CFD ANALYSIS OF PELTON BUCKET

[1]Mr.Lijo Sebastian
[1] Asst. Manager in Designs – FAYAT Group, AMIE mech.

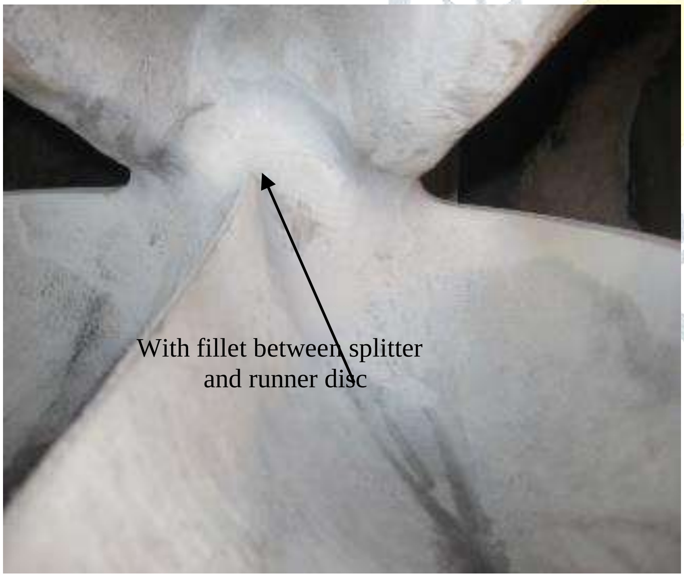
***Abstract:** This paper presents Computational Fluid Dynamics (CFD) analysis of Pelton turbine of Koyna Hydropower in Satara Gujarat. The purpose of CFD analysis is to determine torque generated by the turbine and pressure distributions in bucket for further work on fatigue analysis. The paper describes the methods used for CFD analysis using ANSYS CFX software. Consequently, the present study has also the ambition to reduce the size of the runner to have a cost effective runner design. The CFD analysis is carried out on model size Pelton runner reduced at 1:3.5 scale to minimize computational cost and time. The operating conditions for model size runner are selected in accordance with IEC 60193 and IEC 1116. The pressure distribution is found maximum at bucket tip and runner Pitch Circle Diameter (PCD). 3 buckets are used to predict the flow behavior of complete Pelton turbine. K-ε and SST turbulence model with interphase transfer method as free surface and mixture model is compared in the paper. The torque generated by the middle bucket is replicated over time to determine total torque generated by Pelton turbine. The case study described in this paper provides an example of how the size of turbine can affect the efficiency of the turbine. These were discussed in detail which helps in understanding of the underlying fluid dynamic design problem as an aid for improving the efficiency and lowering the manufacturing cost for future study.*

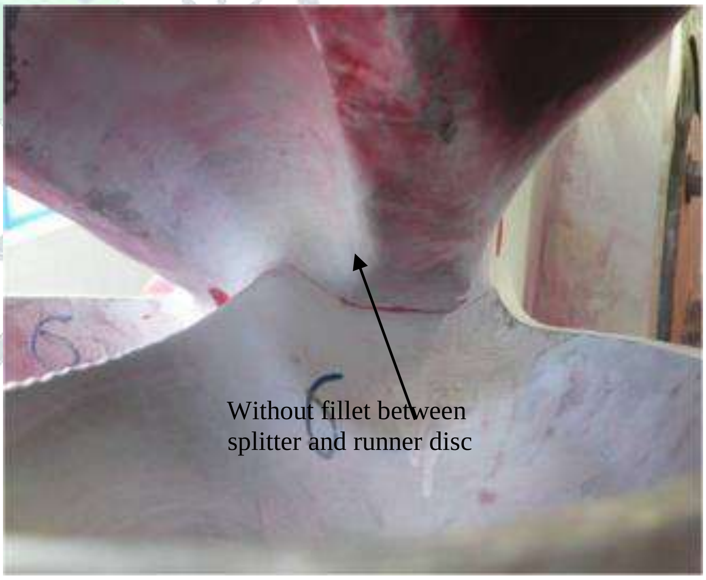
***Index Terms** - CFD analysis using ANSYS CFX, cost effective runner design, improving the efficiency…*

I. INTRODUCTION

Computational Fluid Dynamics (CFD) is a branch of fluid dynamics which uses numerical methods and algorithms to solve fluid flow problems. Reduction of time and cost to predict the model behavior in real environment is key advantage of CFD analysis. However, the CFD analysis results should be analyzed and validated before the model is accepted [1]. ANSYS CFX and ANSYS Fluent are the commercial CFD codes available. The main difference between these is the way solvers integrate the flow equations and solution strategies. CFX uses finite volume elements to discretize the domain. Contrarily, Fluent utilizes finite volumes. They are both control volume based solvers, which ensures conservation of flow quantities. The CFD analysis of Pelton turbine in the paper is carried out using ANSYS CFX [2].

The Pelton turbine is a good choice in situation where the volume flow is small relatively to head. The paper describes the CFD analysis of scaled Pelton turbine of Koyna Hydropower in Gujarat. The purpose of CFD analysis of model Pelton turbine is to determine the pressure distribution in the Pelton bucket which shall be used for fatigue analysis of Pelton runner with two different bucket geometry, with and without fillet at root section, as shown in figure 1. The design geometry of bucket was changed during welding repair and runner failure occurred after 10000 hours of operation [3]. In addition, the torque generated by the turbine is determined in the paper.





II. METHODOLOGY

The methodology used for CFD analysis of Pelton turbine is shown in Figure 2. The 3D model of Pelton turbine was created with reference to Koyna Hydropower in Satara Gujarat. The rotating domain and stationary domain for CFD analysis was modeled using Catia V5 and ANSYS Design Modeler respectively. The model size runner was selected considering laboratory test facility and IEC 60193 test requirements. The numerical methods & boundary conditions were defined in ANSYS CFX & the numerical results were computationally and analytically validated. The torque was calculated and computationally valid pressure distribution was exported for fatigue analysis.

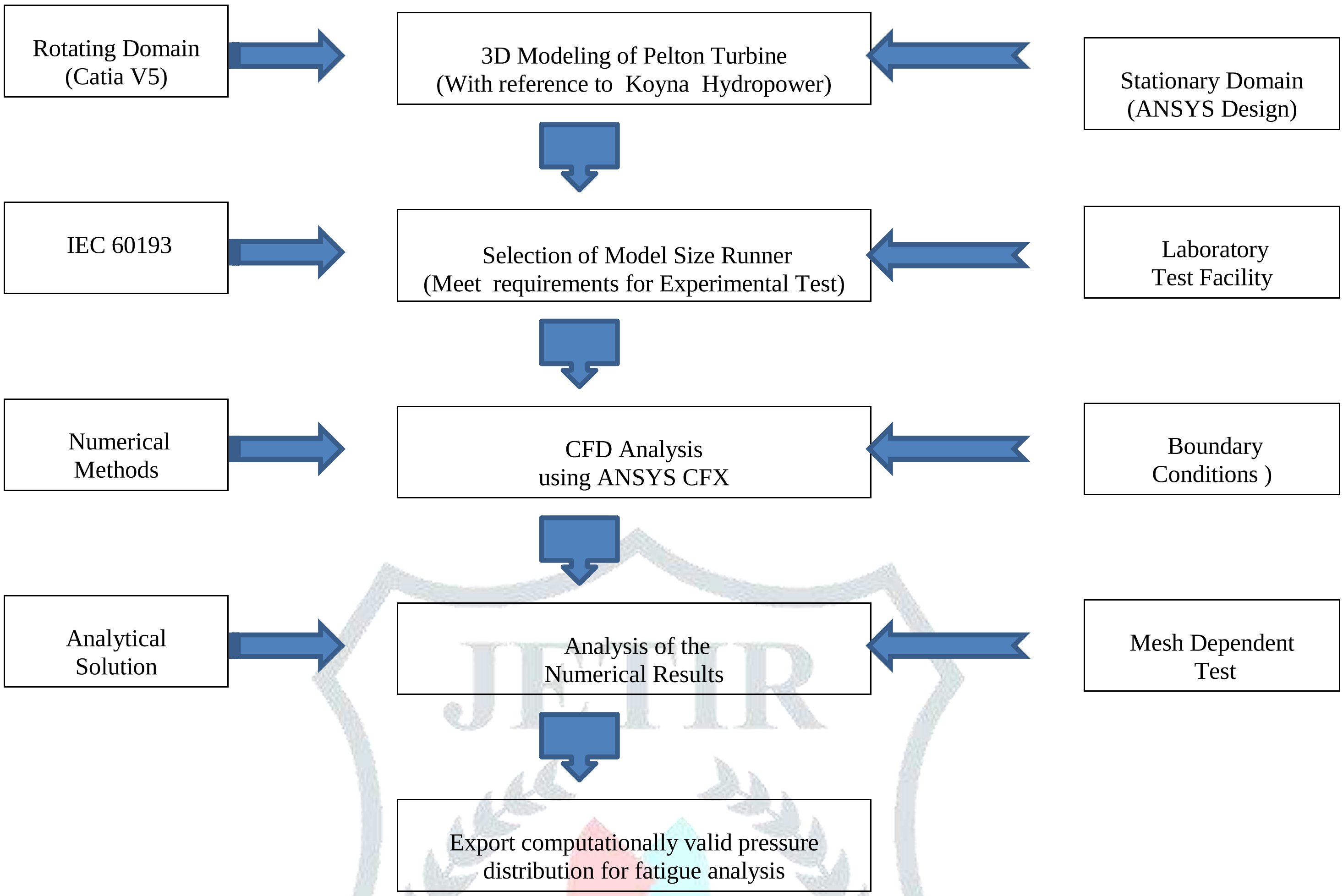
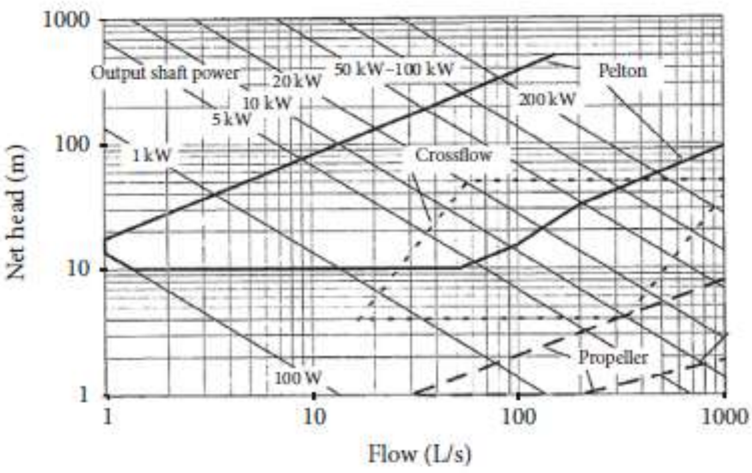


Figure 2. Methodology for CFD analysis

III. 3D MODELING OF PELTON TURBINE

2D contour plot of Pelton bucket was used to model 3D Pelton runner for CFD analysis. Each contour line was exported using AutoCAD and imported in Catia V5 software to build 3D model, shown in Figure 3. Figure 4 shows the selected Pelton bucket for CFD analysis. Half bucket was selected for analysis since the Pelton bucket is symmetric about the splitter. This reduces the total time for computational analysis [4].



Application ranges for different types of turbine

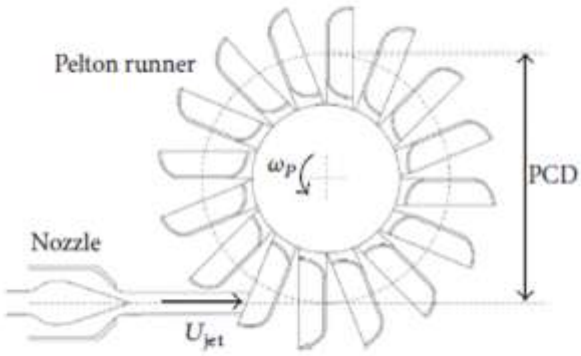


Diagram of a Pelton runner showing PCD

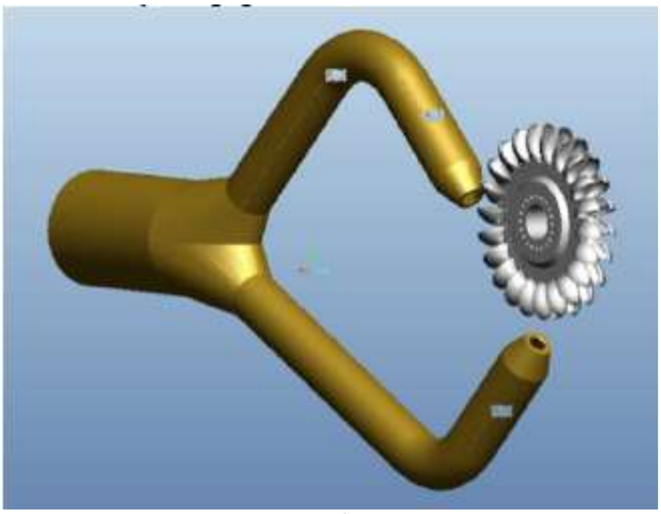


Figure 3. 3D model of Pelton Turbine

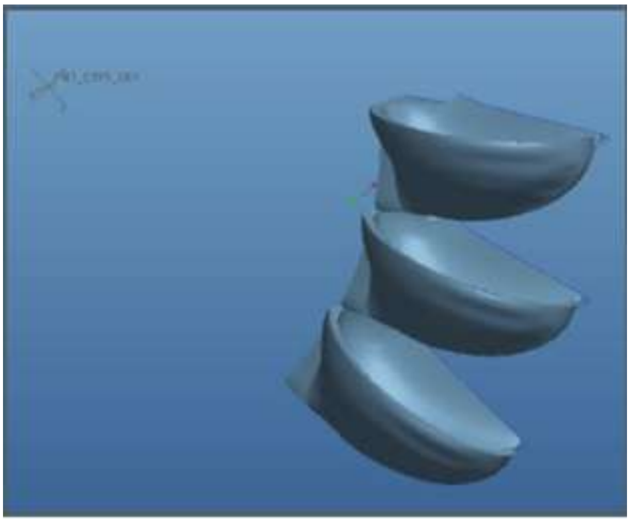


Figure 4. Selected Bucket for CFD Analysis

IV. SELECTION OF MODEL SIZE RUNNER

Scaled runner has been selected to reduce the computational cost and considering the future prospects of verifying the CFD result at Turbine Testing Lab, Gujarat University. The laboratory test facility and minimum requirements for model test of hydraulic turbines mentioned in IEC 60193 and IEC 1116 has been used to obtain hydraulic similitude conditions between the model and prototype. The turbine data for prototype is shown in Table 1.

A kinematically similar turbine is obtained when the model and prototype are geometrically similar and ratio of their fluid velocity and peripheral velocity is equal. A complete similarity is achieved when the Reynolds number is equal between the model runner and prototype in addition to kinematic similarity. These conditions are satisfied when the speed number is equal in both the turbines [5]. Therefore, equations 1 – 3 are used to determine the hydraulic similitude conditions [6]...

$$Speed\ Number\ (\Omega) = \omega\sqrt{Q} \qquad Equation\ 1$$

$$Speed\ Facctor\ (n) = \frac{(n \times D)}{\sqrt{E}} \qquad Equation\ 2$$

$$Flow\ Factor\ (\Omega ed) = Q/(D^2 \times \sqrt{E}) \qquad Equation\ 3$$

Besides, minimum requirements for model Pelton turbine as stated in IEC 60193 and hydraulic similitude conditions, shown in Table 2, and the laboratory test facility available at Turbine Testing Lab, shown in table 3, are used to determine the appropriate operating conditions for model Pelton turbine, shown in Table 4 [6] [7].

Table 1; Turbine Data: Prototype

Parameter	Unit	Value
Pitch Circle Diameter (PCD)	mm	1400
Number of Buckets	-	22
Number of Nozzles	-	2
Head (H)	m	660
Discharge (Q)	m3/S	2.15
Rotational Speed	RPM	600

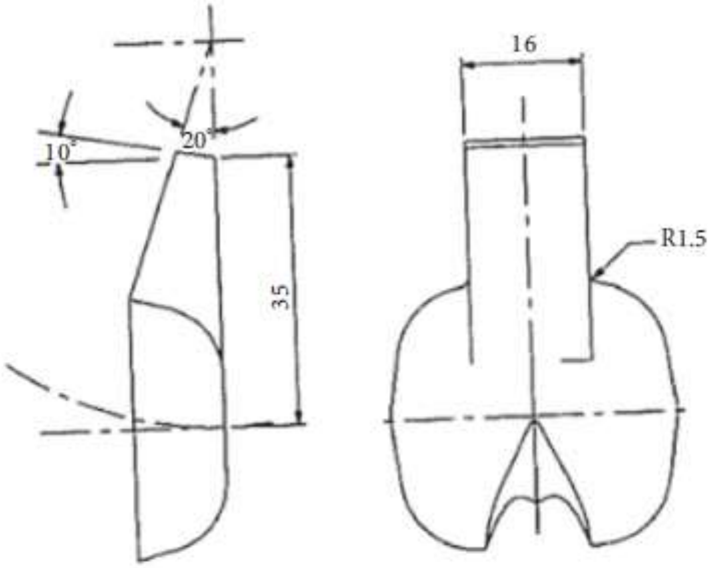
Table 2; Hydraulic Similitude Conditions

Parameter	Unit	Value
Speed Number	-	0.076
Speed Factor	-	10.44
Flow Factor	-	0.014
Minimum Reynold’s Number	-	2x10^6
Minimum Hydraulic Specific Energy	J/Kg	500
Minimum Bucket Width	mm	80

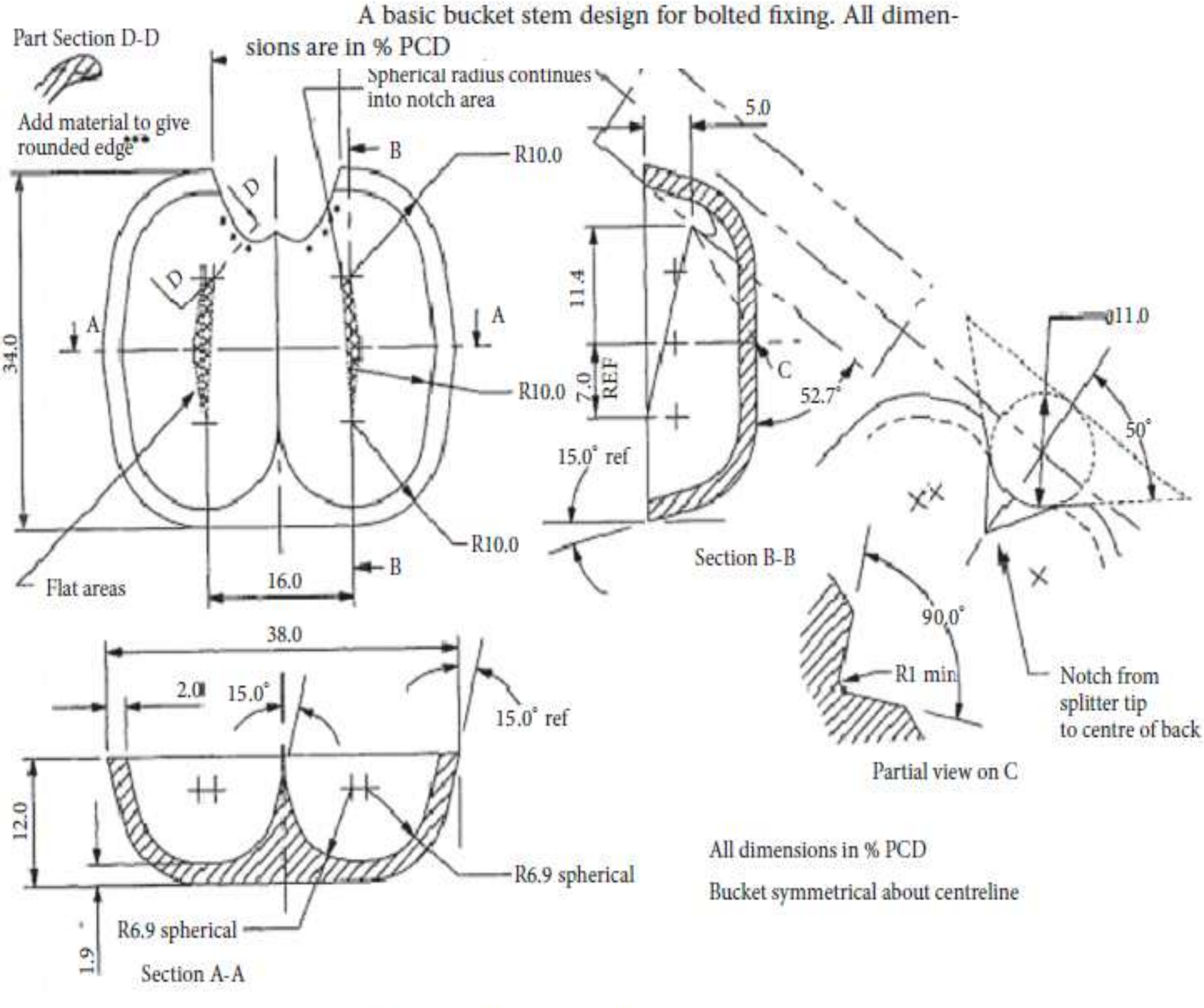
Table 3; Laboratory Constraint

Parameter		Unit	Value
Head (H)	Open System	m	30
	Closed System	m	150
Discharge (Q)		m^3/s	0.5
Torque (T)		Nm	2000

Table 4; Selected Model Turbine Operating Condition		
Parameter	**Unit**	**Value**
Head (H)	m	53.9
Discharge (Q)	m^3/s	0.05
Pitch Circle Diameter (PCD)	mm	400
Scale Factor	-	1:3.5



A basic bucket stem design for bolted fixing. All dimensions are in % PCD



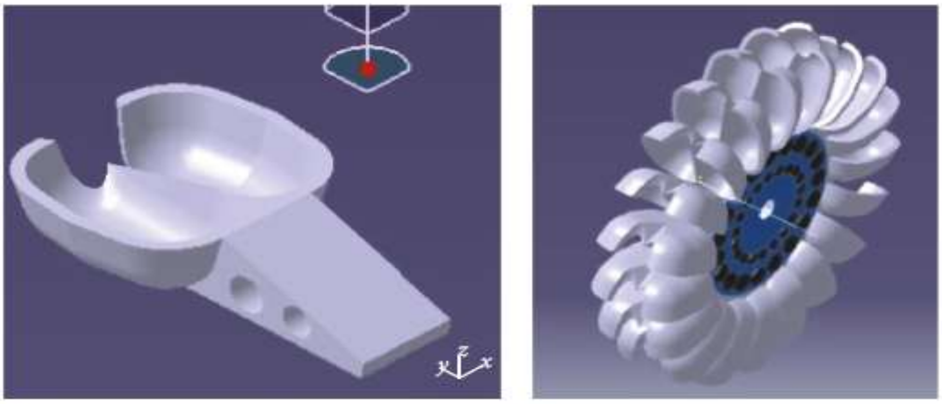
A scalable Pelton bucket. All dimensions are in % of PCD

Pelton turbine parts' assumed efficiency

Part	Symbol	Assumed efficiency
Penstock	η_p	0.95
Manifold	η_m	0.98
Nozzle	η_n	0.94
Runner	η_r	0.8
Drive	η_d	1
Generator	η_g	0.8
Overall efficiency	η_o	*0.56*

Physical baseline bucket dimensions result for the selected site data.

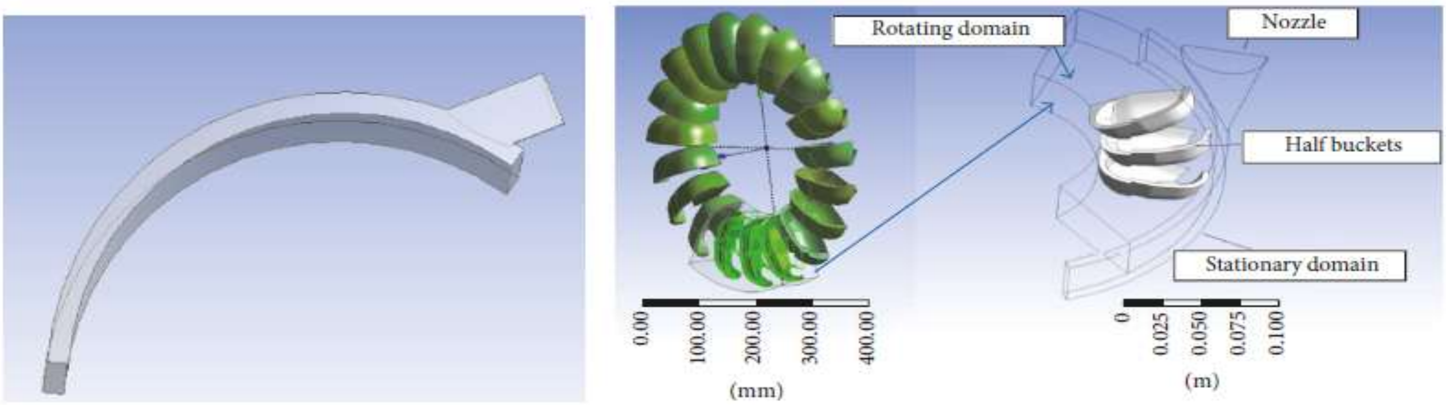
Parameters, formula [6]	Calculation	Dimensions/result	Unit
Height of bucket, $h = 0.34D$	$h = 0.34 \times 500$	170	mm
Cavity length: $h1 = 5.6\%D$	$h1 = (0.056) \times 500$	28	mm
Length to impact point: $h2 = 0.114D$	$h2 = 0.114 \times 500$	57	mm
Width of bucket opening, $a = 0.14D$	$a = 0.14 \times 500$	56	mm
Bucket thickness: $t1 = 0.002D$	$t1 = 0.002 \times 500$	1	mm
Approximate number of buckets, Z	$Z = D/2d + 15$	18	
Depth of the bucket, $t = 0.121D$	$t = 0.121 \times 500$	60.5	mm
Width of the bucket, $b = 0.38D$	$b = 0.38 \times 500$	190	mm



Solid model of Pelton turbine for the selected site data, (a) bucket and (b) 3D view of the runner right.

V. MESHING

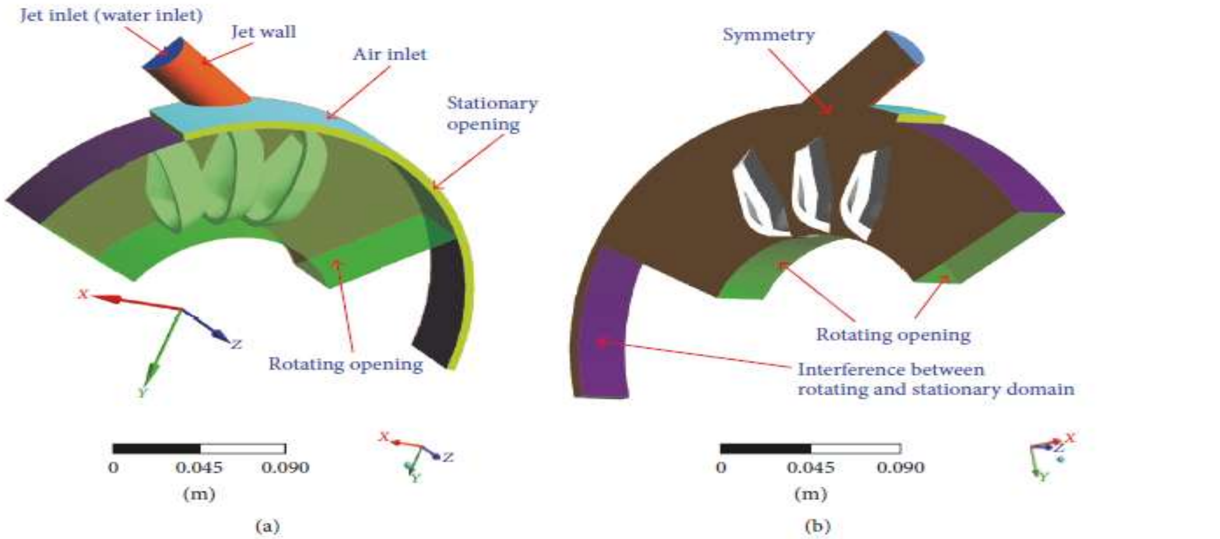
Stationary and rotating domain was discretized separately using ANSYS Meshing. The stationary domain, shown in Figure 5 (a), consists of two regions, water and air flow region. Water flow region is of prime interest while discretizing the stationary domain. Sweep method was used in core region of water while inflation method was used in boundary region of water and air [8]. The mesh in stationary domain consisted of structured hexahedral type mesh. Figure 5 (b), (c) shows the mesh in stationary domain.



Domain geometries: stationary (a) and assembly of the rotating and stationary domain (b).

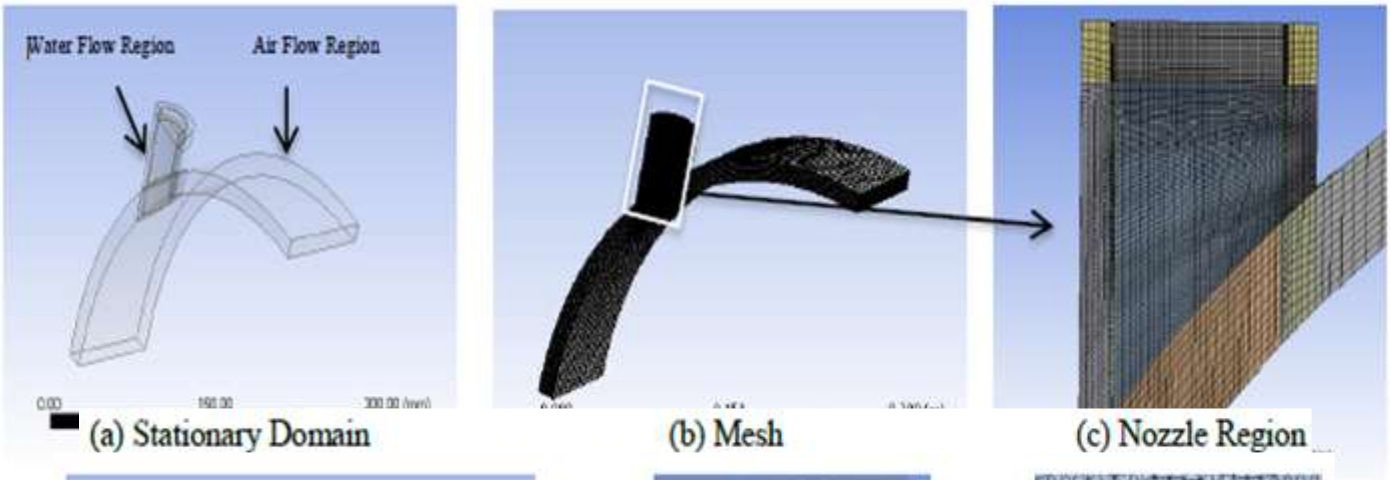
Expressions defined in ANSYS Pre for the baseline design

Name	Expression	Description
Gravity	9.82 [ms^{-2}]	Acceleration due to gravity
Head	13.34 [m]	Model head based on scaling
Turbine radius	132.5 [mm]	Model turbine radius based on scaling
Inlet velocity	(2 × gravity × head)$^{0.5}$ [m/s]	Water velocity at the nozzle inlet
Omega	Inlet velocity/(2 × turbine radius) [rad/s]	Angular velocity: rotation in the negative direction is selected
Torque middle bucket wall	*Torque_z@Middel Bucket Wall*	The entire middle bucket is selected
Total time	((120 × pi)/180)/Omega	Total simulation running time



Boundaries applied on the domains.

The rotating domain was divided into three region of interest where fine mesh was created. A body of influence method was used in inlet and outlet region of the bucket, and the bucket surface meshed using inflation method, shown in Figure 6 (a) [8].



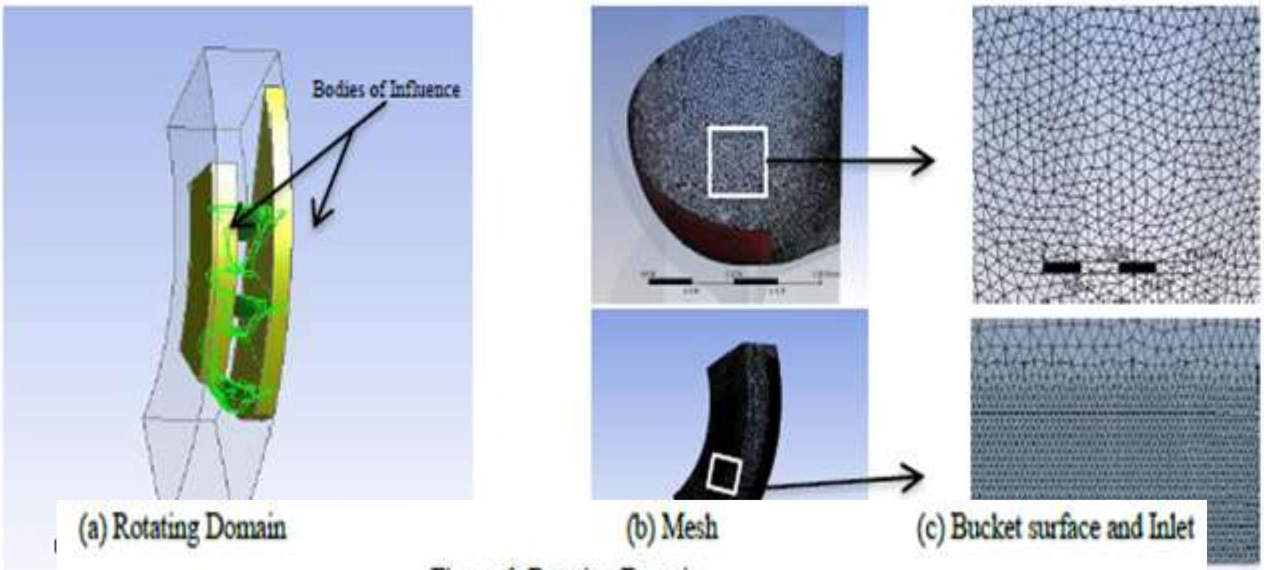


Figure 6: Rotating Domain

VI. NUMERICAL ANALYSIS

The governing equations of viscous flow are based on conservation of mass, momentum and energy which are langrangian in nature. The governing equations are expressed using equations 4 – 6 [9].

$$\textit{Conservation of Mass;} \quad \frac{\partial p}{\partial t} + \rho \Delta V = 0 \qquad \textit{Equation 4}$$

$$\textit{Conservation of Momentum;} \quad \rho\left(\frac{DV}{Dt}\right) = \rho g + \nabla.\tau'_{ij} - \nabla p \qquad \textit{Equation 5}$$

$$\textit{Conservation of Energy;} \quad \rho\left(\frac{Dh}{Dt}\right) = \frac{Dp}{Dt} + \nabla(k\nabla T) + \phi \qquad \textit{Equation 6}$$

The numerical analysis of CFD in Pelton Turbine consists of incompressible fluid flow that reduces the conservation of mass and momentum to equation 7-8 respectively. In addition, the temperature effect is negligible during the analysis. Therefor conservation of energy is ignored during analysis (9)

$$\nabla.V = 0 \qquad \textit{Equation 7}$$

$$\rho\left(\frac{DV}{Dt}\right) = \rho g + \mu \nabla^2 V - \nabla p \qquad \textit{Equation 8}$$

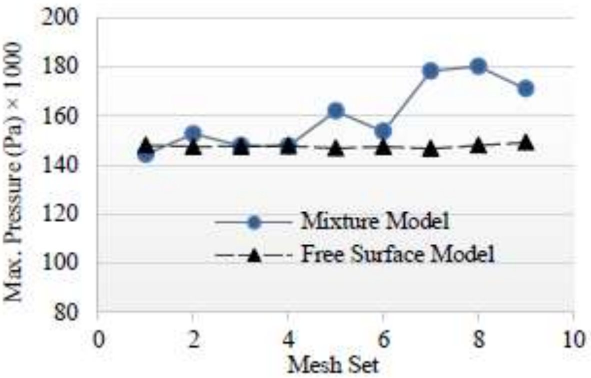


Figure 7: Maximum Pressure Vs. Mesh Set

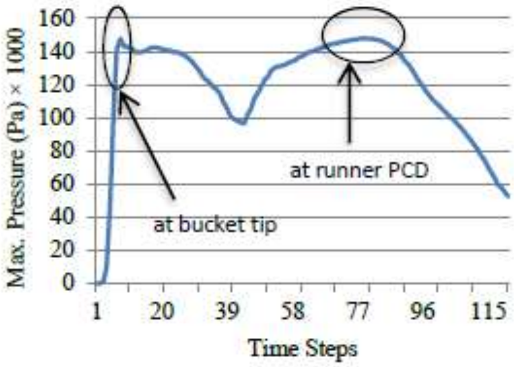
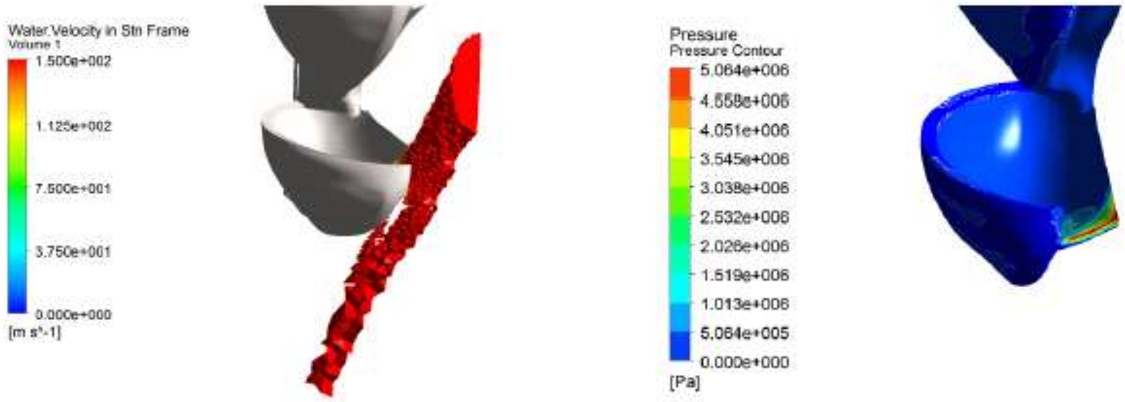
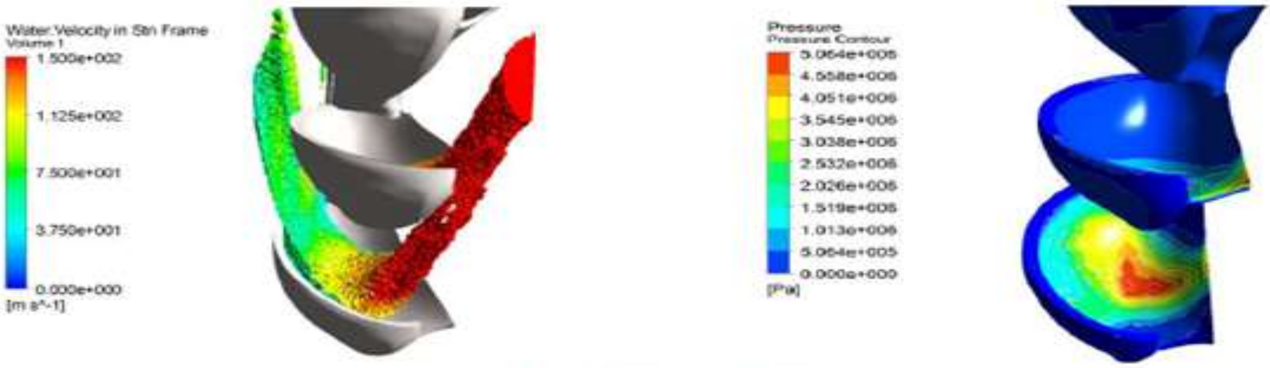


Figure 8. Maximum Pressure Vs. Time Steps



(a) At bucket tip

The standard k-ε model is extensively used due to its excellent performance. But, it shows poor performance in unconfined flow regions where the boundary is curved, and in rotating and swirling flows. Similarly, the Wilcox model does not require wall damping functions and it is robust in near wall regions. However, its robustness is decreased due to sensitivity in free stream region. SST model is the hybrid model which used the Wilcox model in near wall region and standard k-ε model in the fully turbulent region. A comparative study of turbulence model has been carried out using k-ε and k-ω based SST model separately in the paper [10].



(b) At Runner PCD

Figure 9: Water Volume Fraction above 0.75 (left) and Pressure Contour in Pelton bucket (right)

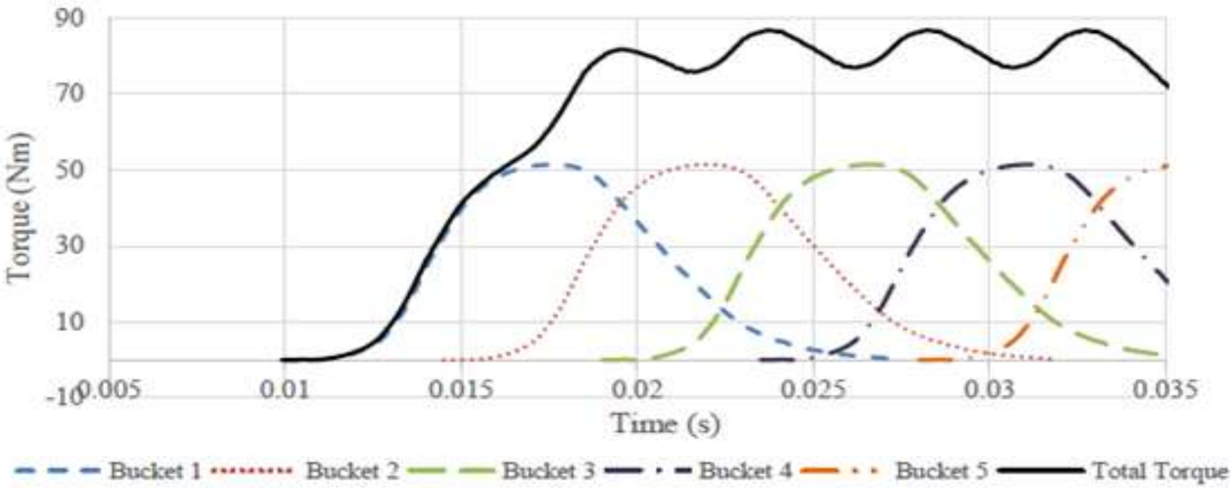
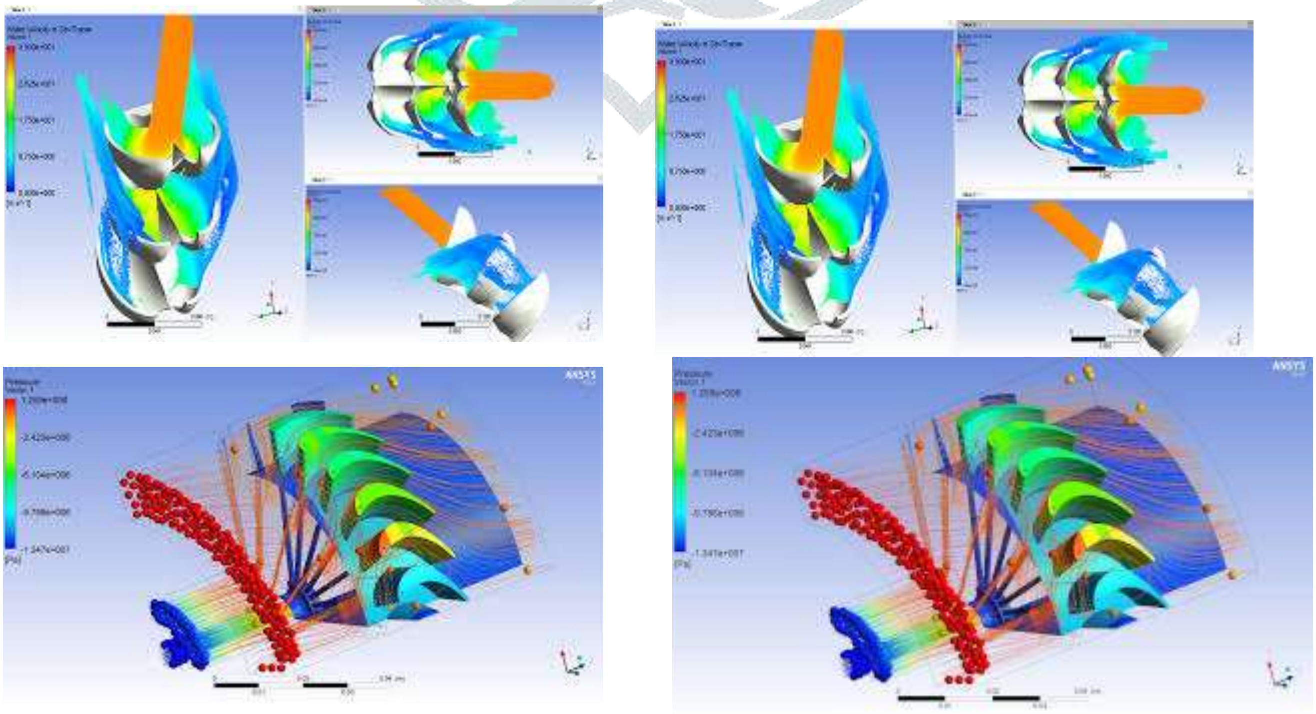
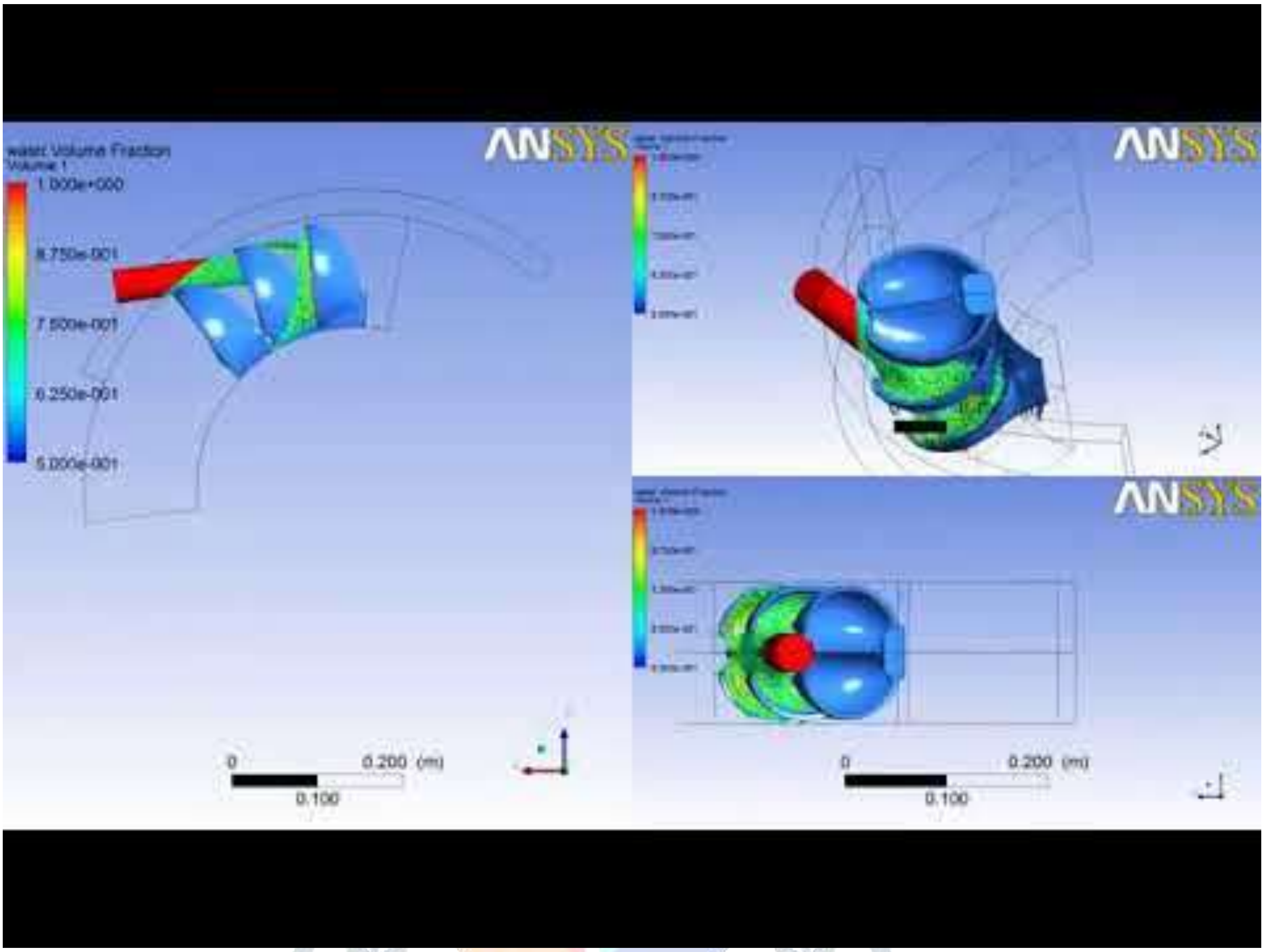


Figure 10. Torque generated by buckets over time

The domain type for both the rotating and stationary domain was defined fluid. Air and water, both at 25 0C, with continuous fluid morphology was used in analysis. The fluid model was selected as homogeneous model with standard free surface model. The interface compression level was set to 2 to produce better convergence. Shear Stress Transport (SST) type turbulence model has been used with automatic wall function. The surface tension model has been set as continuum surface force with primary fluid as water. Interphase transfer model was studied separately using free surface and mixture model. The initial conditions for the fluid volume fractions were defined 1 and 0 for air and water respectively. Since there is no pressure difference between the inlet and the outlet, the reference pressure was set to 1 atmosphere. And the domain motion option was set to rotating and stationary for rotating and stationary domain respectively. The symmetry boundary condition was applied in the middle plane diving the turbine into two sections. Smooth wall with no slip conditions was applied in the bucket wall.





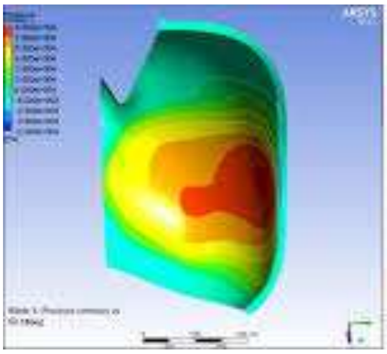
VII. RESULTS AND DISCUSSION

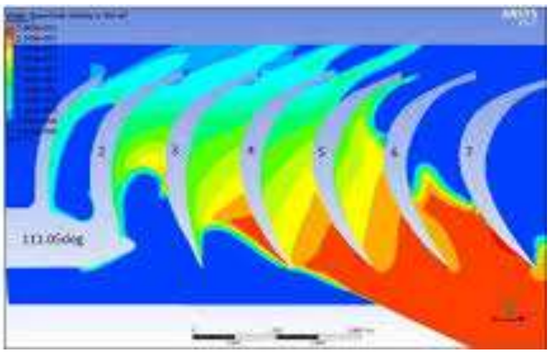
The CFD analysis was carried out using k-ε and SST model separately using mixture model and free surface model as interphase transfer method, as shown in Table 5. The simulation failed to converge using k-□ turbulence model. However, SST model converge for both mixture and free

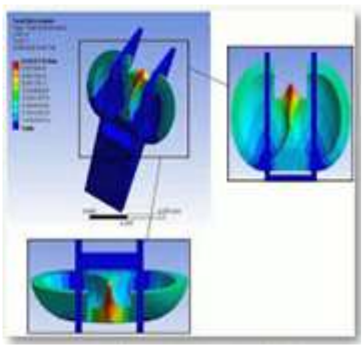
Surface interphase models. This is due to the fact that k-e model has poor performance in unconfined flow where the boundary is curved. But, SST model uses k-ε and Wilcox model in fully turbulent and near wall region respectively.

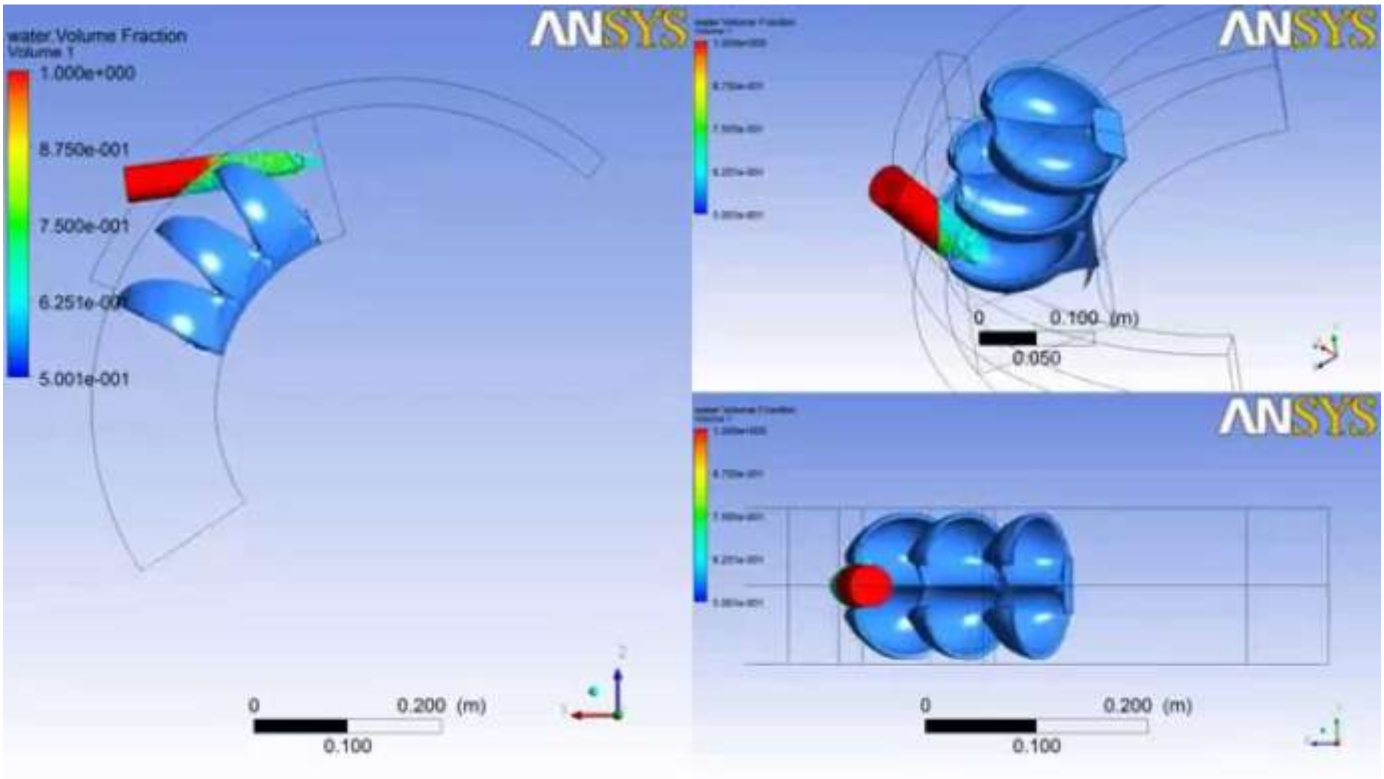
Table 5: Convergence vs. Turbulence Model and Interphase Transfer

	Mixture Model	Free Surface Model 1
K E Model	Failed	Failed
SST Model	Converged	Converged

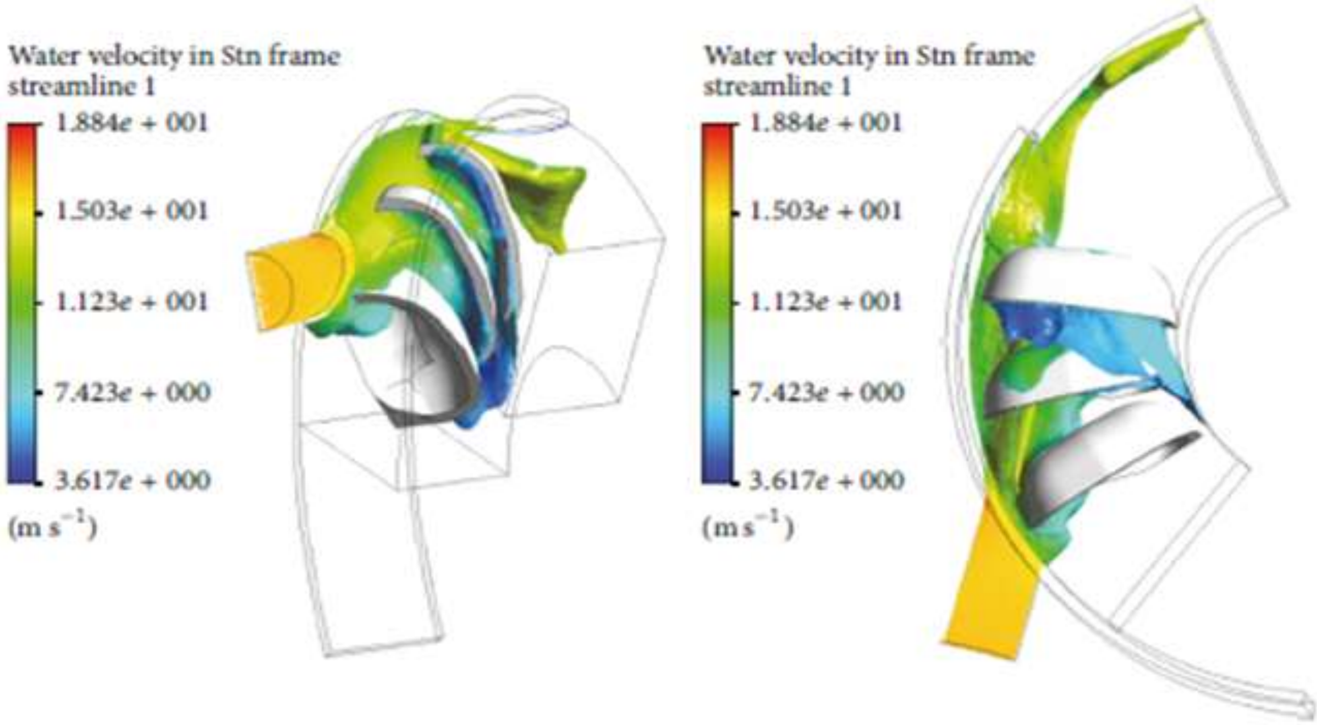




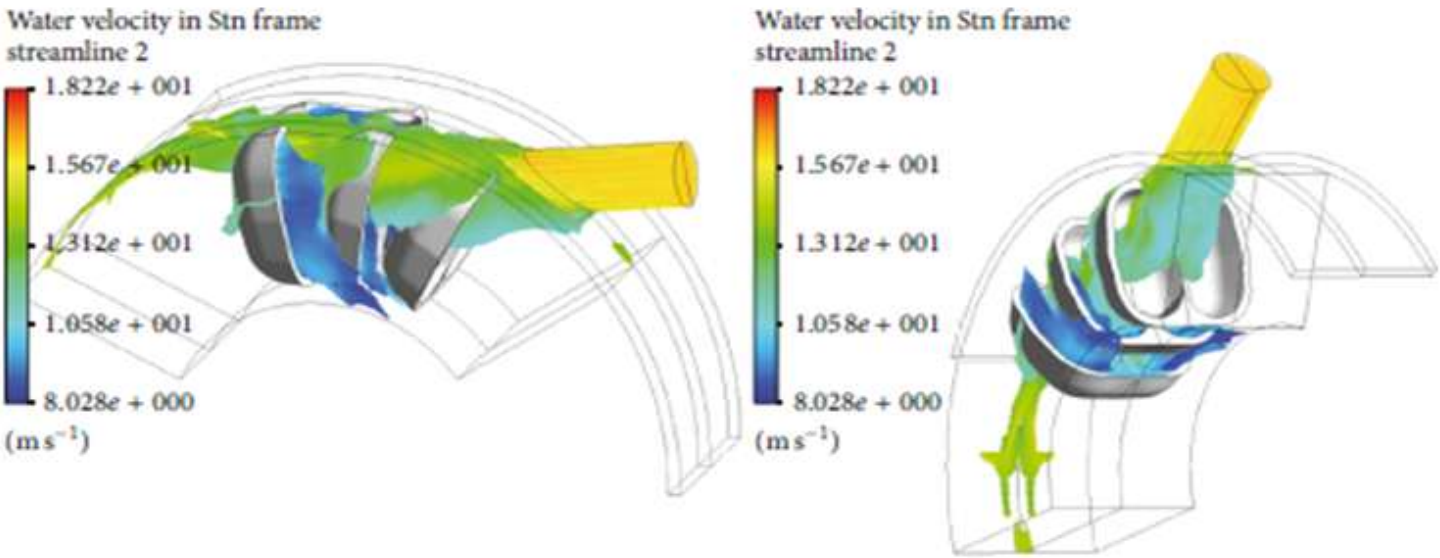




Mesh dependent test was carried out to computationally validate the result in seven different mesh sets. The total number of nodes was varied from 0.75 million to 4.3 million. Figure 7 shows the mesh dependent test using SST model using mixture model and free surface model as interphase transfer method. It was found that the results are better when using free surface model [11].



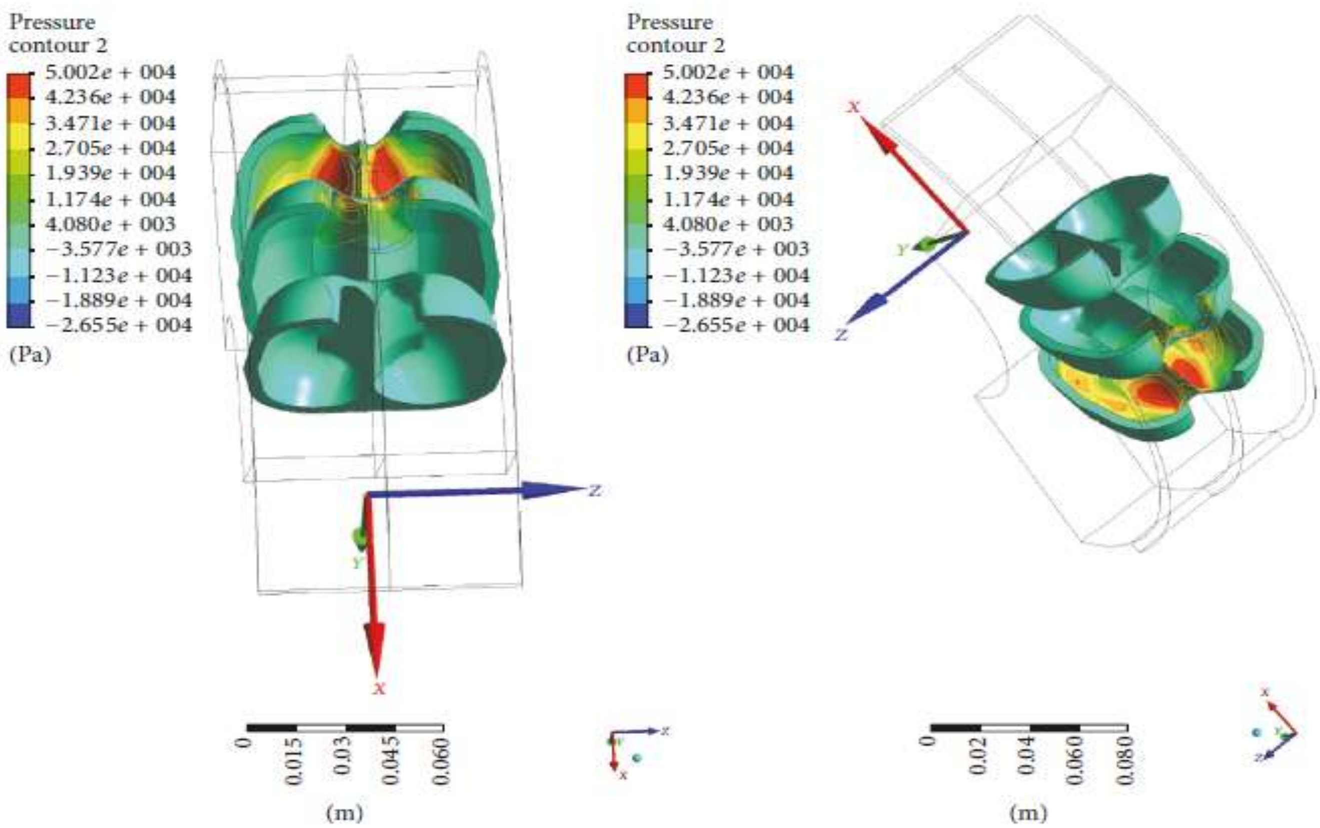
Flow visualization of the baseline design, side views, and face views



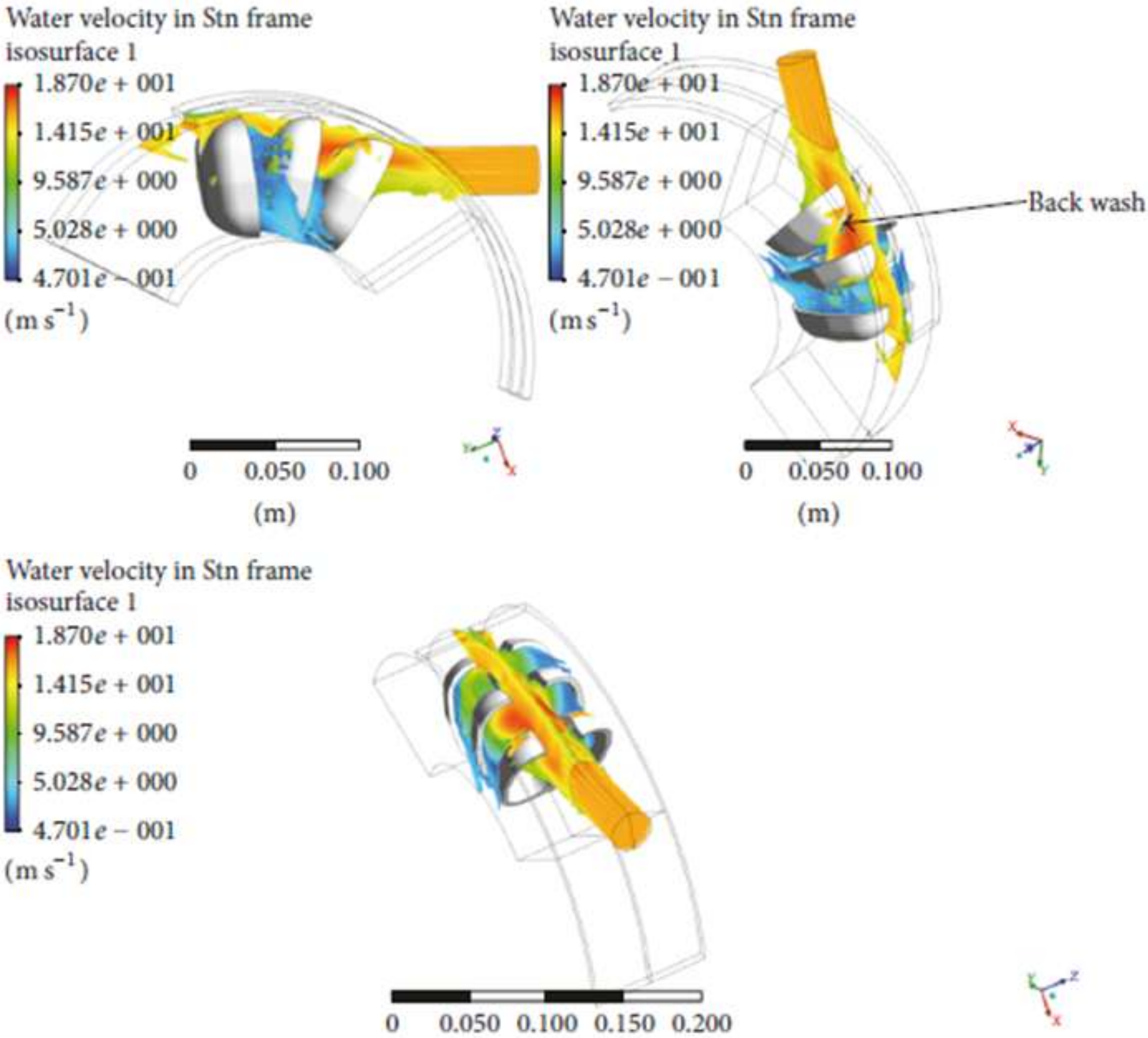
Flow distribution of the runner: water velocity in Stn frame, side views, and face views

The pressure distribution in middle bucket was exported from ANSYS CFX, shown in Figure 8. It was found that the pressure peaks are obtained at bucket tip and PCD of runner. The pressure peak in bucket tip is due to flow disturbance when jet strikes bucket tip. It is obvious to obtain the pressure peak at runner PCD since the Pelton runner are designed such that it would convert most of the hydraulic energy to mechanical energy when the jet strikes the runner PCD [12]. The water volume fraction with velocity index and pressure distribution in Pelton bucket is shown in Figure 9.

The torque generated by the runner can be predicted by using the torque data produced by middle bucket. A single torque data is replicated over time calculating the frequency of bucket during rotation, using equation 9. The calculated frequency for the model turbine is 0.0045 seconds. Figure 10 shows torque generated by different buckets and total torque produced by Pelton turbine. The comparison of computational torque with analytical solution obtained using equation 10 showed that mechanical efficiency of the turbine is 82.5%

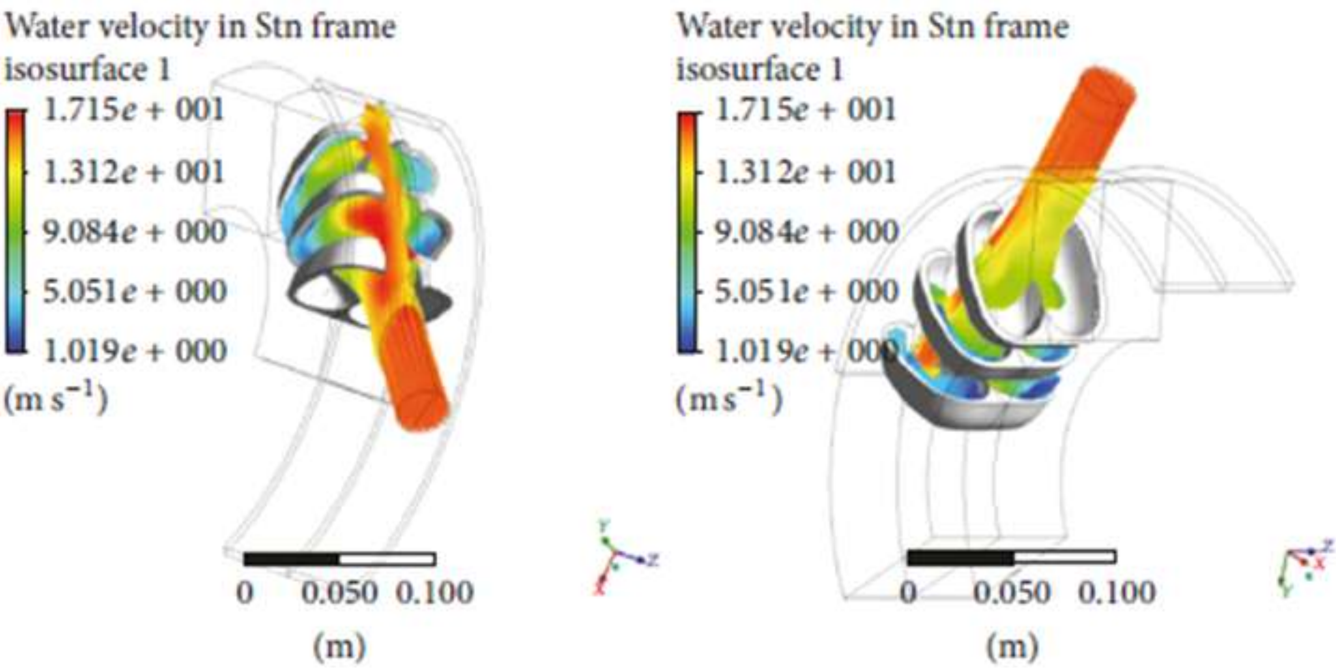


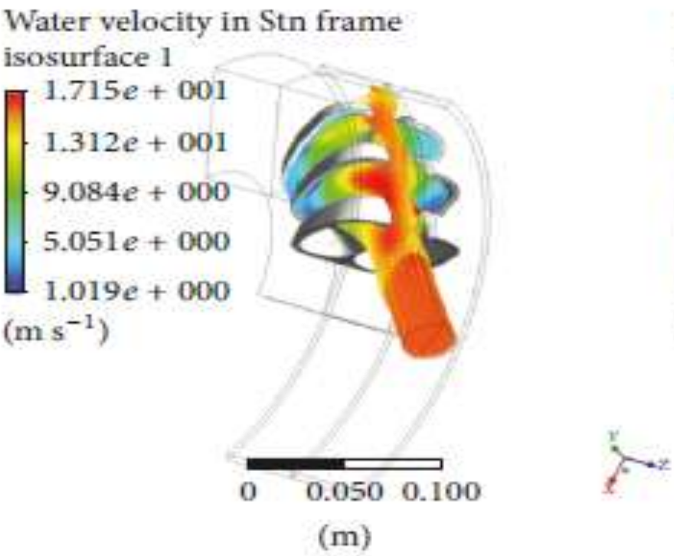
Pressure contours on the bucket

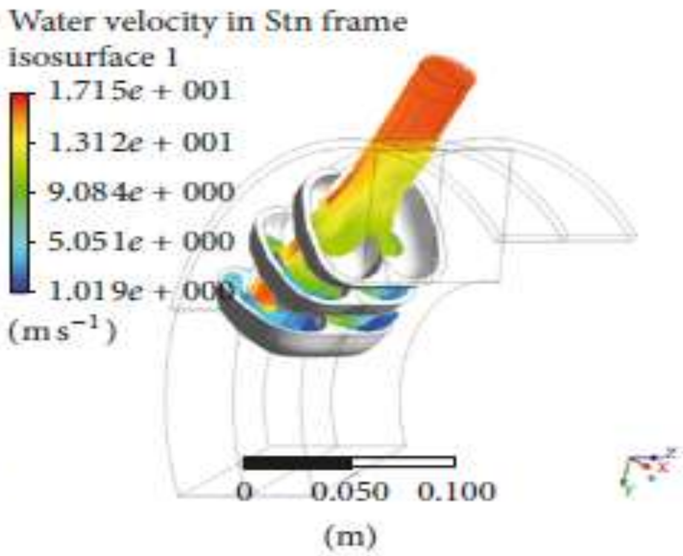


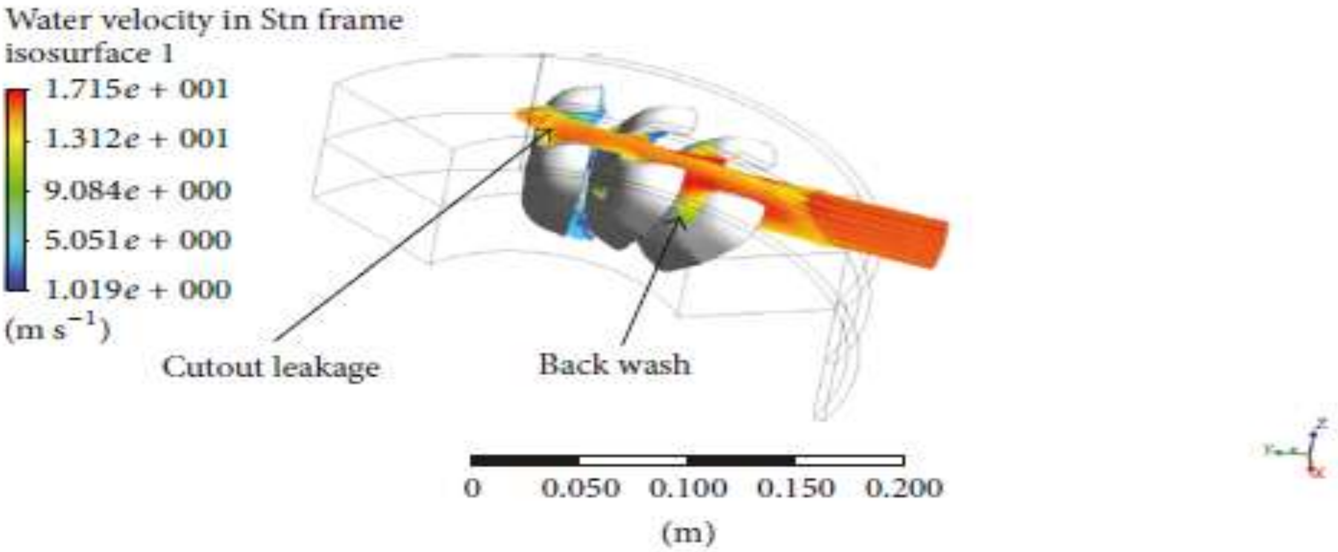
(b)

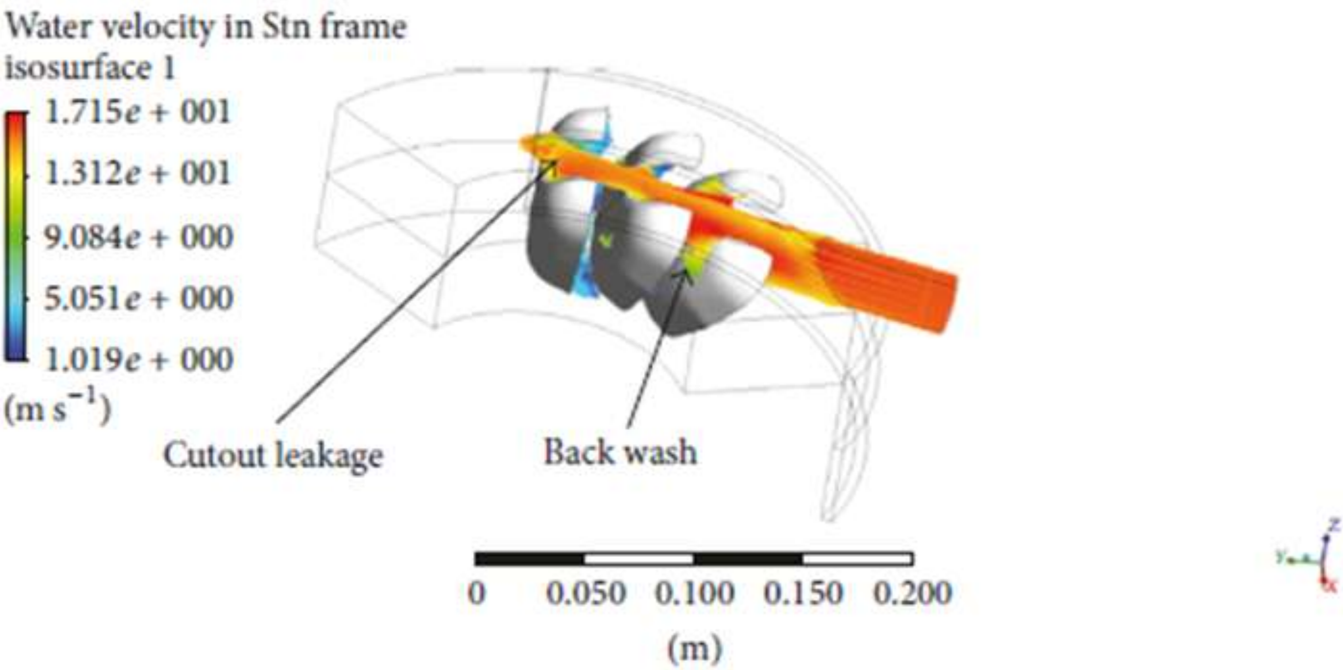
Flow distribution of the runner: water velocity in stn frame, side views and face views

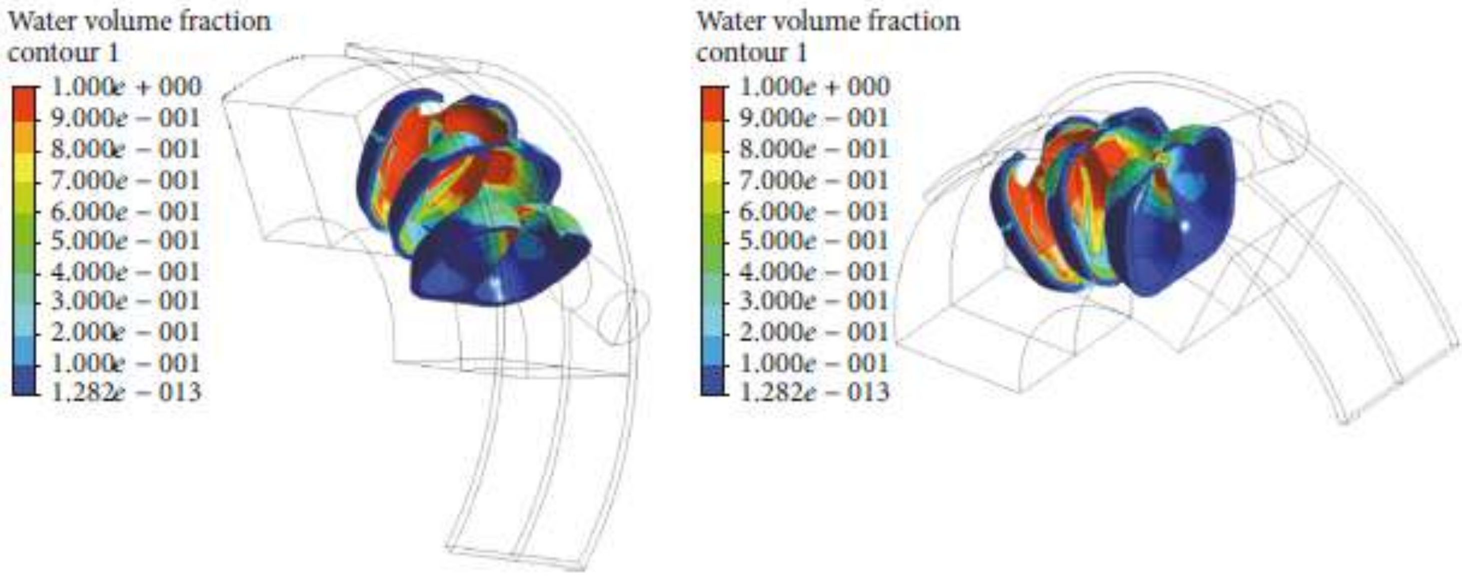



Water velocity in Stn frame
isosurface 1
1.715e + 001
1.312e + 001
9.084e + 000
5.051e + 000
1.019e + 000
(m s^-1)
0 0.050 0.100
(m)

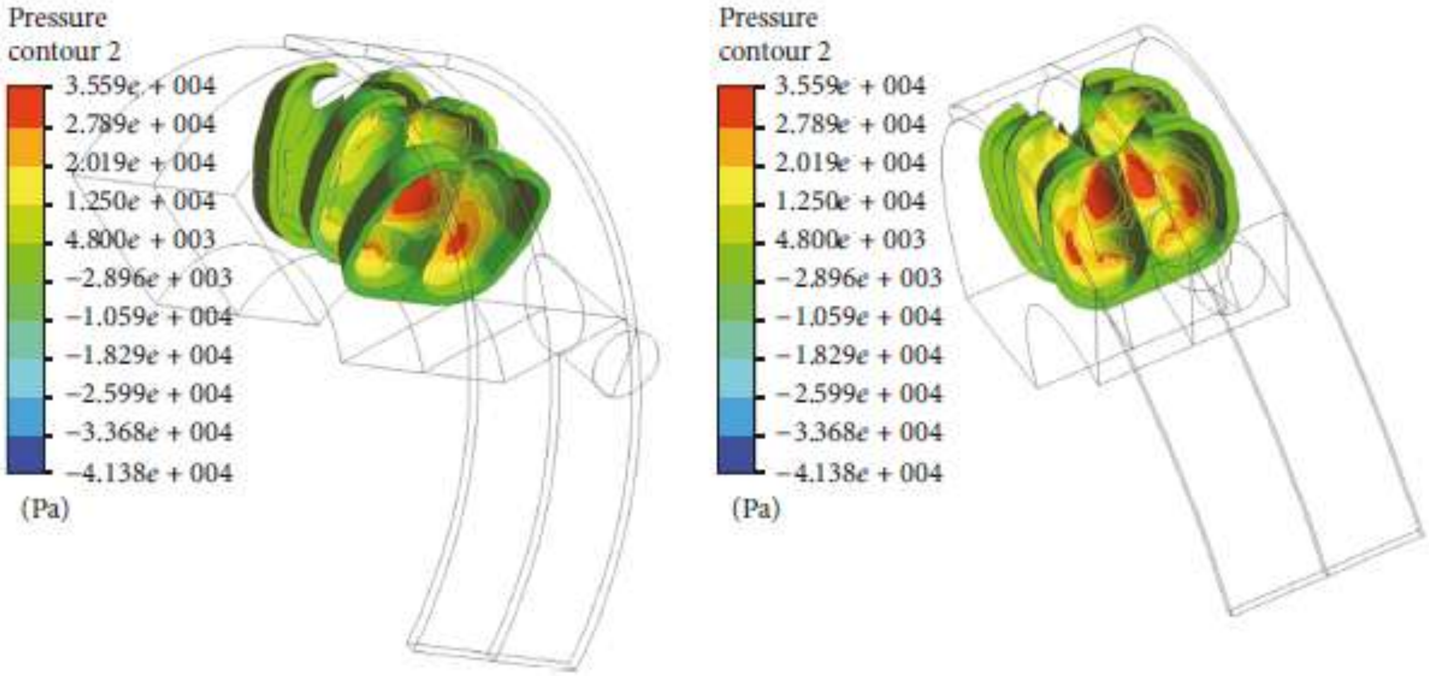

Water velocity in Stn frame
isosurface 1
1.715e + 001
1.312e + 001
9.084e + 000
5.051e + 000
1.019e + 000
(m s^-1)
0 0.050 0.100
(m)


Water velocity in Stn frame
isosurface 1
1.715e + 001
1.312e + 001
9.084e + 000
5.051e + 000
1.019e + 000
(m s^-1)
Cutout leakage
Back wash
0 0.050 0.100 0.150 0.200
(m)


Water velocity in Stn frame
isosurface 1
1.715e + 001
1.312e + 001
9.084e + 000
5.051e + 000
1.019e + 000
(m s^-1)
Cutout leakage
Back wash
0 0.050 0.100 0.150 0.200
(m)



Volume fraction of water for reference bucket design at a particular time step



Pressure distribution at some degree of rotation of reference bucket design

$$Freequency = \frac{Number\ of\ Buckets \times \Omega}{2\Pi} \qquad Equation\ 9$$

$$Power = \frac{2\pi NT}{60} \qquad Equation\ 10$$

VIII. CONCLUSION

The CFD analysis of scaled Pelton turbine of Koyna Hydropower was performed using ANSYS CFX software. The scale factor for selected model turbine was 1:3.5. Scaling of the turbine reduces computational time and cost. The time and cost in CFD analysis of Pelton turbine is also reduced by selecting 3 buckets to predict the behavior of complete turbine. The result showed that SST model is robust turbulence model to conduct CFD analysis of Pelton turbine. In addition, free surface interphase transfer method gives better result than mixture model. It was found that peak pressure is obtained at bucket tip and PCD of runner. The pressure distribution in each bucket surface was exported using monitor tool in ANSYS CFX for further analysis on fatigue of Pelton turbine. The torque results obtained from the single bucket can be replicated over time to predict the total torque transferred by the Pelton turbine. The torque results obtained from CFD showed that the model Pelton turbine has efficiency of 82.5%.

REFERENCES

[1] A. Sharma, P. Sharma, A. Kothari, Numerical Simulation of Pressure Distribution in Pelton Turbine Nozzle for the Different Shapes of Spear, International Journal of Innovations in Engineering and Technology, Vol 1, Issue 4, December 2012.
[2] ANSYS User Guide, Version 14.0, 2011
[3] A. Panthee, B. Thapa, H. P. Neopane, Quality Control in Welding Repair of Pelton Turbine, Proceedings in 3rd Asia Pacific Forum of Renewable Energy (AFORE), Jeju, Korea, November 4 – 7, 2014
[4] L. Souari, M. Hassairi, Numerical Simulation of the Flow into a Rotating Pelton Bucket, International Journal of Emerging Technology and Advanced Engineering, Vol. 3, Issue 2, February 2013.
[5] L. F. Barstad, CFD Analysis of a Pelton Turbine [Master, thesis], Norwegian University of Science and Technology, 2012.
[6] J.Thake,the Micro-Hydro Pelton Turbine Manual: Design, Manufacture and Installation for Small-Scale Hydro-Power, ITDG publishing, London, UK, 2000.
[7] Z. Zhang, Pelton turbines book, published by Springer Nature, the registered company is Springer International Publishing, AG, Switzerland, 2016.
[8] MHPG, Series, “HarnessingWater Power on a Small Scale” Vol. 9 (Micro PeltonTurbines), Published by SKAT, SwissCenter for Appropriate Technology, 1991.
[9] A. Harvey, Micro hydro design manual, a guide to small scale water power schemes.
[10] A. Panthee, B. Thapa, and H. P. Neopane, “CFD Analysis of pelton runner,” International Journal of Scientific and Research Publications (IJSRP), vol. 4, no. 8, 2014.
[11] A. Zidonis, A. Panagiotopoulos, G. A. Aggidis, J. S. Anagnostopoulos, and D. E. Papantonis, “Parametric optimization of two Pelton turbine runner designs using CFD,” Journal of Hydrodynamics, vol. 27, no. 3, pp. 840–847, 2015.
[12] J. D. Anderson, Computational Fluid Dynamics: The Basics with Applications, McGraw-Hill, New York, NY, USA, 1995.
[13] A. Perrig,Hydrodynamics of the free surface low in Pelton turbine buckets [Ph. D. Thesis], ´Ecole polytechnique f´ed, 2007.
[14] A. Perrig, F. Avellan, J.-L. Kueny, M. Farhat, and E. Parkinson, “Flow in a Pelton turbine bucket: numerical and experimental investigations,” Journal of Fluids Engineering, vol. 128, no. 2, pp. 350–358, 2006.
[15] Y. X. Xiao, T. Cui, Z.W.Wang, and Z. G. Yan, “Numerical simulation of unsteady free surface flow and dynamic performance for a Pelton turbine,” in Proceedings of the 26th IAHR Symposium on Hydraulic Machinery and Systems, chn, August 2012.