International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 07 | July - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

DESIGN AND ANALYSIS OF HYDRO TURBINES USING CFD

Mr. Pankaj[1], Mr. Aditya veer Gautam[2], Mr. Anurag kumar[3],
Ms. Shivangi Dixit[4], Mr. Rajneesh Kumar[5]
1Research Scholar, Mechanical Engineering, FET, Rama University, Kanpur
2,3,4,5, Assistant professor, FET, Rama University, Kanpur

---***---

Abstract - The aim of the present work is to carry out the free surface flow in a Pelton turbine bucket. Steady state numerical simulation has been done for the single bucket, for this the fixed configuration of Pelton turbine bucket is considered. The dimensions of the bucket are to be varied for the same conditions of discharge, head, and inclination of bucket to the jet. The numerical analysis is performed with the FLUENT code. The results provided by the analysis are compared for two profiles of Pelton buckets. The flow patterns in the buckets are analyzed from the results. An analysis of the losses due to the edge and the cutout of the bucket is also done. These results give the details as per expectation of steady flow calculations through the optimization process of the design of Pelton turbines.

1. INTRODUCTION

A large amount of energy is wasted throughout the world in river and streams flowing with variable range of head and discharge, as there are less effective ways to convert it to useful energy. Therefore, in the present century a great effort has been made to improve the existing technology for the harness of hydro potential in a cost effective ways.

In today's industrial world, power generation is based predominantly on oil, followed by natural gas and coal. Water power and nuclear energy contribute only small part of the total demand. Growth potential of technically feasible hydro power exists in the world today, mostly in countries where increased power supplies from clean and renewable sources are most urgently needed, in order to allow social and economic development [1].

Zangeneh. M [5] described a fully three dimensional compressible inverse design method for the design of radial and mixed flow turbo machines. In this method the circumferential distribution of averaged swirl velocity on the meridional geometry of the impeller is prescribed and corresponding blade shape is computed iteratively. The flow calculation is done through a designed high speed radial inflow turbine by using a three dimensional inviscid Euler solver.

Dritna P. and Sallaberger M. [6] showed the comparison of experimental data and 3D Euler and 3D Navier Stokes results for the flow in aturbine runner. They also highlighted the state of the art of predicting the performance of an entire Francis turbine by means of numerical simulation.

Hana. M. et.al [7] used the classical grid based CFD approaches using a free surface tracking method. The calculations were performed on a fixed grid with a moving jet atthe inlet.

Kvicinsky et.al [8] compared the calculated pressure distribution to experimental data. For this, the objective of his work is to perform RANS modeling of steady bucket compared to global and local measurements for a significant range of functioning parameters.
Mack and Moser [9] described the sliding mesh technique for the connection of two regions.
Agarwal. A.K.et.al [10] used the discrete distribution of particles, spherical pellets, or strips to discritize the water sheet for three dimensional rotating buckets.

Zoppe.B [11] shows the detailed experimental and numerical analysis of the flow in a fixed bucket of a Pelton turbine. The numerical analysis is performed with the FLUENT code. The experimental analysis provides measurement of pressure and torque as well as flow visualization. A detailed analysis of torque and thrust is done for the evaluating of losses due to the edge and cutout of the bucket.

Alexandre Perrig et.al [12] describes the investigation of free surface flow on a pelton turbine model bucket. The unsteady numerical simulations are performed with wall pressure measurement and flow visualization. The five distinct zones of bucket is taken into consideration for the torque measurement. The back side of bucket is also studied.

The turbine is considered one of the most critical components of small hydro power plant. The hydro turbines for each SHP are designed according to the site head and discharge data, which may not come under the standard size available for the turbine in the market. It is therefore the performance analysis of turbine is essential which predict the characteristics of the prototype. The present work relates the study of hydro turbine and its analysis. A study at Pelton turbine is done for Bassi hydro- power plant and flow analysis of bucket is done. The results for the flow are compared between two standard bucket profiles.

2. DESIGNING METHODOLOGY FOR PELTON TURBINE

2.1 Buckets design

The Pelton bucket shape is cup splitted at the centre and this can be made by having the concept that two cups are joined to make a common edge. It also has a notch which is cut at the tip of the bucket. The dimensions of the bucket are determined according to the diameter of the cross –section of the jet. [Nechleba]. For the analysis purpose in the present work, the two cross –section of the buckets are taken which are shown in the figure 1 &2.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 07 | July - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

The shape of the bucket is approximately defined by the following dimension-

Length of the Bucket	= 2.5 -2.8 times of jet diameter
Width of the Bucket	= 2.8 -4.0 times of jet diameter
Depth of the Bucket	= 0.95 times of jet diameter

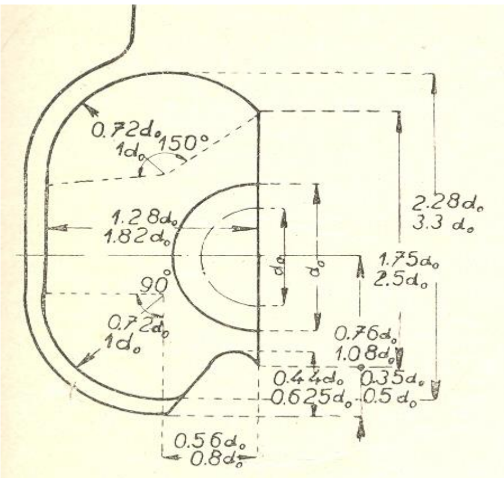


Fig 1: Bucket Profile 1 [13]

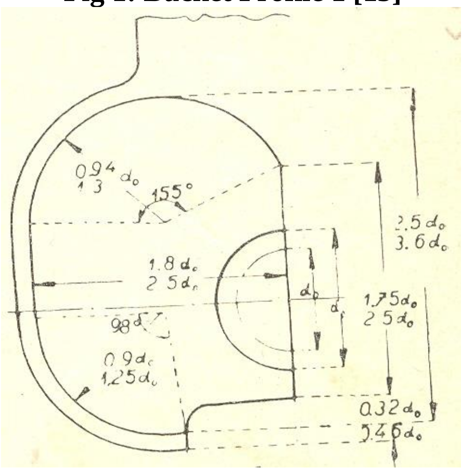
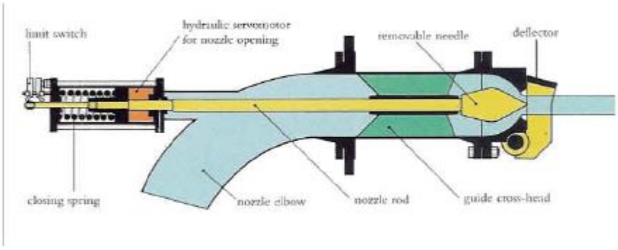


Fig 2: Bucket Profile 2 [13]

2.2 Nozzle and needle

The shape and compactness of the jet depends on the shape of the needle and of the throat. The shape of the needles and nozzles is conical shape because it converges the jet more rapidly without being disturbed by the tip and the needle is subjected less to the corrosion. The needles and nozzles with angles $45^0/60^0$ and $55^0/80^0$ are used. Apart from this deflector, casing for the turbine etc. are the other components of the Pelton turbine. The profile for the nozzle and needle are shown in fig 3. [13]

Fig 3: Cross section of Nozzle and Needle [4]



3. FORMULAS FOR THE PELTON BUCKET

The proposed capacity of the prototype is 15 MW under a net head is 335.5 m. Various parameters to design the bucket are described and presented below:

a) Diameter of jet

The diameter of jet is calculated by the equation given below as:

$$do = \sqrt{\frac{4}{\pi}\frac{Q}{\phi\sqrt{2gH}}}$$

Where ϕ represents the efficiency of nozzle and amounts to 0-95 to 0-98

(b) Jet Velocity

It is given by the equation

$$Co = \phi\sqrt{2gH}$$

(c) Diameter of the number

In order to fined out the runner diameter, it is to be considered the jet velocity for a given head and assuming the suitable runner speed.

(d) Specific speed of number

It is calculated by the equation given below as

$$Nsp = \frac{N\sqrt{P}}{H^{5/4}}$$

Where, P is the power developed by the turbine in kW.

4. VELOCITY DIAGRAM

The work of the water particle within the wheel is not uniform and depends upon the condition and place of their contact with the blade. The velocity diagram draws for the extreme points as shown in fig 4 for the points I to IV.

The angle δ, enclosed by the specific velocity C_o and the peripheral velocity, equals zero for point I and also for the points III and VII which lie on the radius passing through point I. For all other points this angle must be derived.

The outlet velocity triangle is drawn if the outlet angle α2 is considered and the value of peripheral velocity at the outlet is known. The peripheral velocity at the inlet and at the outlet is of the same magnitude only for very few water particles. This

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 07 | July - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

depends not only on the position of the inlet points but also on the position of water particles in the free jet. The jet widens when impinge towards the runner blade, and thus water particles which enters inside the wheel under the same condition assume various path and discharge in various places of the wheel at different peripheral velocities.

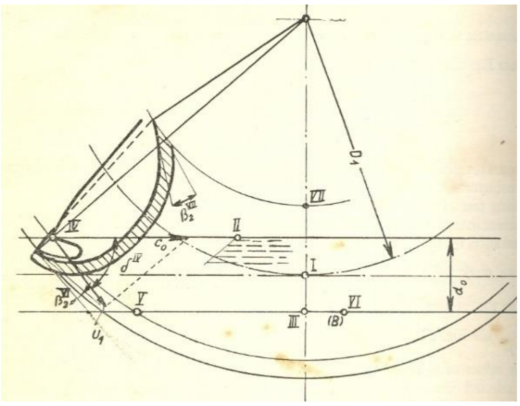


Fig 4: Position of Bucket at Different Points

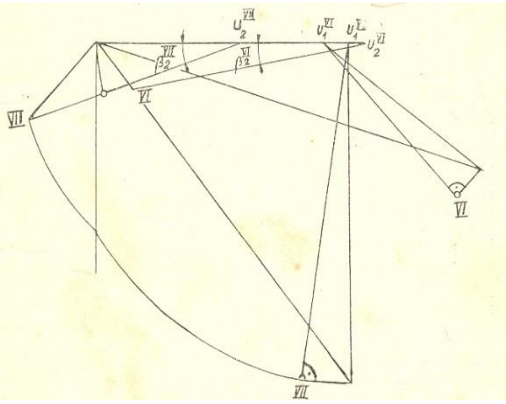


Fig 5: Velocity Diagram [13]

The velocity diagram is shown in fig 5. But if we considered upon the influence of water particles and attempt to determine the path of a single mass point on the blade, the difficulties arises.

The deflection created by an oblique position of the blade and to draw the path of the particle on the blade. The direction of angle β_2 for the outlet point I we then considered the outlet velocity C_2 or its meridional component equal a value in the range from 0.12 to 0.2. This loss a discharge of 1.5 to 4%.

When the inlet and outlet peripheral velocities are different, the value of w_2 is determined by the equation:

$$W_2^2 = W_1^2 - U_1^2 + U_2^2$$

4.1 Determination of the bucket numbers

The number of buckets for the Pelton turbine is selected on the basic of the diameter of the jet in relation to the radial length of the blade. If the large diameter of the jet is taken and the shorter radial length of the bucket, then the number of buckets will increase. The selection of the number of buckets is taken from the following table:

d_0/D	1/6	1/8	1/10	1/15	1/20	1/25
z	17-21	18-22	19-22	22-27	24-30	36-33

4.2 Determination of inlet angle

The angle β_s is given by the constructional value $l_0$and S_0 given by the equation

$$\tan\beta_s = \frac{S_0}{2l_0}$$

The angle β_s' is given by

$$\tan\beta_s' = \frac{S_0}{2l_0'}$$

These parameters are shown in the figure clearly.
If the ratio d_0/D is greater, then the deflection of the direction of on flowing stream becomes perpendicular. In designing the bucket, the magnitude of the outlet angle β_2 which is smaller than the actual mean value of the outlet angle β_2^+, it is because the flattened stream converses at the discharge from the bucket under the angle δ and its central jet proceeds under the angle β_2^+ positive which is greater . β_2

Determination of outlet edge the angle: the outlet edge of the cup is advantages to be at exactly 180^0 to the incoming jet. However, in practice this angle is slightly deviated by an angle 10 to 15^0. By doing so we can close some of the energy but this the operational requirement of the turbine. This deviation in the outlet edge avoid the hitting of outgoing jet to the back of the incoming bucket which is exactly in front of the bucket at which jet is striking

5. SIMILARITYCONSIDERATION OF MODEL AND PROTOTYPE TURBINE

In this concept the results of performance available can be applied to know or to determine the performance of prototype. The mathematics applied is called the geometrical analysis by considering the low of similarity. According to these laws of similarity. The results for performance of a tested machine can be applied or can be considered similar for prototype provided the machines are geometrically similar, kinematically similar and dynamically similar.

5.2 Geometrical similarity

It implies that the model must be similar to the prototype turbine i.e. all the linear dimensions of two machines are compared are proportional and angle between the corresponding elements are equal.

The geometrical similarity indicates,

$$\left(\frac{D_p}{D}\right)_m = \left(\frac{D_p}{D}\right)_p$$

Where,
m stands for model
P stands for prototype

5.3 Kinematic similarity

The similitude of flow fields on the passage of model and prototype. The velocities, acceleration etc at the corresponding points in flow must be proportional and in the same direction. Since velocity and acceleration are vector quantities, hence not only the ratio of the magnitude of these quantities remains the same, but the direction of velocity and acceleration at the corresponding points in the model and prototype also should be parallel.

The kinematic similarity indicates

$$\left(\frac{C_0}{U}\right)_m = \left(\frac{C_0}{U}\right)_p$$

For Pelton turbine it is modified as given below:

$$\left(\frac{Q}{ND^3}\right)_m = \left(\frac{Q}{ND^3}\right)_p$$

5.4 Dynamic similarity

The proportionality of process pressure etc. acting on the streamlined elements of turbine passage is to be considered. The similarity of forces between the model and prototype. Also the direction of forces at the corresponding points should be same.

The dynamic similarity indicates.

$$\left(\frac{H}{D^2N^2}\right)_m = \left(\frac{H}{D^2N^2}\right)_p$$

To discuss the performance of hydro turbines based on model testing the similarity law are considered.

6. DEFINING THE MATHEMATICAL MODEL

6.1 Designing of pelton turbine bucket

In the present study, the designing and analysis of the turbine is analyzed. Therefore, a site was being selected at Bassi fall power project of Himachal Pradesh. The site has the proposed type at turbine is Pelton-turbine. The designing for the bucket of this turbine is done.

6.2 Data for the design

The technical data for the Pelton – turbine is listed below:

Table 1: Design parameters of Pelton turbine bucket

Sl. No.	Parameter	
1	Design net head	335.5 m
2	Rated output	15 MW
3	Seed	500 rpm.
4	Runner diameter	1422.0 mm
5	Number of buckets	18
6	Nozzle exit diameter	187.3 mm

6.3 Design of pelton–bucket

The correlation available [13] for designing the Pelton-bucket profile, has been done. Firstly the various essential parameters are calculated.

The, the profile of the bucket is drawn by calculating the various points dimensions of the bucket have been calculated and presented in table 3.

Table 2: Design parameters for proposed Pelton turbine

Sl. No.	Parameters	Unit	Value	Remark
1	Diameters of runner ,D	m	1.82	$\sqrt{\frac{4Q}{\pi co}}$
2	Pitch circle diameter , Dp	m	2.72	
3	Jet velocity co	m/s	80	Co = $\phi\sqrt{2gH}$
4	Velocity of wheel, U	m/s	38.1	U = $Ku\sqrt{2gH}$
5	Jet ratio, m		6.0	m= D/d_o
6	No. of buckets on runner, Z		18	For $\frac{d_o}{D}=\frac{1}{6}to\frac{1}{8}$ = 17 to 21, 18 to 22
7	Specific speed, Nsp		34	

6.4 Bucket profile

Bucket section so obtained is drawn in correct position show in fig 4 & 5 shows the wetted area of the bucket. It is drawn by plotting the corresponding pointes listed in table 3 &4.
In routine design the dimension of the buckets is determined according to the diameter of the cross section of the jet diameter. The shape of the bucket is approximately defined by the following dimensions:

Bucket 1 dimensions:

Length of the bucket, $3.2d_o$ = 0.96 m
Width of the bucket, $3.4d_o$ = 1.02 m
Depth of the bucket, $0.95d_o$ = 0.285 m

Bucket 2 dimensions:
Length of the bucket, $3.4d_o$ = 1.0 m
Width of the bucket, 4.6do = 1.38 m
Depth of the bucket, $0.95d_o$ = 0.285 m

Sl. No.	X (m)	Y (m)
1	0.225	0.48
2	0.5	0.2
3	0.42	0.6
4	0.42	0.45
5	0.42	0.20
6	0.225	0.20
7	0.1	0.345
8	0.215	0.36

Table 3: Points for Bucket 1

Table 4: Points for Bucket 2

Sl No.	X (m)	Y (m)
1	0.69	0.15
2	0.3	0.51
3	0	0.375
4	0.33	0.15
5	0.6	0.375
6	0.2	0.51

7. RESULTS ANALYSIS

A profile for analysis purpose is made to predict the performance of hydro-turbine prototype for the rated condition. The half –cup of the Pelton bucket is made for the analysis purpose and the angle of incidence of jet is considered for the centre of the bucket at 90^0. The analysis of the turbine model is required to predict the performance of a prototype. The hydro dynamic behavior of the flow is taken as a basic for the acceptance of turbine components profile with some of the manufacturing and material constraints.

Both single precision (2d) and double precision (2dp) version of FLUENT are available on all computer platforms. For most cases, the single precision solver will be sufficiently accurate. The following steps are involved in the modeling and analysis of the problem:

7.1 STEPS FOR MODELING

Step 1. Creating Geometry in GAMBIT

In order to create the profile we will first create the vertices of the Pelton turbine bucket profile is summarized We will then join adjacent vertices by straight lines and arcs to form the "edges" of these points. Lastly we will create a "face" corresponding to the area enclosed by the edges.

The geometry description is that, the splitter (right hand side of bucket) middle point is considered as the origin (0, 0) and the left hand side of bucket is at negative x-axis. With the help of various points described in table 4 the contour of the half cup of the bucket is drawn.

Step 2. Mesh Geometry in GAMBIT

For the meshing of the geometry firstly the edges to be meshed and then the faces. When a edge has been selected, the selected interval count from the drop down box that says interval size in mesh edge window. Then, the required numbers of nodes are to be appeared. Repeat the procedure for different edges. Now, we are ready to create a 2D mesh for the faces. The completed mesh geometry of buckets is shown in fig 6 & 7 along with the boundary conditions.

Step 3. Specifying the Boundry Types in GAMBIT

Now, the next set the boundry types in GAMBIT. The semi-circular portion shown in fig 4 & 5 is the inlet for the flow. The boundary conditions for the geometry are according to the following table:

Table 5: Boundary conditions

Edges Position	Name	Type
Centred	Inlet	VELOCITY_INLET
Left	Outlet	PRESSURE_OUTLET
Right-Corner Edges	Wall	WALL

The file is now in .msh and exported to 2d mesh. Since this is a two dimensional mesh .

Step 4: Set up Problem in FLUENT:

The FLUENT 6.2 version is launched. The "2d" option is used to select the two dimensional problem. Then, import the grid of. mesh file. Check the number of nodes, faces and cells. Check the grid for errors. Any error in the grid would be reported at this time is noted. Check the output and make sure that there are no error is reported. Check the grid size. The detail of the grid for the two Pelton turbine bucket is shown in table 6 & 7

Table 6: Mesh information for bucket 1

CELLS	FACES	NODES
4578	9306	4729

In this 1 cell, 5 face zones.

Table 7: Mesh information for bucket 2

CELLS	FACES	NODES
7939	16053	8115

In this 1 cell, 4 face-zones.

Step 5: In FLUENT the following procedure is adopted for the analysis purposes:

Discretization –scheme: The power law discretization scheme is used. This scheme interpolates the face value of a variable φ, using the exact-solution to a one dimensional convection diffusion equation.

Pressure velocity coupling: FLUENT provides the option to choose among four pressure-velocity coupling algorithms: SIMPLE, SIMPLEC, PISO and Fractional Step (FSM).

The SIMPLE algorithm is used. It is a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field. It is used because the high grid-skew ness was there in the problem. The κ-epsilon solver is used.

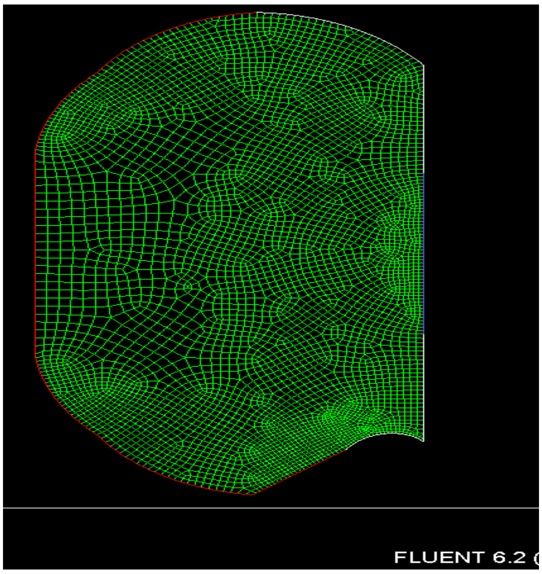


Fig 6: Modeled Profile for Bucket 1

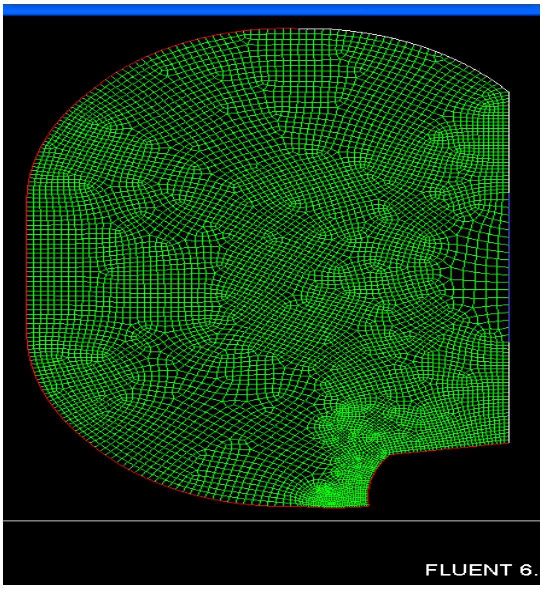


Fig. 7: Modeled Profile for Bucket 2

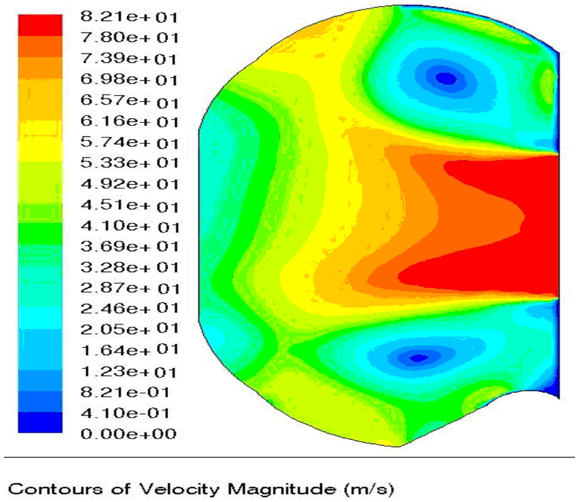


Fig 8: Velocity contour for bucket 1

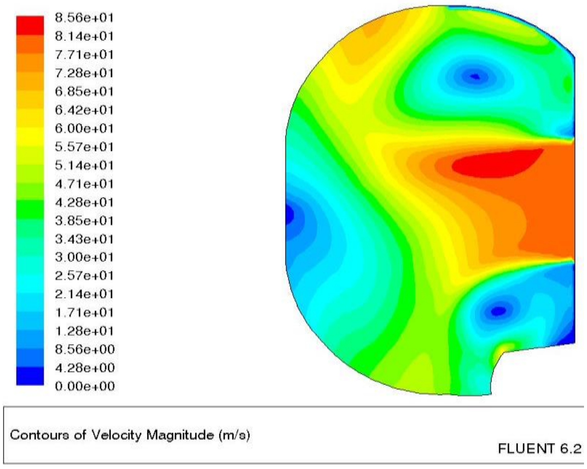


Fig 9: Velocity contour for bucket 2

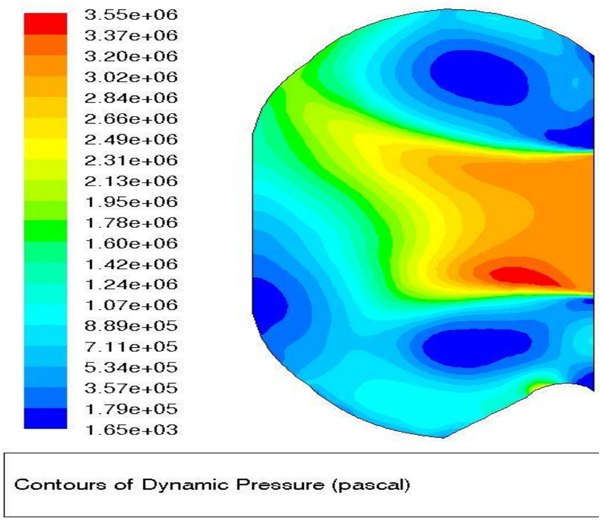


Fig 10: Dynamic pressure contour for bucket 1

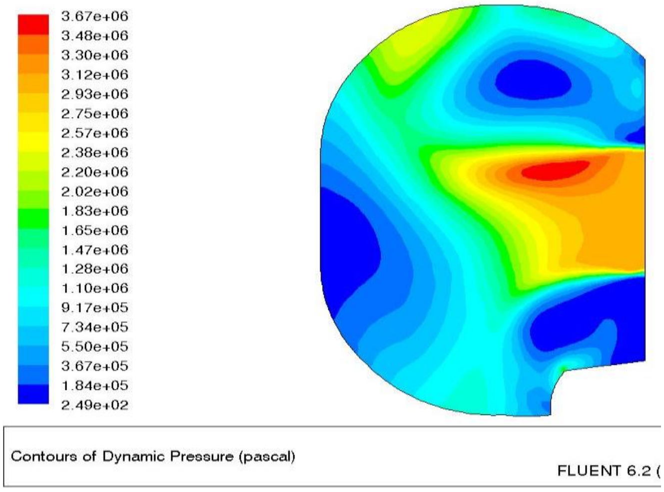


Fig 11: Dynamic pressure contour for bucket 2

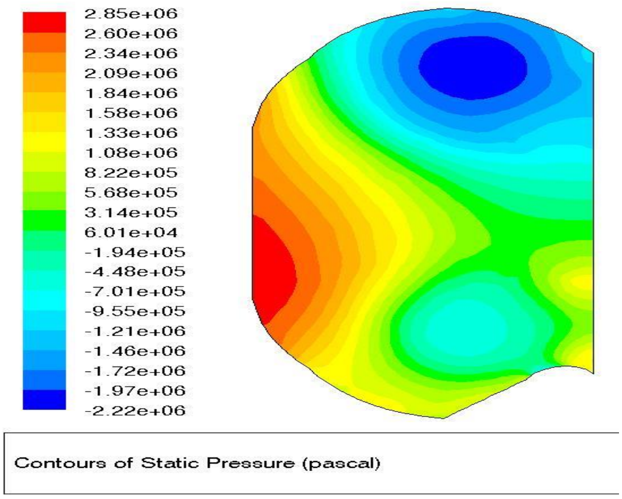


Fig 12: Static pressure contour for bucket 1Fig 13: Static pressure contour for bucket 2

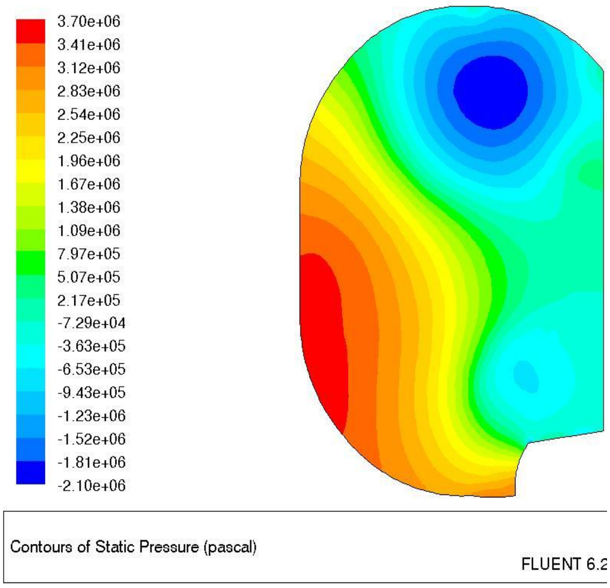


Fig 13: Static pressure contour for bucket 2

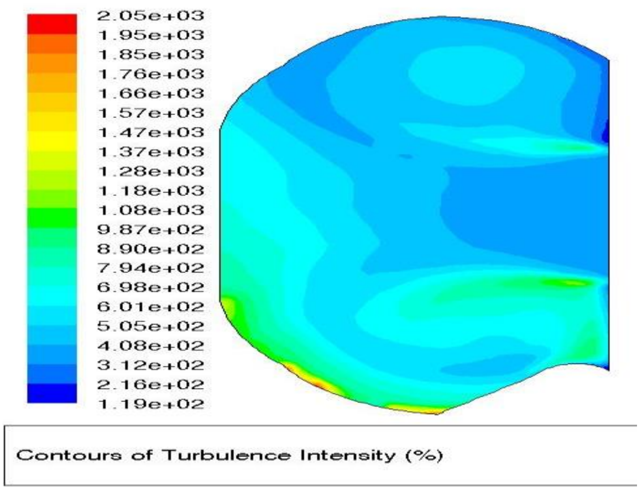
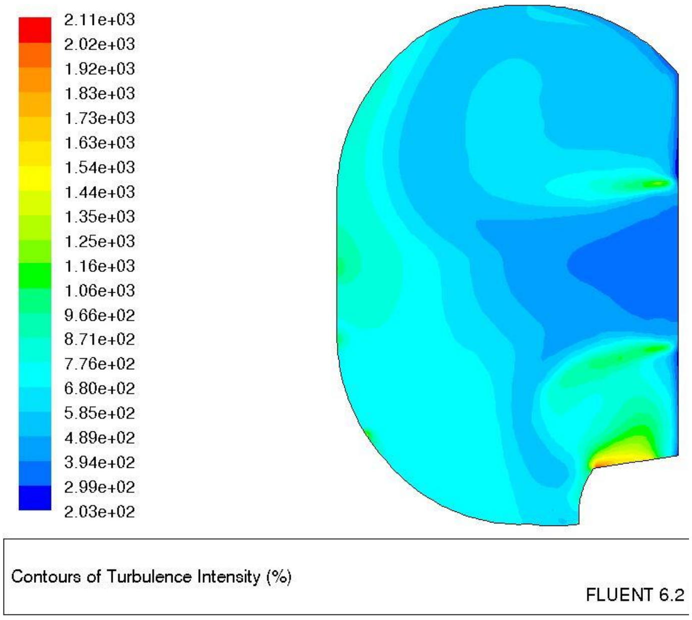


Fig 14: Turbulent intensity contour for bucket 1Fig



15 : Turbulent intensity contour for bucket 2

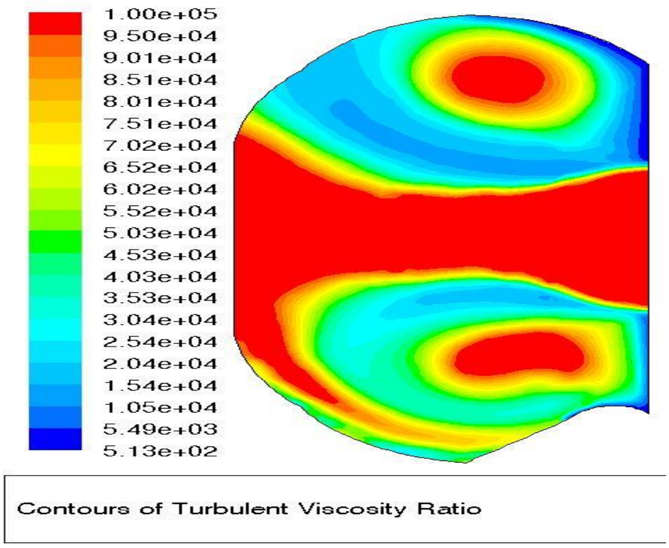


Fig 16: Turbulent Viscosity ratio contour for bucket 1

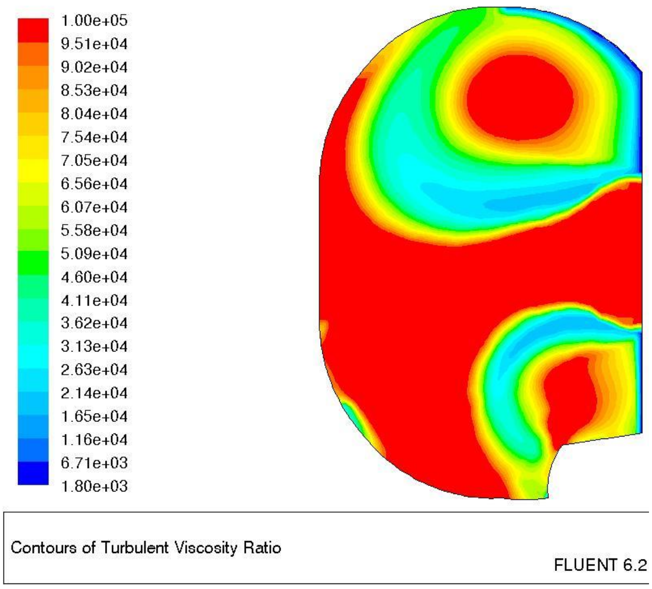


Fig 17: Turbulent Viscosity ratio contour for bucket 2

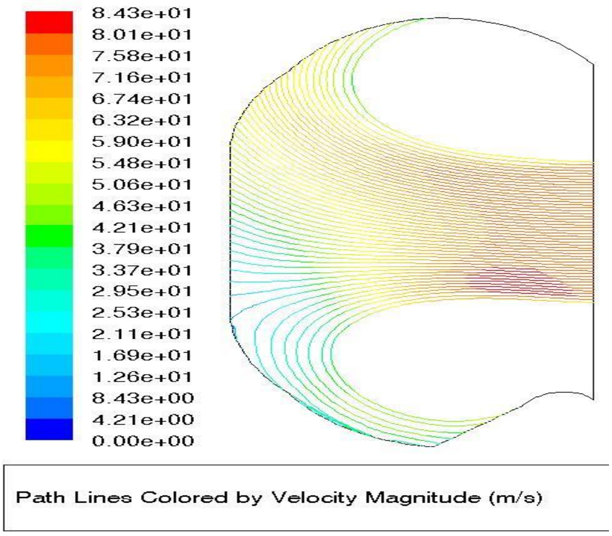


Fig 20 Path lines for bucket 1

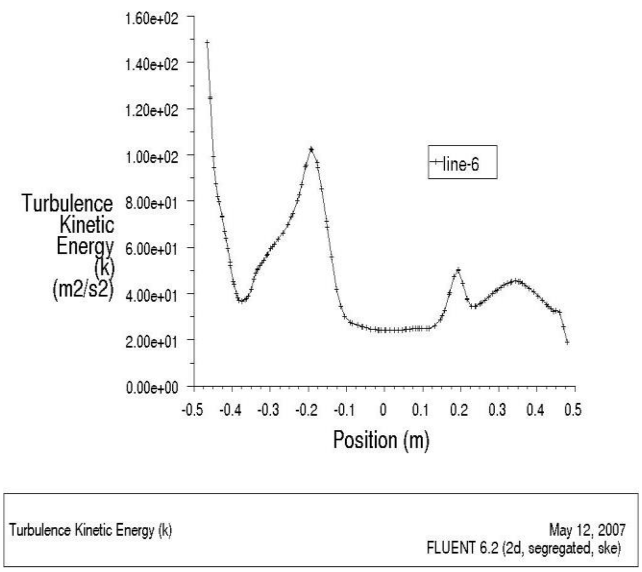


Fig 18: Turbulent K.E for bucket 1

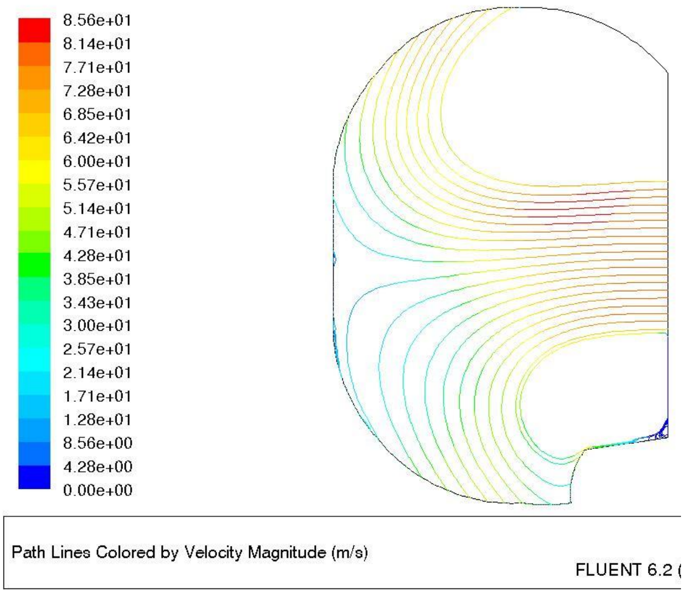


Fig 21 : Path lines for bucket

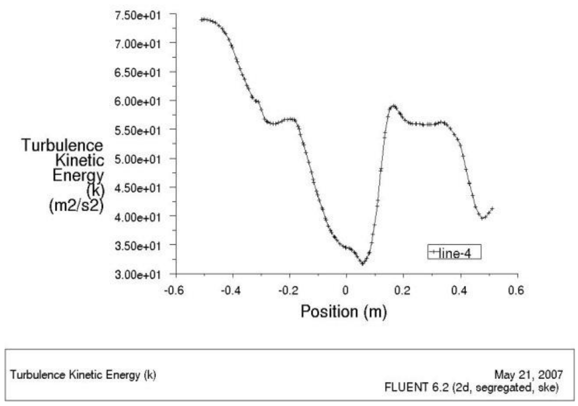


Fig 19 Turbulent K.E for bucket 2

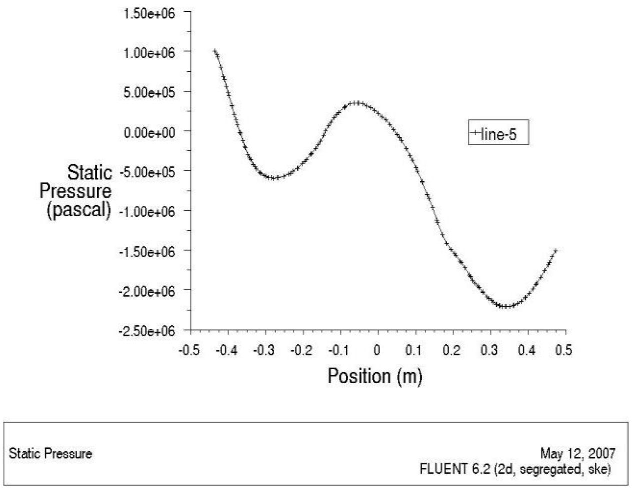


Fig 22: Static pressure plot for bucket 1

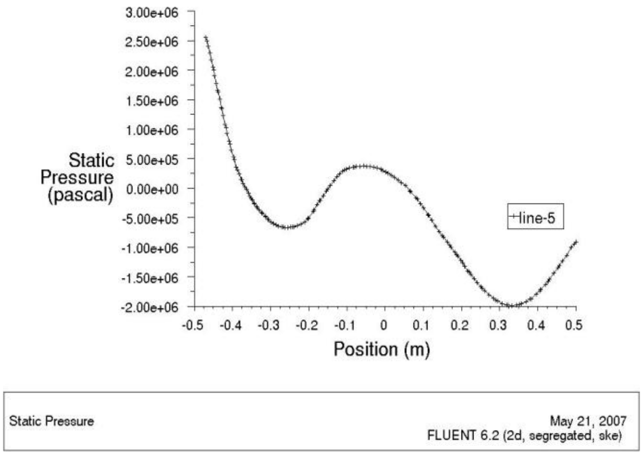


Fig 23 : Static pressure plot for bucket 2

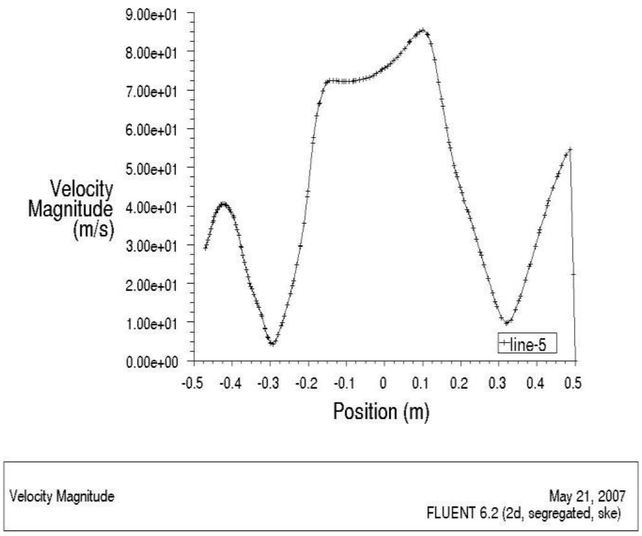


Fig 24 : Velocity plot for bucket 1

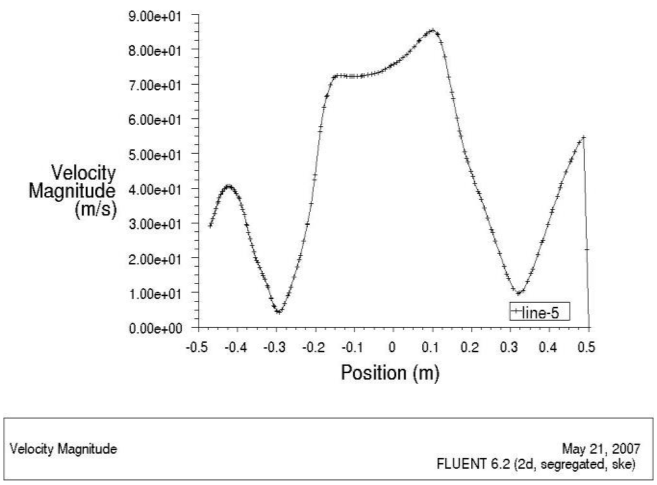


Fig 25: Velocity plot for bucket 2

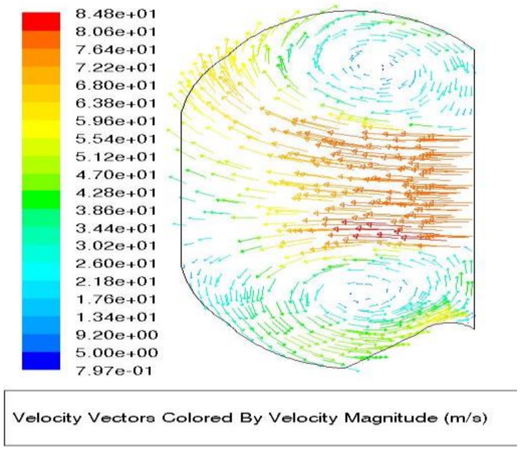


Fig 26 : Velocity Vector for bucket 1

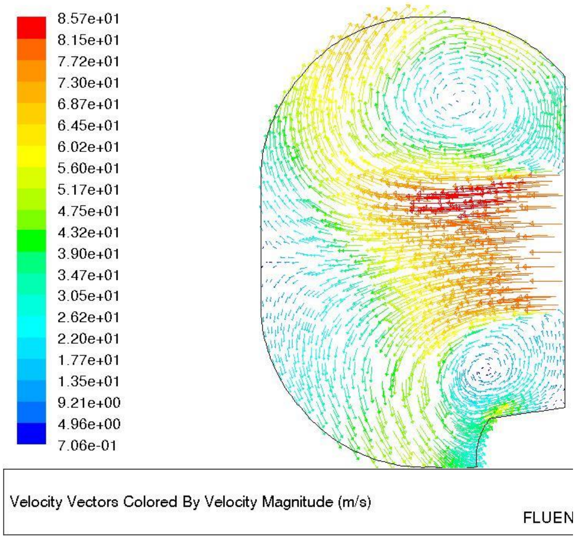


Fig 27: Velocity Vector for bucket 2

The usual convergence criteria to the Reynolds stress residuals, normalized residuals in the range of 10^{-3} which indicate a practically converged solution. However, the tighter convergence criteria are below 10^{-4} to ensure full convergence.

As the numerical values of the flow equations are plotted in this curve. The values should approach to zero. When comparing the relative merits of two or more different algorithms for a time marching solution to the steady state, the magnitude of the residuals and their rate of decay are often used as figures of merit. The fast decay of the residuals to the smallest value is usually looked upon most-favorable.

The fig 8 & 9 shows the contours of velocity magnitude. The middle portion of the bucket is shown, which is subjected to the large impact of jet. Hence, this is the region where the maximum energy of jet is transformed.

It can be observed from these figures. The fig 8 there is more velocity at the outlet of the bucket in comparison to the fig 9 which indicates the small size bucket is more efficient. It is because the more velocity is leaving from the outlet of the bucket in fixed configuration which indicates the more force is transmitted to the bucket, hence more power is delivered to the bucket. Although, the vortex formation takes place in bucket 1, near splitter and flow separation also occurs at the splitter. In the bucket 2, the formation of vortices takes place at the outlet, bottom side of the splitter and upper right corner of the bucket.

The fig 10& 11 shows the contours of dynamic pressure. As the dynamic pressure contours are the indication of velocity-contours. The behavior of these contours shows that the low pressure-developed at the upper & lower part of right hand side of the bucket and at the outlet edge. Although, the bucket is considered for uniform pressure but sometimes cavitation may occurs. The large pressure- gradient is seen at the lower-middle part of the right hand side of the buckets, which indicates the region where shock waves can be presented on the surface.

The fig 12 & 13 shows the contours of static pressure. The contour of static pressure shows the vortex formation at the upper part of the buckets and thus the flow becomes turbulent. In both the buckets the static pressure contour is not too different.

The fig 14 & 15 shows the contours of turbulent intensity. Turbulence is taken in to account using the κ-epsilon model with wall functions. On the jet inlet face the κ values are expressed according to the mean characteristics of the flow i.e turbulent intensity. The turbulence intensity is taken equal to 5%. The maximum turbulence intensity is shown at the outlet edge at bucket 1 and at the notch of the bucket 2. This is because of the non- uniform curvature for the bucket and the sharp corner presence. However, the large- gradients are not shown by both the buckets.

The fig 16 & 17 shows the contours of turbulent viscosity ratio. For the main stream line the high turbulent viscosity is found for both the buckets. For the other stream lines the viscosity is the dominating factor. But the turbulent effect becomes more where the vortices are formed. In bucket 2, the vortex formation at the bottom part is enlarged by the notch profile than the bucket 1.

The fig 18 & 19 shows the turbulence kinetic energy. In this figure the abscissa indicates the vertical position of the bucket for the maximum height of the bucket which is 0.5 and 0.6m above and below of the origin taken for these geometries and ordinate indicates the turbulent kinetic- energy. The horizontal position is taken at 0.3w for bucket 1 from the origin and for bucket 2 it is 0.35w from the origin.It can be seen that the turbulence kinetic energy at the lower end of the bucket is maximum for the buckets. At the upper part of the bucket i.e at a distance of 0.5 and 0.6m for bucket 1 and bucket 2 the turbulent K.E is very less compared to the -0.5 to -0.6 m position of the respective buckets.

The fig 20 & 21 shows the path line by velocity magnitude. As the path is the indication for flow- existence. In the fig 20 the path lines are well defined but in the fig 21 the path lines at the outlet edge becomes wider. At the upper and lower part of the bucket no path lines are shown because there is the vortex formation. The flow is intersecting here. Along the path lines the flow is irrational.

The fig 22 & 23 shows the static pressure where the abscissa denotes the vertical position of the bucket and ordinate indicates the static pressure. Both the buckets show almost the similar behavior. It is to be noted that the maximum pressure is developed at the lower edge of the bucket i.e at position -0.5 and -0.6m for the bucket 1 and bucket 2 respectively. It may be because of the presence of notch.

The fig 24 & 25 shows the velocity magnitude at different position of the bucket. It can be seen that the velocity-graph is same for the two buckets. Again here the maximum vertical position of bucket is defined at the abscissa. The lowest velocity indicates the position of vortices formation in the bucket. The highest velocity is seen at the central position of the bucket, which is corresponds to the maximum transfer of the energy to the bucket.

The fig 26 & 27 shows the velocity-vectors. Inside the bucket, the wetted surface increases with the jet diameter. The less water leakage flow through the cutout is noted for the bucket 1, than the bucket 2. In bucket 2 a more number of velocity-vectors are passing away from the cutout, thus indicating the loss of discharge. The velocity at the outlet edge of bucket 1 is more in comparison to the bucket 2, indicated by the velocity vectors.

8. CONCLUSION

The turbine is the basic component of hydro power plant as the last of both the civil structure and electrical equipments depends on turbine selection. Therefore, the characteristic of turbine prototype based on numerical analysis is required to establish a data base for hydro –turbine performance. The main conclusions drawn from the present study are given below:

i. The Pelton-turbine bucket is modeled on GAMBIT software for a prototype installed at Bassi hydro power plant having site conditions as, head 335.5 m, discharge 5.36 cumecs and power output of 15 MW. The analysis is done on FLUENT software.
ii. The numerical study of the flow inside a Pelton turbine bucket under fixed configuration is carried out. For that, two standard profile of buckets have been studied for normal impingement of the jet to the bucket surface under similar head and discharge conditions. The pressure distribution on the bucket and jet trajectory inside the bucket has been analyzed.
iii. In both the cases, a leakage of flow through the notch is found. The more leakage is found in bigger size bucket compared to smaller size one. The outlet velocity is more through smaller size bucket than that bigger size bucket which results in better energy transfer in case of the smaller bucket.

RECOMMENDATION

- Present work is limited upto the 2D fixed configuration of the Pelton turbine bucket. So, 3D analysis can be done with rotating buckets.
- The efficiency and other performance parameters can be analyzed by using FLUENT and other software packages.