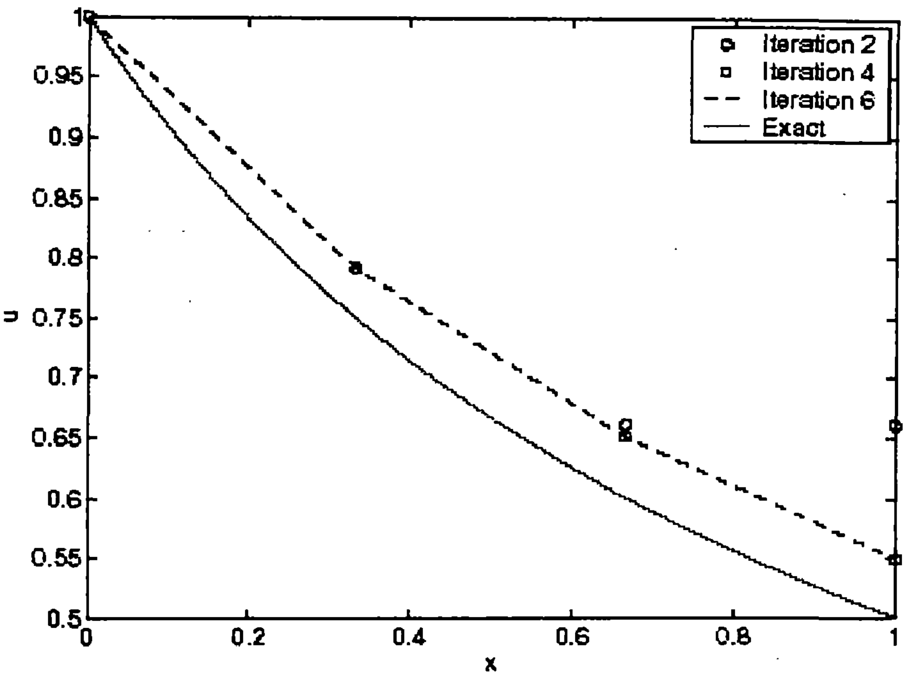

Fig.3.7 Residual convergence [4]

A common strategy used in CFD codes for steady problems is to solve the unsteady equations and march the solution in time until it converges to a steady state. A stability analysis is usually performed in the context of time marching. While using time marching to a steady state, only interested in accurately obtaining the asymptotic behavior at large times. So like to take as large a time-step Δt as possible to reach the steady state in the least number of time-steps.

There is usually a maximum allowable time-step Δt_{max} beyond which the numerical scheme is unstable. If $\Delta t > \Delta t_{max}$, the numerical errors grow exponentially in time causing the solution to diverge from the steady-state result. The value of Δt_{max} depends on the numerical discretization scheme used. There are two classes of numerical schemes, explicit and implicit, with very different stability characteristics.

3.10 TURBULENCE MODELING

There are two different states of flows that are easily identified and distinguished laminar flow and turbulent flow. Laminar flows are characterized by smoothly varying velocity fields in space and time in which individual laminae move past one another without generating cross currents. These flows arise when the fluid viscosity is sufficiently large to damp out any perturbations to the flow that may occur due to boundary imperfections or other irregularities. These flows occur at low-to-moderate values of the Reynolds number. The turbulent flows are characterized by large, nearly random fluctuations in velocity and pressure in both space and time. These fluctuations arise from instabilities that grow until nonlinear interactions cause them to break down into finer and finer whirls that eventually are dissipated (into heat) by the action of viscosity.

Turbulent flows occur in the opposite limit of high Reynolds numbers. The typical time history of the flow variable u at a fixed point in space as shown in Fig.3.8. The dashed line through the curve indicates the average velocity.

The three types of averages are as shown below,

1. Time average
2. Volume average
3. Ensemble average

The concepts discussed in this chapter have been applied to study the flow in the power channel of the Small hydropower plant and presented in the next chapter. The flow behavior can be predicted by taking some assumptions into consideration made while studying the flow in the channel are, flow in the channel is taken as the laminar flow. The shear stress acting across the top surface is taken zero since the force acting here is only the drag force, which is very negligible. The pressure acting at the top of the channel at the inlet is taken as 1 atmosphere.

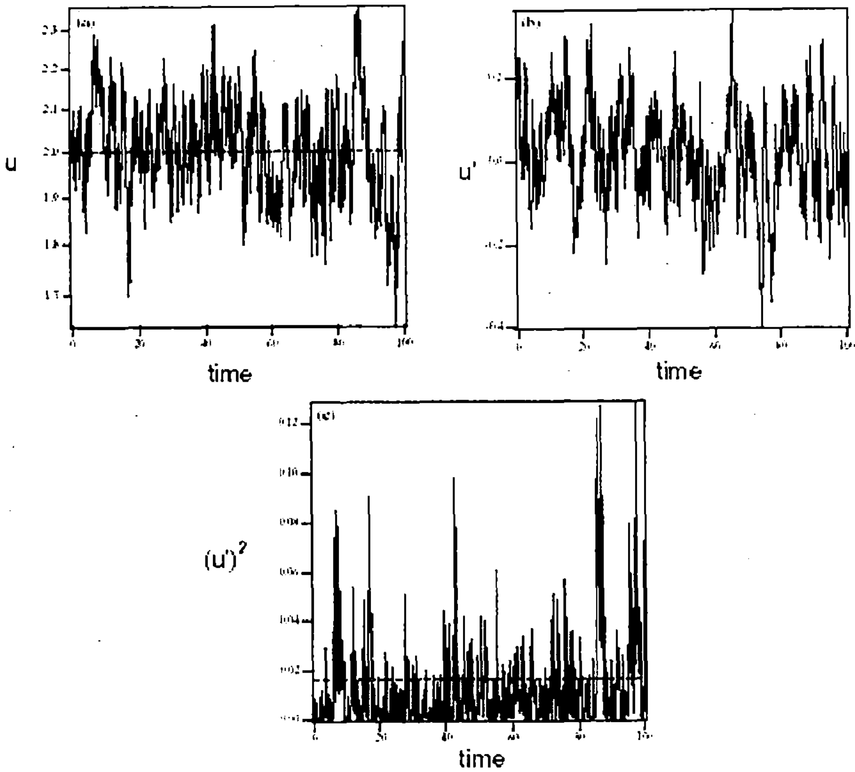

Fig.3.8 Velocity with respective to time [4]

ANALYSIS OF FLOW IN SHP POWER CHANNEL BY USING *FLUENT*

4.1 GENERAL

As discussed earlier *FLUENT* is a computational fluid dynamics (CFD) software package to simulate fluid flow problems [13]. It uses the finite-volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, in viscid or viscous, laminar or turbulent, etc. Geometry and grid generation is done using *GAMBIT* which is the preprocessor bundled with *FLUENT*. In order to apply CFD to analyze the flow behavior in a power channel proposed for Mohammadpur SHP plant has been consider as a case study.

4.2 SALIENT FEATURES OF PROPOSED POWER CHANNEL OF MOHAMMADPUR SMALL HYDRO PLANT

Mohammadpur Small hydropower station is chosen for the case study. It is located at 49.5 km from Mayapur (Haridwar) head works. The Fig.4.1 shows the X-section of the power channel proposed for renovation modernization and upgradtion (RMU) of Mohammadpur SHP station. The are following specifications

i.	Shape	:	Trapezoidal
ii.	Size	:	Bed width –45.0 m, water depth- 3.70m
iii.	Bed Slope	:	1 in 8800
iv.	Materials of Lining	:	P.C.C. M10/M20 R.C.C.
v.	Design Discharge	:	228 m^3/s
vi.	Free Board	:	1.50m

vii. Side Slop : 1.5 H : 1.0 V

viii. Length of power Channel : 480 m

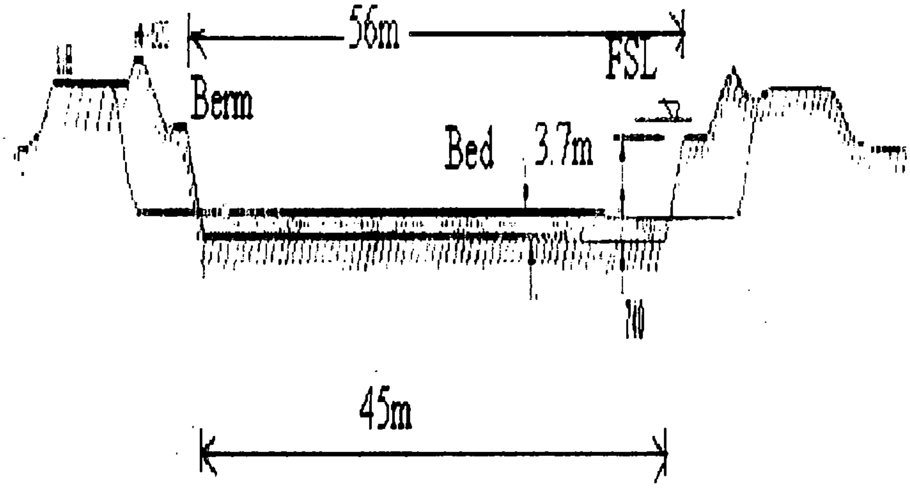

Fig.4.1 X- Section of the Power Channel [20]

4.3 STEPS INVOLVED IN FLOW ANALYSIS OF A PROPOSED POWER CHANNEL

Power channel is the component of the Small Hydropower Station, which is used to convey water from the intake to the Forebay tank. The flow in power channel is taken as the laminar flow. Various steps involved in solving as shown in Fig.4.1. Fluent and Gambit is user-friendly software environment, which are used to solve various problems, which occur in the common practice.

4.3.1 Creating Geometry in GAMBIT

For creating the Trapezoidal cross section of Power Channel, first created the vertices at the eight corners. Then join adjacent vertices by straight lines to form the "edges" of the Trapezoidal. Then create a "face" corresponding to the area enclosed by the edges. And from the volume form a "volume" from faces. The hierarchy of geometric objects in GAMBIT is

Vertices → Edges→ Faces→ Volumes

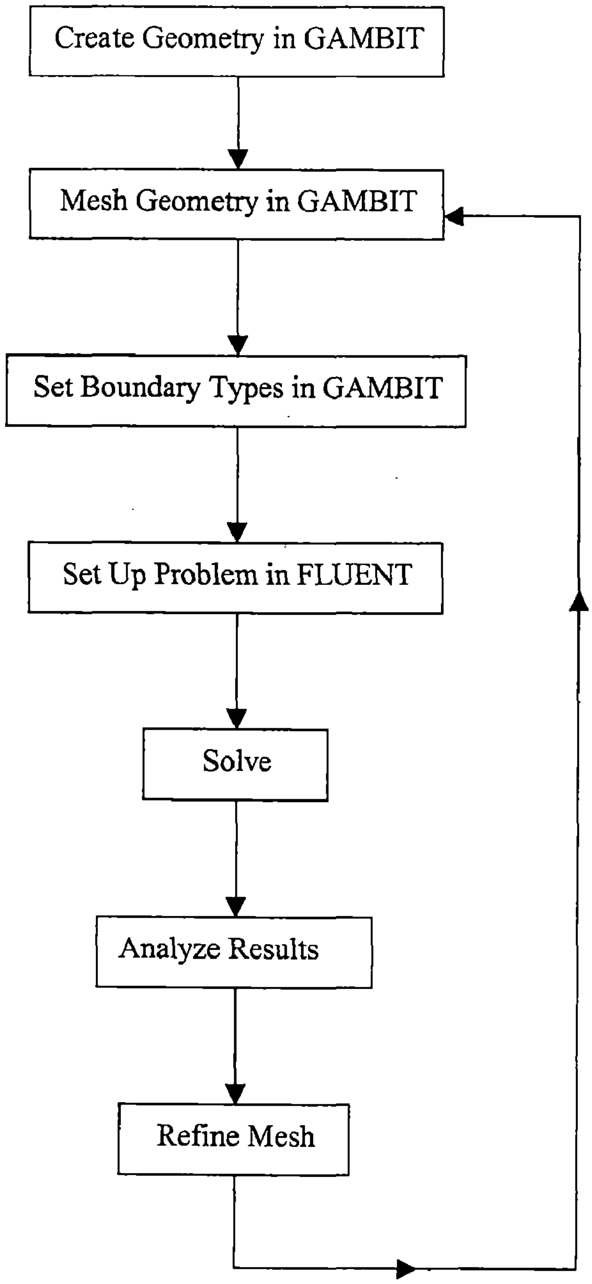

Fig.4.2 Steps Involved in the Flow Analysis

The GAMBIT Interface consists of the following

i. Main Menu Bar

In the main menu job name ntrappow.1 appears after ID in the title bar of the Main menu bar, which is the title of the present job. Fig. 4.3 shows the main menu Bar.

ii. Operation Tool Pad

The operation tool pad of the Gambit will be as shown in Fig.4.4; it contains various sub options in the main options. The options in the main button are the operation, geometry and vertex. When each of the buttons is selected, a different option in the Geometry sub-pad will be appeared. After selecting the option in the geometry sub-pad the option sub window appears, as shown Fig.4.4, which shows the coordinates of the vertices, will create the real vertex.

In the sub window coordinate system, type of the vertex is selected. The coordinates specified will be the global and the local coordinates as shown Fig.4.4 in the sub option given.

iii. Global Control Tool Pad

The global control tool pad has options such as fit to screen and undo that are used during the course of geometry and mesh creation. Apart from that it has the select pivot, presentation configuration, lighting options, orient representation and examine mesh options, which are used while working. It appears as shown Fig. 4.5.

iv. GAMBIT Graphics

This is the window where the graphical results of operations are displayed. Also called as the working window where the drawings are appeared. The Fig.4.6 shows the Gambit graphics window.

v. GAMBIT Description Panel

The description panel contains descriptions of buttons or objects that the mouse is pointing to. By moving mouse over buttons and notice the corresponding text in the description panel.

vi. GAMBIT Transcript Window

This is the window shows output from GAMBIT commands is written and which provides feedback on the actions taken by GAMBIT as it performs operations. The arrow button in the upper right hand corner to make the Transcript window full-sized. The arrow again to return the window to its original position size.

4.3.2 Mesh Geometry in GAMBIT

After creating the geometry a mesh on the Trapezoidal face grid spacing is specified through the edge mesh. The following are the sequences of the operations involves for obtaining the mesh edges.

Operation Tool pad → Mesh Command Button→ Edge Command Button → Mesh Edges

By shift+click or bring up the edge list window and select both the vertical lines. By clicking and dragging the mouse to specify an area to zoom in on, and releasing the Ctrl button. To return to the main view, clicking on the

Global Control Tool pad → Fit to Window Button again.

Once a vertical edge has been selected, select interval count from the drop down box that says interval size in the mesh edges windows. Repeat the same process for the horizontal edges, now that the edges are meshed which are ready to create a 3-D mesh for the face. The sequences of the operations involved in the Mesh Face are as shown below Operation Tool pad → Mesh Command Button → Face Command Button →Mesh Faces

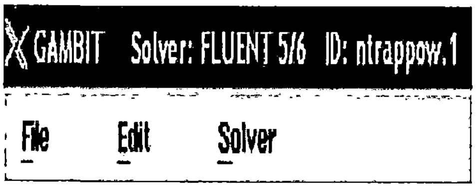

Fig. 4.3 Main Menu Bar [21]

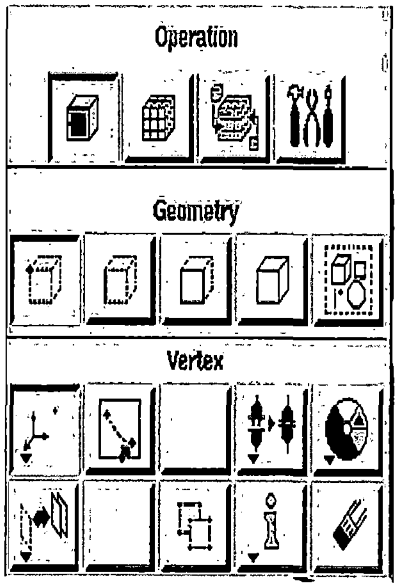

Fig.4.4 Operation Tool Pad [21]

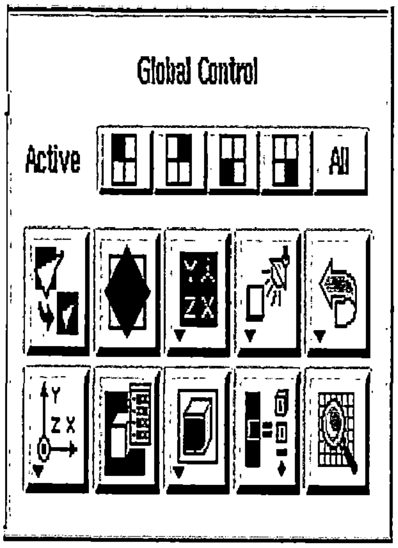

Fig.4.5 Global Control Tool Pad [21]

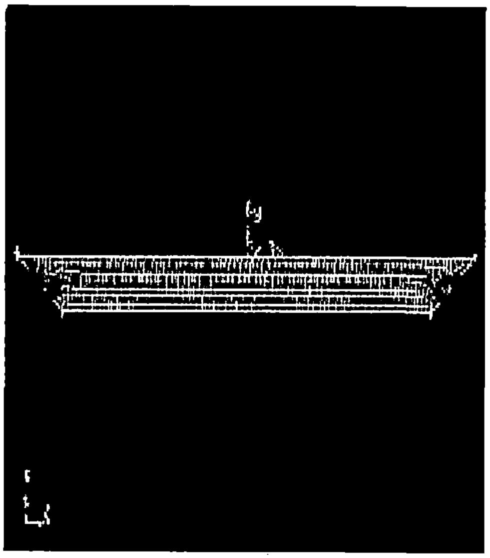

Fig.4.6 GAMBIT Graphics [21]

4.3.3 Specify Boundary Types in GAMBIT

The boundary types in the power channel are specified. The edge which is the rear side of the Fig.4.5 is the inlet of the flow, the edge which in the front side is outlet of the flow. The remaining all four sides is specified as the walls. The sequences of the operations involved are as shown below.

Operation Tool pad → Zones Command Button→ Specify Boundary Types →Command Button

This will bring the specific boundary types window on the operation panel. First specify that the edge is the inlet. Now selecting the edge by shift clicking on it. The selected edge appears in the yellow box next to the edges box we just worked. Next to name in the toolbox enter inlet or is the name required.

Type is selected by picking the type of the boundary condition across that edge. The new entry appears under name/type box near the top of the window. The all other boundary conditions for the power channel are specified.

4.3.4 Save and Export

After performs all work required in the Gambit it is saved in the following manner,

Main Menu → File → Save

The work that has to been in the Gambit has been finished and now the mesh generated mesh can be exported to Fluent where the further work can be done. This can be done as shown in the below,

Main Menu → File→ Export→ Mesh

4.3.5 Setting up problem in FLUENT

After generating mesh in Gambit to perform further operations done in Fluent. By launching Fluent 6.2 we are working in the 3D selecting the 3ddp from the list of options and clicking the option to run.

Start → Programs→ Fluent Inc → FLUENT 6.2

The 3ddp option is the 3-dimensional, double-precision solver. In the double-precision solver, each floating-point number is represented by using 64 bits in contrast to the single-precision solver, uses 32 bits. The extra bits increase not only for the precision but also range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.

4.3.6 Importing Grid

Navigating to the working directory and selecting meshed file. The mesh file was created using the preprocessor GAMBIT in the previous step. FLUENT reports the mesh statistics as it reads in the mesh. The grid drawn in the gambit opened in the Fluent by reading the case as follows,

Main Menu → File → Read→ Case

4.3.7 Checking and Grid Size

First, checking the grid to make sure that there are no errors while drawing the grid and meshing in the Gambit. The Fig.4.7 and Fig.4.8 shows the grid check and grid size window. This can be done by as shown below,

Main Menu → Grid → Check

Any errors in the grid would be reported at this time. Checking the output and making sure that there are no errors reported. Checking the grid size this can be done as follows,

Main Menu → Grid → Info → Size

4.3.8 Displaying the Grid

This function displays the Grid that is generated in Gambit, in different orientations. Graphics window opens and the grid is displayed in it.

By Closing the Grid Display menu to get back some desktop space. The graphics window will remain. The following are the sequences of the operations involved here

Main Menu → Display → Grid

4.3.9 Defining Solver Properties

The solvers properties are segregated solver, implicit formulation, steady flow, 3D, absolute velocity formulation and cell-based are chosen for the flow in the power channel. The snap of the window appeared as shown in Fig.4.9. The operations involved has be done as shown in the below,

Main Menu → Define → Models → Solver

Since here dealing with the Laminar flow is the default. The flow is other than the laminar then picking the nature of the flow in list mentioned in the side. The following series of has operation been done,

Main Menu → Define → Models → Viscous

The energy equation is decoupled from the continuity and momentum equations. It is not needed to solve the energy equation because not interested in determining the temperature distribution. So by leaving the energy equation unselected and click cancel to exit the menu.

Main Menu → Define → Models → Energy

4.3.10 Defining Material Properties

The material can be selected from the fluent database and we get the properties the material like Density, Viscosity and other parameters required. By clicking on the

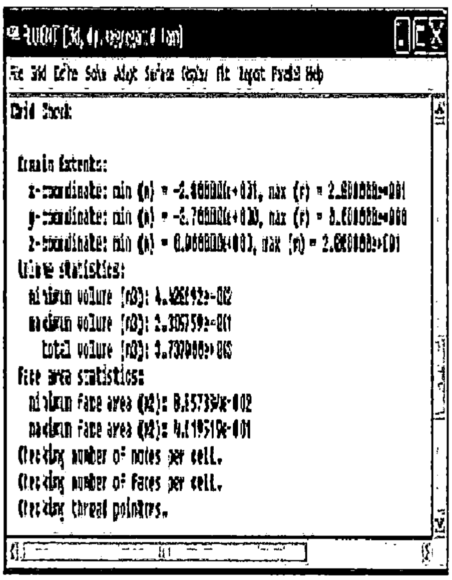

Fig.4.7 Grid Check

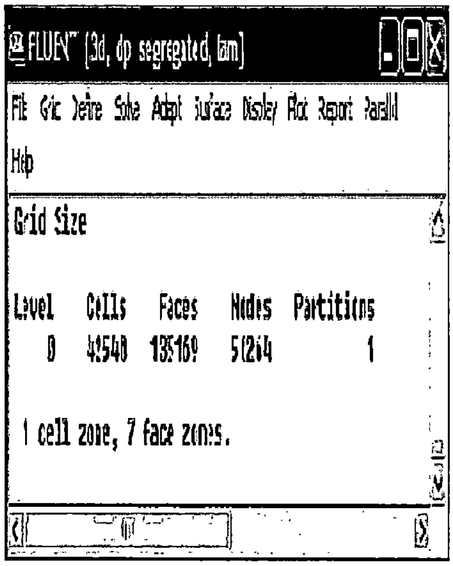

Fig. 4.8 Grid size

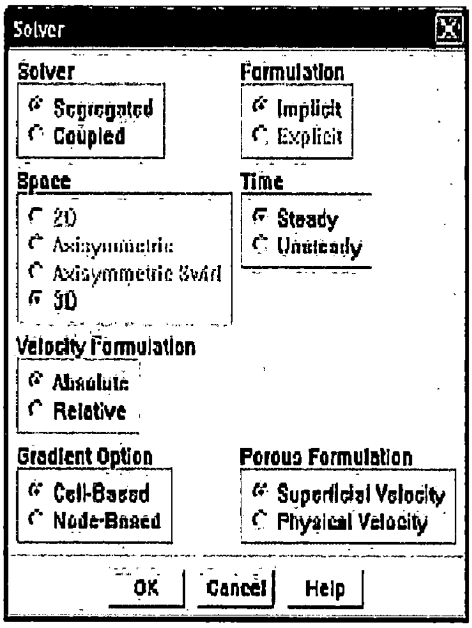

Fig.4.9 Solver Snap Shot

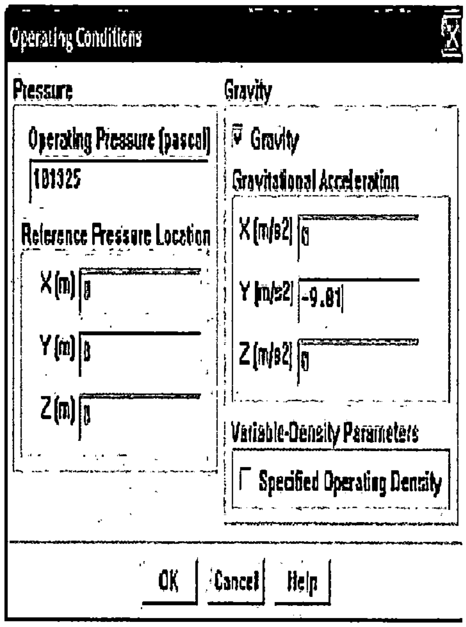

Fig.4.10 Snap Shot of Operation Conditions

Change/Create, we can adopt the newly selected Properties. This can be done as shown in the below,

Main Menu → Define → Materials

4.3.11 Defining Operating Conditions

FLUENT uses gauge pressure internally the default conditions has been set. The required change in the pressure can also be done by adding the operating pressure to the gauge pressure. The default value of 1 atmosphere (101,325 Pa) as the operating pressure is selected. The snap shot is as shown in the Fig.4.10.The sequences of the operations involved in the process are as show in the below,

Main Menu → Define → Operating Conditions

4.3.12 Defining Boundary Conditions

Boundaries were defined specified as zones of the chosen 3D shape on the left side of the boundary conditions window. The boundary conditions given in the Gambit can be verified; new conditions can also be initialized if required. Then type of boundary is selected corresponding to each side. Outflow is selected for the face.4 as shown in the Fig.4.11 and clicking set. Other boundary conditions are also mentioned in the same way as follows,

Main Menu → Define→ Boundary Conditions

Moving down the list and select inlet under zone. The FLUENT indicates that the Type of this boundary is velocity-inlet. The boundary type for the "inlet" was set in GAMBIT. If necessary, we can change the boundary type set previously in GAMBIT in this menu by selecting a different type from the list on the right. By performing the above-mentioned series of the operations the problem is specified in the gambit and successfully imported to Fluent the various conditions acting over the problem to obtain the required parameters acting over it.

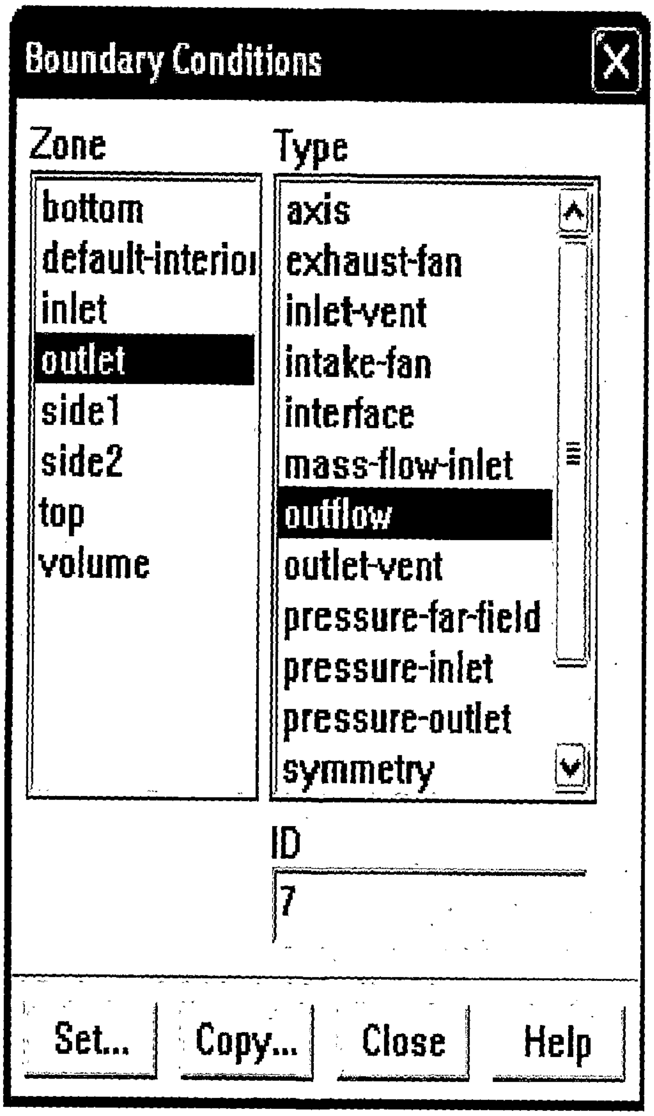

Fig.4.11 Snap Shot of Boundary Conditions

4.4 RESULTS AND DISCUSSION

The study across the proposed power channel is done in order to know the flow behavior and to visualize the flow. The flow along the length of power channel is similar to the segment of the channel is taken for the study. The dimensions of proposed power channel taken to study are the 45 m at the base, 56 m at the top, height of the 3.7m and length of 30m.

4.4.1 Generated Mesh

The mesh generated in the Gambit along with the boundary conditions was as shown in the Fig. 4.12. The features of the generated mesh is as follows,

i. The total number of nodes generated are 50264

ii. The number of the mixed wall faces in zone 3 is 5400

iii. The number of the mixed wall faces in zone 4 is 6720

iv. The number of the mixed wall faces in zone 5 is 780

v. The number of the mixed wall faces in zone 6 is 780

vi. The number of the mixed outflow faces is 709

vii. The number of mixed interior faces in zone 10 is 120071

viii. The number of the hexdrahedral cells in the zone 2 is 42540

Domain extents of the power channel are as flows,

X- coordinate; min (m) = -2.800000e+001, max (m) = 2.800000e+001

Y- coordinate; min (m) = -3.700000e+001, max (m) = 0.000000e+000

Z- coordinate; min (m) = 0.000000e+000, max (m) = 3.000000e+001

Face area Statistics of the generated mesh as shown in the below,

Minimum face area (m^2): 8.857384e-002

Maximum face area (m^2): 4.619519e-001

4.4.2 Residual Plot

The plots obtained by residual are shown in the Fig.4.13, which show the residual convergence of the continuity equation in the White line. The residual values show in the Y-axis and the number of the iterations on the X- axis. The residuals of the Velocity in the X, Y, and Z direction are as in the red, green and blue respectively. The solution is converged after performing 79 iterations.

The process of the drawing the residual plot is from the main menu picking the solve sub option and click then monitors which leads to another sub option residual. The convergence criterion of 10^{-6} is set in option available in the window and by starting the iteration operation. The sequences of the operations involved in obtaining the plot will be as shown in the following

Main menu → Solve → Monitors → Residuals

4.4.3 Velocity Plot

The critical parameter to monitor the flow in the open channel is the velocity which play's a vital role in the present analysis. It is obtained in the following way as shown below, in the main menu the display is selected the sub option of the vectors. In the vectors velocity option if picked. In the vectors the various parameters that can be monitored are like pressure, density and any other user-defined functions can also be calculated.

Main menu → Display → Vectors → Velocity

The velocity profile in trapezoidal power channel have been obtained as shown in Fig. 4.14. The velocity with different magnitudes are indicated with the different colours with red at the surface near the top of channel, which indicates the maximum magnitude of the 9.97e-01 and blue color with the velocity of 0.00e+00, which occurred at the interface of the water with the walls.

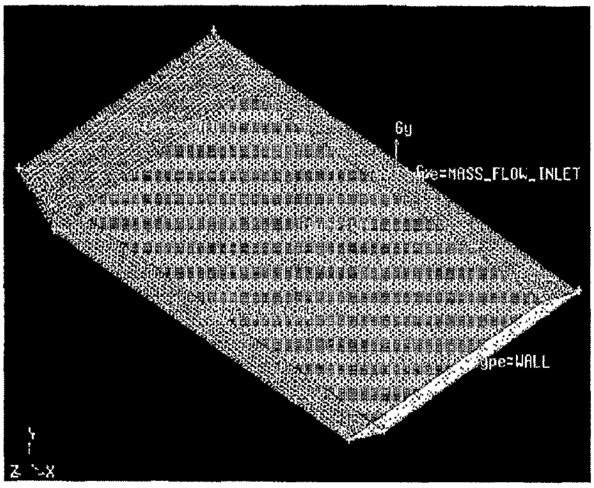

Fig. 4.12 Power channel with the Trapezoidal cross section

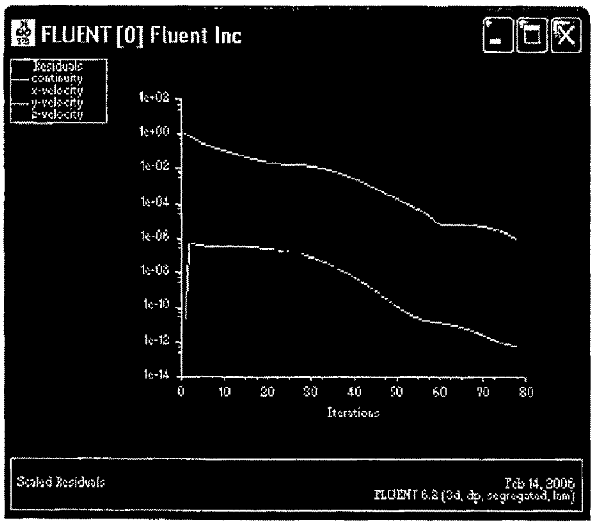

Fig. 4.13 Residual Plot

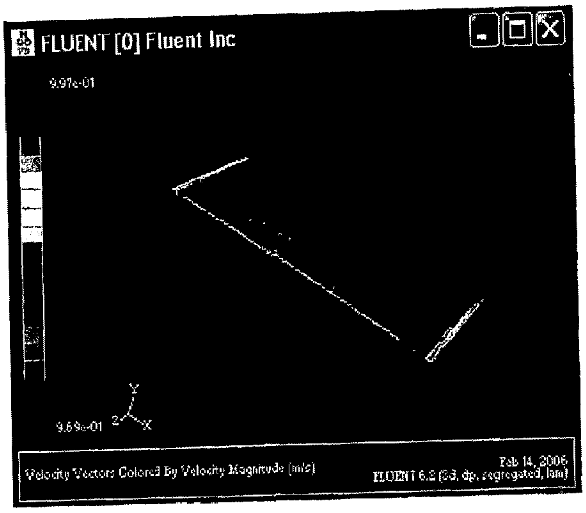

Fig. 4.14 Velocity profile across the power channel

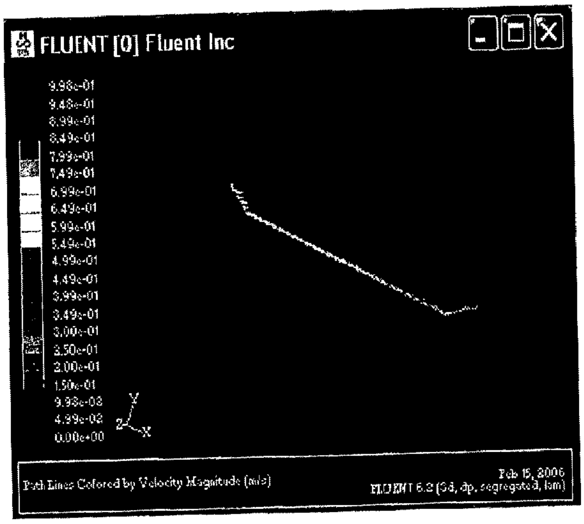

Fig.4.15 Velocity across inlet of the channel

The intermediate magnitude of the velocity are displayed by the orange, yellow, green with different shades as the corresponding magnitude displayed in the bar shown beside. The Fig.4.15 shows the velocity distribution across inlet the section of the trapezoidal power channel. In which the velocities at surfaces of the wall are zero and it gradually increases and reach the mean velocity trough out the section can be seen.

The velocity pattern obtained by velocity plot indicates that the iso-velocity plot show previously for the trapezoidal channel. The flow visualization across the given domain is done with respective to the velocity magnitude. The parameters obtained can be applied across the entire domain with fair degree of accuracy.

The assumptions made while studying the flow in the channel are, flow in the channel is taken as the laminar flow. The solver used to segregated solver, implicit formulation, steady flow, and 3D, absolute velocity formulation and cell-based are chosen for the flow in the power channel. The boundary conditions given here are the mass flowing across the section. The shear stress acting across the top surface is taken zero since the force acting here is only the drag force, which is very negligible. The pressure acting at the top of the channel at the inlet is taken as the 1 atmosphere. Due to no slip condition the flow velocity at the surface contact between the channel walls is zero, which is also visible in the results obtained.

In the similar manner CFD techniques can be applied to study flow across other components also to analysis the flow. It can be used to study flow in the zone of the flow reversal across the components.

CONCLUSIONS AND RECOMMENDATIONS

5.1 CONCLUSIONS

Flow analysis in SHP Station is critical in the initial stages of the design to know fluid behavior across various sections. By using numerical methods physical laws governing a fluid flow problem are represented by a system of partial differential equations including the continuity equation, the Navier-Stokes equations and any additional conservation equations. The numerical analysis resolves these equations by accurate and complex numerical schemes. For this work the commercially available Fluent 6.2 version along with Gambit 2.2 has been used. It uses the finite-volume method to solve the governing equations for a fluid.

The main conclusions drawn from the present study are

i. Application of the CFD techniques while design of the powerhouse structure leads to the cost effective design of the power plant. By avoiding the physical model studies which are necessary.

ii. A case study of the existing powerhouse Mohammadpur SHP Station has performed along the Power cannel the behavior of the flow.

iii. By using Fluent software, Velocity profile is calculated numerically and flow visualization along the power cannel.

iv. Color visualizations of flow, quantitative analysis, a view of the flow field and better understanding of the fluid mechanics in a power channel has been done. The red color indicates the maximum velocity in channel and the blue indicates the minimum velocity zero.

v. The velocity profile obtained here can apply along the cross section with the fair degree of the accuracy.

vi. The application of the CFD techniques has become an essential part of design, testing and optimizing process of the SHP station.

5.2 RECOMMENDATIONS

In the present dissertation work Flow analysis in SHP station analysis of flow is done by using the commercially available software Fluent 6.0 flow conditions in the power channel of a SHP station.

It is recommendation that future study can be extend to analysis the flow trough other components such as turbine, flow over the spillways etc.

v. The velocity profile obtained here can apply along the cross section with the fair degree of the accuracy.

vi. The application of the CFD techniques has become an essential part of design, testing and optimizing process of the SHP station.

5.2 RECOMMENDATIONS

In the present dissertation work Flow analysis in SHP station analysis of flow is done by using the commercially available software Fluent 6.0 flow conditions in the power channel of a SHP station.

It is recommendation that future study can be extend to analysis the flow trough other components such as turbine, flow over the spillways etc.

[1] Annual Report 2003-04, Ministry of Non-Conventional Energy Sources, Government of India.

[2] Small Hydro Power, "Initiatives and Private Sector Participation", Third edition by AHEC IITR.

[3] Efrem Teklemariam, Brian W. Korbaylo, Joe L. Groeneveld and David M. Fuchs "Computational Fluid Dynamics: Diverse Applications in Hydropower Projects Design and Analysis".

[4] Rajesh Bhaskaran, Lance Collins, " Introduction to CFD basics ", Cornell University

[5] I.Gunnar J Hellstrom, " Redesigning of an existing Hydropower Draft Tube", Master's Thesis Report Lulea University of Technology 2005.

[6] Suhas V. Patankar "Numerical Heat Transfer and Fluid flow", Hemisphere Publishing Corporation, 1992.

[7] Subhash C. Jain, "Open channel flow", John Wiley &sons, INC, 1999.

[8] John D.Anderson "Computational Fluid Dynamics the basics with applications", Mc Graw Hill International Editions, 1998.

[9] Layman's Guide Book, on "how to develop a small hydro site", European Small Hydropower Association (ESHA) 1998.

[10] Albert Ruprecht, "Unsteady Flow Analysis in Hydraulic Turbo machinery", Appeared in the 20th IAHR Symposium on Hydraulic Machinery and Cavitation, Charlotte, 2000.

[11] K.L.Kumar. " Engineering Fluid Mechanics", 1999, Eurasia Publishing House Ltd.

[12] ASME Hydropower technical committee, "The Guide to Hydro Power Mechanical Design " 1996, HCI publications.

dropower technical committee, "The Guide to Hydro Power Mechanical 1996, HCI publications.

adlo, Ronald Peikert and Etienne Parkinson, "Vorticity Based Flow Analysis Visualization for Pelton Turbine Design Optimization", Appeared in the Proceedings of IEEE Visualization 2004.

[15] Balint D, Susan-Resiga R, and Muntean S, "Numerical Simulation of 3D Flow in Kaplan Hydraulic Turbine". Appeared in the Proceedings of the International Conference Classics and Fashion in the Fluid Machinery, October 18-20,2002, Belgrade, YU –Serbia.

[16] Helena Ramos, "Guideline for the Design of the Small Hydro power plants", 2000.

[17] Fluent 6.2 User's guide, January 2005.

[18] Hirt, C.W., and Nichols, B.D. (1981). Volume of fluid (VOF) method for the Dynamics of free boundaries. J. Computational Physics, 39, 201–225.

[19] Vincent De Henau, "Turbine rehabilitation: CFD analysis of distributations", Water Power 95 proceedings of the International Conference on Hydropower.

[20] Mohammadpur Small Hydropower Station Tender Specification for renovation, Modernisation and Uprating Civil Works,Uttranchal Jal Vidyut Nigam Ltd, November 2004

[21] www.cfd-online.com

[22] www.fluent.com

[23] www.microhydropower.net

[24] www.canren.gc.ca