Blade Profile Optimization of Kaplan Turbine Using CFD Analysis

AJAZ BASHIR JANJUA*, MUHAMMAD SHAHID KHALIL**, AND MUHAMMAD SAEED***

RECEIVED ON 26.07.2012 ACCEPTED ON 18.09.2012

ABSTRACT

Utilization of hydro-power as renewable energy source is of prime importance in the world now. Hydro-power energy is available in abundant in form of falls, canals rivers, dams etc. It means, there are various types of sites with different parameters like flow rate, heads, etc. Depending upon the sites, water turbines are designed and manufactured to avail hydro-power energy. Low head turbines on run-of-river are widely used for the purpose. Low head turbines are classified as reaction turbines. For run-of river, depending upon the variety of site data, low head Kaplan turbines are selected, designed and manufactured. For any given site requirement, it becomes very essential to design the turbine runner blades through optimization of the CAD model of blades profile. This paper presents the optimization technique carried out on a complex geometry of blade profile through static and dynamic computational analysis. It is used through change of the blade profile geometry at five different angles in the 3D (Three Dimensional) CAD model. Blade complex geometry and design have been developed by using the coordinates point system on the blade in PRO-E /CREO software. Five different blade models are developed for analysis purpose. Based on the flow rate and heads, blade profiles are analyzed using ANSYS software to check and compare the output results for optimization of the blades for improved results which show that by changing blade profile angle and its geometry, different blade sizes and geometry can be optimized using the computational techniques with changes in CAD models.

Key Words: **Low Head, Kaplan Turbine, Runner Blade, CFD, Optimization, Hydropower.**

1. INTRODUCTION

Demand for increasing the use of renewable energy has risen over the last two decades due to environmental issues. The high emissions of greenhouse gases have led to serious changes in the climate. Although the higher usage of renewable energy would not solve the problems over night, it is an important move in the right direction. Use of water power energy can be very much effective in this regard.

Hydropower remained the most important source of the renewable energies for electrical power production worldwide, providing 19% of the planet's electricity. Small-

* Ph.D. Student, Department of Mechanical Engineering, University of Engineering & Technology, Taxila.
** Professor, Department of Mechanical Engineering, University of Engineering & Technology, Taxila.
*** Assistant Professor, Department of Mechanical Engineering, MA Jinnah University Islamabad.

scale hydro is in most cases "run-of-river", and is one of the most cost-effective and environmentally benign energy technologies to be considered both for rural electrification in less developed countries and further hydro-power development worldwide [1].

Although Pakistan is enrich in hydro-power potential which is around 50,000 MW [2], but hydropower does not play a significant role in energy production for the country. Hydro power generation is the only solution to control the severe energy crisis in Pakistan.

Waterpower as an energy source was used dates back centuries. It was the first renewable source used for electricity generation over 100 years ago. For this purpose, simple water wheels were used. Over the years, this equipment has been developed and become more and more advanced. Hydro-power is an important source of producing electrical energy; approximately 20% of the world electricity is supplied by hydroelectric power plants [3]. World's first hydroelectric power scheme was developed in Northumberland, England by William Armstrong in 1878.

Hydroelectric power plants can be equipped with a specific turbine in order to get the highest efficiency, depending on the head and discharge of the sites. Water turbine is a water engine which converts the energy of water into effective work in a rotating runner. It uses pressure energy and kinetic energy of water to produce mechanical power due to action of water on blades of runner. This exchange of energy between water and rotating runner blades creates changes in flow velocities and angles from inlet of runner to outlet [4].

Water turbines are classified as impulse turbines and reaction turbines. Impulse turbine converts only the kinetic energy of the water (the kinetic energy = $v^2/2g$). It means that water with atmospheric pressure is directed onto a set of blades placed circumferential around a shaft. Pelton turbines are the most commonly used impulse turbine. Reaction turbines convert both the potential energy of the water (pressure, p/w) and the kinetic energy (velocity, $v^2/2g$) into useful work. Propeller/Kaplan Turbines and Francis Turbines are the reaction turbines.

Kaplan turbines work in the lower head range, from only a few meters up to around 30m [5]. Low head hydro sites ranging from 2-10m have even larger potential for the provision of electric power in rural areas of developing countries, but on the other hand the harnessing of this potential is severely hampered due to lack of an appropriate turbine design. Fixed geometry propeller turbines are the most cost-effective turbine design options for low head hydropower. [6-7].

Francis turbine is a reaction turbine and likely the Kaplan turbines, design and geometry of the Francis turbine blades is also very important. Jose, et. al. [8], applied CFD (Computational Fluid Dynamics) performance on the runner blades design to improve their efficiency and cost-effectiveness beyond the traditional redesign practices.

Helena, et. al. [9], carried research work on the hydraulic machines with the aim of optimization and the selection of adequate turbines of low power based on analyses of 3D hydrodynamic flows have been carried out leading to the best possible results by adopting different criteria. Peng, et. al. [10], carried research work on the design optimization of axial flow hydraulic turbine runner, in order to obtain a better design plan with good performance. Research review on the previous work indicates that attempt on the research work on subject area is rare. In this study, it is being focused to fill this gap and carry forward the latest research.

In the reviewed research work as referred in this section [9-10], it is to be pointed that optimization work has been carried on the axial turbine components on single geometry/profile basis. Research on the single component with multiple profiles for optimization is not addressed in the previous work. To overcome this drawback, research has been carried out in this paper by developing the turbine blade with changes in the profile geometry for five different cases.

2. DESIGN AND DEVELOPMENT OF RUNNER BLADE

Keeping in view the previous research on optimization in water turbine runner blades, area of design and development/optimization through geometrical changes in the blade runner profile was focused for new research. The same is illustrated in the next sections. Water turbines are the most important part for hydro-power generation in the country. For different range of heads and flow rates, different types of turbines are used. Kaplan turbines are used for low head applications. The main characteristics of design are the data on which the design of the runner is based.

From the localization and topology of the power station in which the turbine is going to operate, the fundamental design parameters for any turbine design can be determined. Head and the amount of water flow give not only information of how much energy that can be produced but also to determine the type of turbine, the basic design parameters.

Material for turbine runners is generally cast iron, steel or stainless steel. Mostly material ASTM A743 CA6NM stainless steel is used [11].

A selection criterion of this material is based on its better strength, resistance ability to cavitations, easily cast-able and to fabricate, weld-able and protection against erosion. Mechanical properties and chemical composition of this material are adequate for use in water turbines.

Turbine basic data was obtained from the Nandi-Pur Hydro Power Plant near Gujranwala, Pakistan, and based on site data like design flow rate of 3040 cusecs (86 m^3/sec), gross head of 6.7 meters and hydraulic efficiency of around 80%, technical characteristics of the turbine are:

Maximum power output=4.6 MW, Runner diameter=4.24m, and Hub diameter=1.76m.

Typical plan view of the turbine runner-blade assembly is shown in Fig. 1.

For optimization of the turbine runner blades using the latest computational techniques, it is essential to develop the complex 3D model of the blade. For this purpose, actual blade sample is used to create the model in CAD. Blade geometry was then obtained through x-axis, y-axis and z-axis coordinates by assuming the test bed level as the datum line reference. Based on this data, development of the complex blade geometry for initial design, have been carried out using AutoCAD and Pro-E/Creo.

In Table 1, coordinates for inner side and outer side of the blade geometry are shown and similarly in Table 2, coordinates for the development of right hand side and left hand side are shown. CMM machine and 3D scanner are now-a-days commonly used for the development of complex CAD models in the industry as well as in the academic institutes. In this study, Pro-E/Creo have been utilized to develop CAD models of turbine runner blade with the help of coordinate points system. Through this procedure, different CAD models of blade profiles with required geometries were possible to be developed.

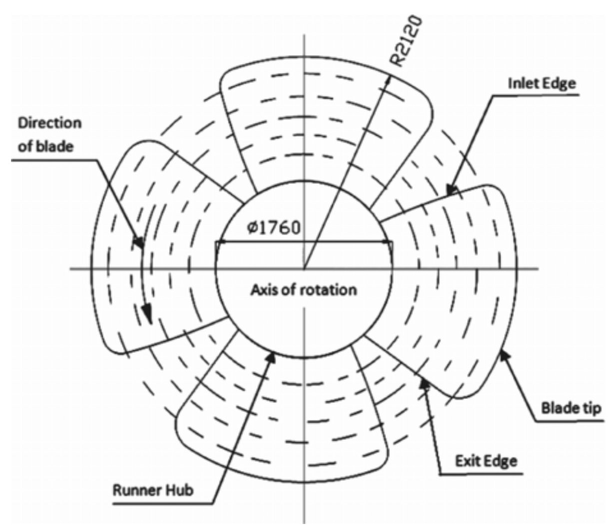

FIG. 1. PLAN VIEW OF RUNNER BLADE ASSEMBLY

TABLE 1. COORDINATE POINTS FOR BLADE DEVELOPMENT(FOR INNER AND OUTER SIDE)

INNER UPPER SIDE													
X (mm)	643	600	500	400	300	200	100	0	-100	-200	-300	-400	-457
Y (mm)	-1519	-1476	-1396	-1336	-1293	-1263	-1246	-1240	-1246	-1263	-1293	-1336	-1368
Z (mm)	13	32	74	117	159	215	266	311	348	381	409	439	456
INNER LOWER SIDE													
X (mm)	643	600	500	400	300	200	100	0	-100	-200	-300	-400	-457
Y (mm)	-1519	-1476	-1396	-1336	-1293	-1263	-1246	-1240	-1246	-1263	-1293	-1336	-1368
Z (mm)	0	6	21	36	50	68	94	131	178	238	310	383	425

OUTER SIDE UPPER		
X (mm)	Y (mm)	Z (mm)
-100	-2	287
-200	-10	284
-300	-21	281
-400	-38	280
-500	-60	280
-600	-87	282
-700	-119	287
-750	-140	290
-775	-158	292
-800	-185	296
-825	-229	301
0	0	291
100	-2	293
200	-10	295
300	-21	297
400	-38	298
500	-60	298
600	-87	298
700	-119	298
800	-157	296
900	-201	294
1000	-251	290
1100	-308	283
1200	-379	275
1250	-447	269
1270	-497	265
1281	-569	257

OUTER SIDE LOWER		
X (mm)	Y (mm)	Z (mm)
-100	-2	215
-200	-10	217
-300	-21	218
-400	-38	222
-500	-60	224
-600	-87	226
-700	-119	236
-750	-140	245
-775	-158	249
-800	-185	261
-825	-229	271
0	0	223
100	-2	221
200	-10	224
300	-21	232
400	-38	236
500	-60	240
600	-87	244
700	-119	247
800	-157	249
900	-201	253
1000	-251	254
1100	-308	253
1200	-379	247
1250	-447	243
1270	-497	242
1281	-569	241

Points were measured physically from the runner blade on test bench i.e. surface table by using the well calibrated measuring instruments like height gauges to ensure the accuracy of the measurements taken by qualified quality personnel. True leveled surface of the test bench was taken as reference for these measurements.

In this procedure, runner blade was placed and leveled on the test bench/surface table. Blade upper and lower surfaces were properly cleaned before marking of points. Outer periphery of the blade on upper and lower sides was measured by using calibrated measuring tape. At start, initial reference point was marked as coordinate (0,0,0) and from this reference point, next point was marked on the periphery line at an interval of 100mm with the help of divider. This point was then measured with respect to initial point by measuring the dimensions along x-axis, y-axis and z-axis.

Dimensions along x and y-axis were measured with the help of calibrated vernier calipers, whereas dimension along z-axis was measured by using the calibrated height gauge. In this way, all the points/coordinates were measured and plotted on the drawing.

Blade geometries are developed with utilization of the point's coordinate system. Sequence of the working on drawings to finally develop CAD 3D model is as shown in Fig. 2.

CFD is becoming an increasingly reliable tool for the design of water turbines. Using different CFD codes, it is possible to find out and compare criteria for classifying runner blade geometry regarding the strengths of their characteristics. The final decision of runner geometry, always remains for the design engineer. To reach the final result, designer have to compare the flow analysis results of a great number of different geometries. Liplej, et. al. [12], carried research work on the optimization methods and techniques for the design of axial hydraulic turbines.

For the optimization and analysis purpose and to obtain solutions for comparison of the results, blade model was re-developed for five different angle positions as

TABLE. 2 COORDINATE POINTS FOR BLADE DEVELOPMENT (FOR LEFT AND RIGHT SIDE)

LEFT HAND UPPER SIDE										
X (mm)	-837	-825	-800	-775	-750	-700	-650	-600	-550	-500
Y (mm)	-295	-364	-452	-536	-616	-766	-905	-1035	-1158	-1274
Z (mm)	300	309	320	330	341	361	382	401	421	440
LEFT HAND LOWER SIDE										
X (mm)	-837	-825	-800	-775	-750	-700	-650	-600	-550	-500
Y (mm)	-295	-364	-452	-536	-616	-766	-905	-1035	-1158	-1274
Z (mm)	268	276	287	298	308	329	349	369	388	407
RIGHT HAND UPPER SID										
X (mm)	1270	1250	1200	1100	1000	900	800	700		
Y (mm)	-650	-700	-773	-907	-1041	-1175	-1308	-1442		
Z (mm)	237	224	205	170	136	102	67	33		
RIGHT HAND LOWER SIDE										
X (mm)	1270	1250	1200	1100	1000	900	800	700		
Y (mm)	-650	-700	-773	-907	-1041	-1175	-1308	-1442		
Z (mm)	223	211	191	157	123	88	54	20		

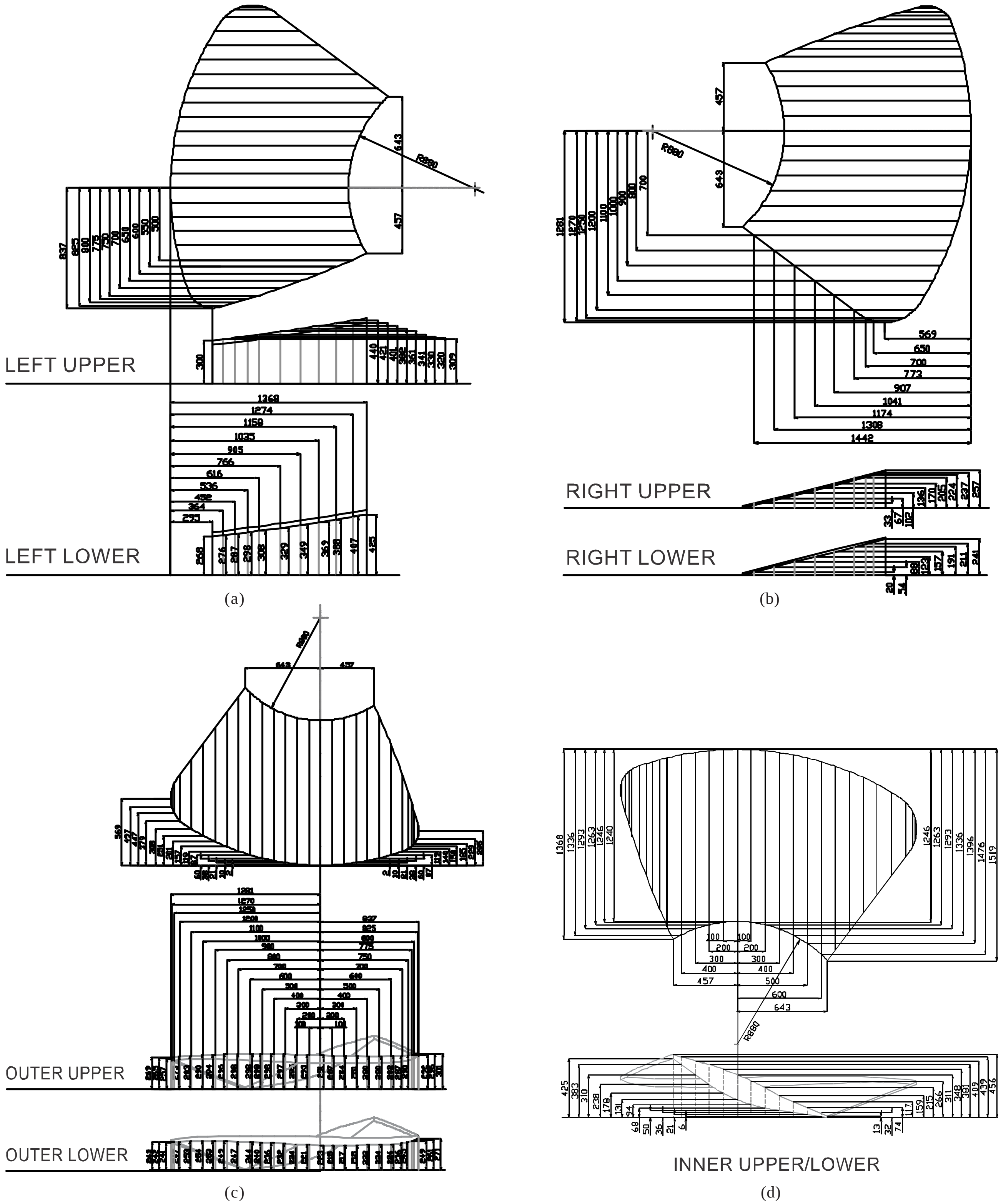

FIG. 2. DEVELOPMENT OF DRAWING FOR BLADE MODEL

shown in Fig. 3. These five blade models are developed at five different angles named as Case-A with angle 14.120^o, Case-B with angle 16.120^o, Case-C with angle 18.120^o, Case-D with angle 12.120^o and Case-E with angle 10.120^o.

Now-a-days CFD compliments experimental and theoretical approach by providing an alternate cost effective means of simulating real flow. Prasad, et. al. [13-14] carried out the numerical flow simulation in the axial turbines to study the effect of operating conditions on the turbine using CFX module.

CFD techniques have also been applied for various types of hydro turbines. A computational fluid dynamics-based design system enables a quick and efficient design optimization of turbine components. Jingchun, et, al. [15] applied it to a Francis turbine runner, guide vanes, and stay vanes using CFX.

3. CFD OF TURBINE RUNNER BLADE

This research presents the utilization of CFX computational fluid dynamics to analyze the different runner blade profile models for optimization. CFD study is done using a commercial Software package ANSYS, Inc.[16]. CFD modules like ICEM-CFD and ANSYS-CFX are used for meshing, pre-processing, solution and post-processing. CFD work on the original geometry was carried out and results obtained were validated with the operational / experimental results of Kaplan turbine that is installed at Nandi-Pur Hydro Power Plant near Gujranwala, Pakistan. After validations of parameters of turbine among experimental and obtained from CFD four further models were analyzed as discussed in Section 2 and shown in Fig. 3.

3.1 Mesh Generation

Mesh was generated using one of the best meshing software ICEM-CFD. ANSYS ICEM CFD is intuitive built-in geometry creation and highly automated functionality for modifying and repairing imported CAD data. It is robust tools for creating all mesh types including hexahedral, tetrahedral, prism, pyramid, quad, tri or bar elements. ICEM CFD is highly tolerant of imperfect/over detailed CAD data containing sliver surfaces, gaps, holes and overlaps (patch independence) and has smoothing, coarsening, refinement, element type conversion, linear or quadratic element support. It has more than 100 CFD and CAE (Computer Aided Engineering) solver interface formats.

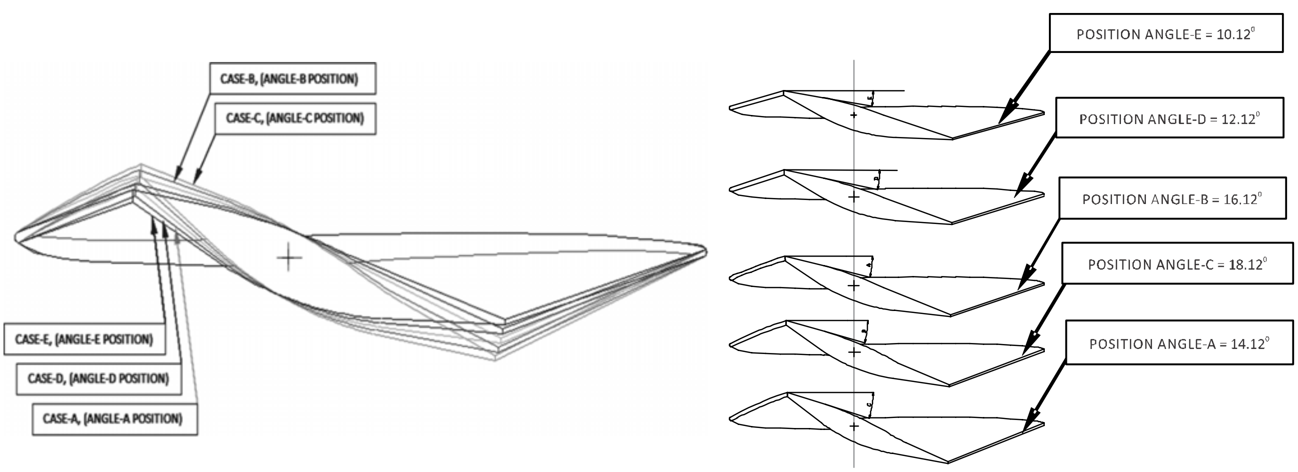

FIG. 3. DEVELOPMENT OF BLADE GEOMETRY AT DIFFERENT ANGLES

Geometry from Pro-E /Creo 2.0 was imported to ICEM CFD where after performing geometry repairs and dividing into different parts like Hub, Shroud, Periodic surfaces and blades initial topology was built as shown in Fig. 4.

Topology (blocking) was associated with the geometry and initial mesh was generated as shown in Fig. 5 (high periodic is shown in the Fig. 5). Mesh was generated of 90^{o} section containing one blade. After that periodic surfaces were generated to estimate the effect of other three blades as shown in Fig. 6.

3.2 Boundary Conditions

Total pressure was imposed at inlet boundary while mass flow condition was imposed at outlet boundary. Outer casing (shroud) was given stationary wall boundary conditions while blades and hub were given rotating wall

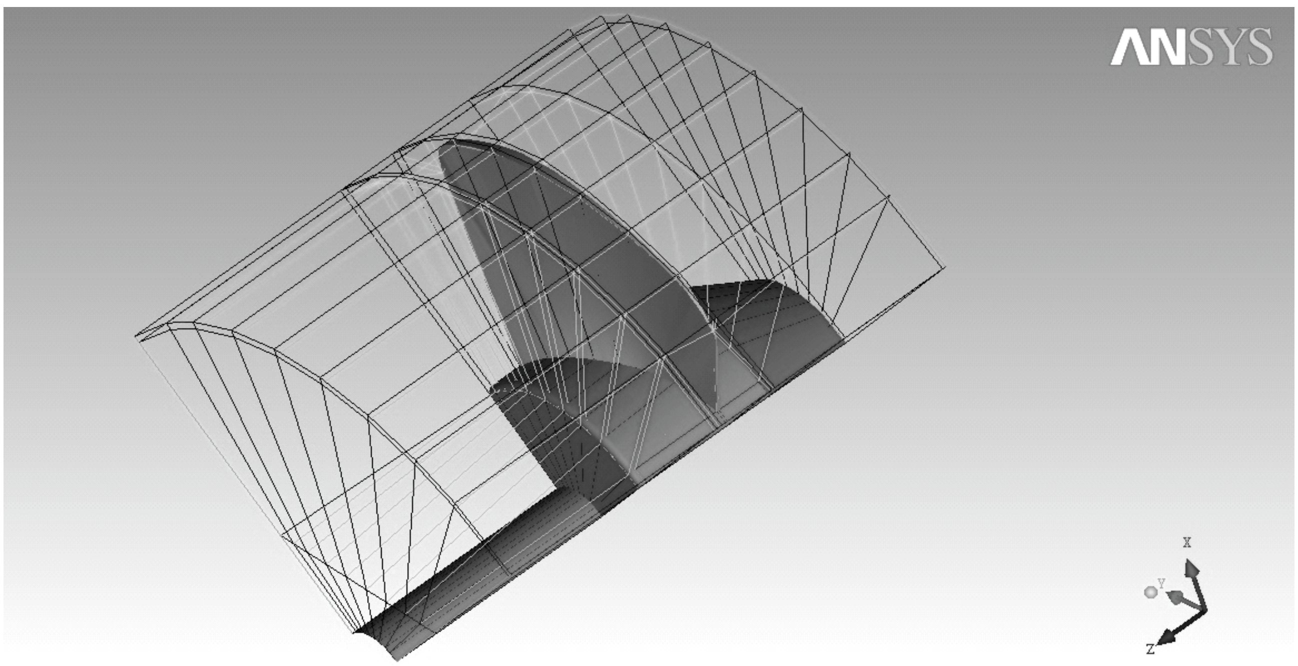

FIG. 4. TOPOLOGY AND BLOCKING OF THE BLADE

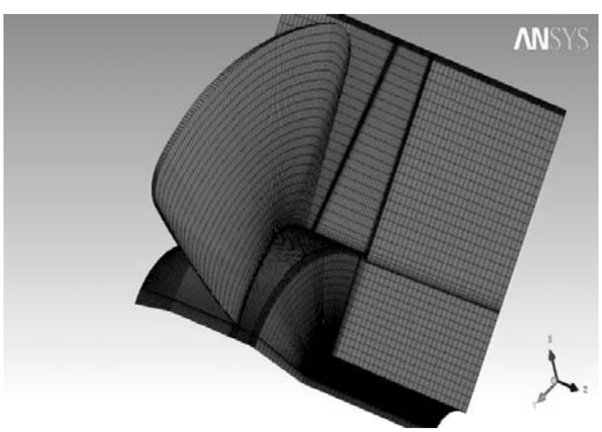

FIG. 5. MESHING OF THE BLADE

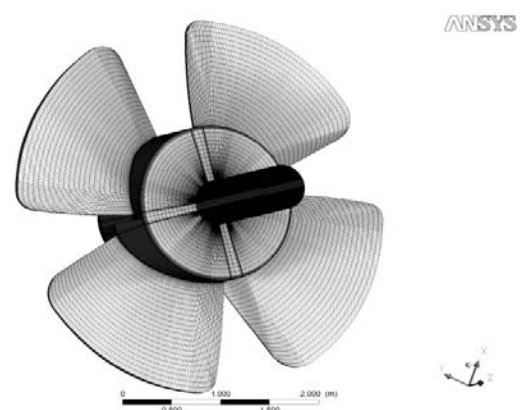

FIG. 6. MESHING OF ALL BLADES

boundary condition. Periodic surfaces are given as periodic boundary condition interface in CFX-Pre.

4. RESULTS AND DISCUSSION

4.1 Case A (Original Geometry)

As a first case analysis of original geometry were carried out at different mass flow conditions. And CFD results were compared with the experimental results of the turbine. Operational data of the hydro turbine unit have been obtained from the Nandi Pur, Hydro Power Plant, near Gujranwala, Pakistan for experimental purposes. Experimental/operational data is being compared with CFD results to analyze the final results and to set a reference line for optimization. Comparison of the results is shown in Fig. 7. It indicates the trend that power output of experimental data and CFD analysis are approximately at same level at higher flow rates. It means high flow rates play key role for the improvement in the turbines output efficiency.

To check and validate the resulted power output from CFD as an example, power was calculated for flow rate of 80.7 m^3/sec, by using the formula of power i.e. P=wQH. Power output of 4.24 MW was obtained after calculation. Accordingly power output obtained from CFD was found 4.3 MW, so validating the results calculation.

Pressure contours and flow velocity vectors on the blade surface and on hub are shown in Fig. 8. This pressure data can be exported to ANSYS structural solver for structure analysis. It can be seen clearly that pressure before the blade is high while as water passes. Through the turbine blades its pressure decrease as energy is delivered by the water to turbines that make turbine blades to rotate.

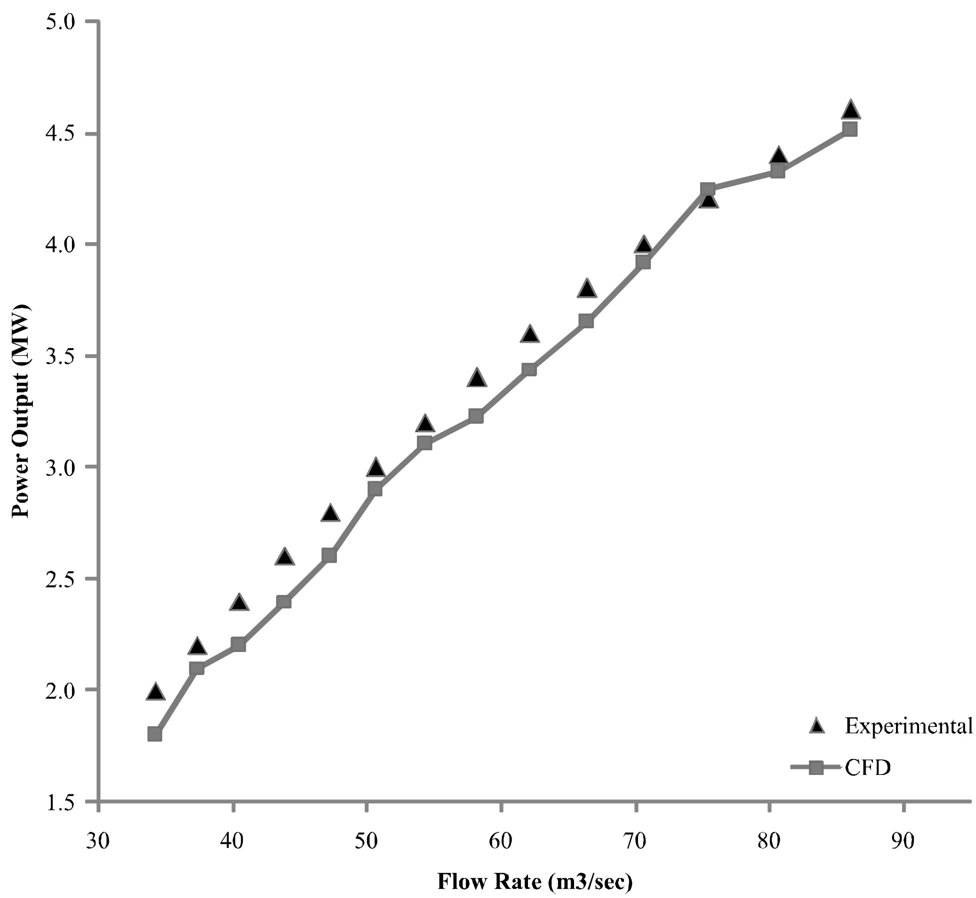

FIG. 7. COMPARISON OF EXPERIMENTAL AND CFD DATA

Pressure contours and flow velocity vectors on a 50% span of the turbo surface of turbine assembly are shown in Fig. 9.

Pressure contours and velocity vectors on 25, 50, 75 and 97% spans are shown in Figs. 10-13 respectively. These pictures provide flow visualization obtained from CFD results carried on the turbine blade assembly. This flow visualization suggests that flow is streamline along the blade at different spans. No major re-circulation and flow dead zones could be seen in these figures that it-self proves the validity and accuracy of CFD results.

4.2 Cases B-E (Developed Geometries)

In the previous section, original blade geometry i.e. Case-A was discussed. In this section, runner blades have been geometrically developed at four angles on hub side for optimization. Two blades named Case-B and Case-C are at an angle position of 16.120^o and 18.120^o respectively as shown in Fig. 3. Whereas other two blades named Case-D and Case-E are at an angle position of 12.120^o and 10.120^o respectively.

CFD analysis of the developed geometries for Case-B to Case-E was carried out at different mass flow conditions. Results obtained for these four cases and first Case-A, were then compared with the experimental results of the turbine.

Results obtained from the CFD analysis of blade for different cases are plotted in Fig. 14. By taking CFD results of Case-A as reference point, other results for the blade optimization have been compared, which indicated the improved power output in the system.

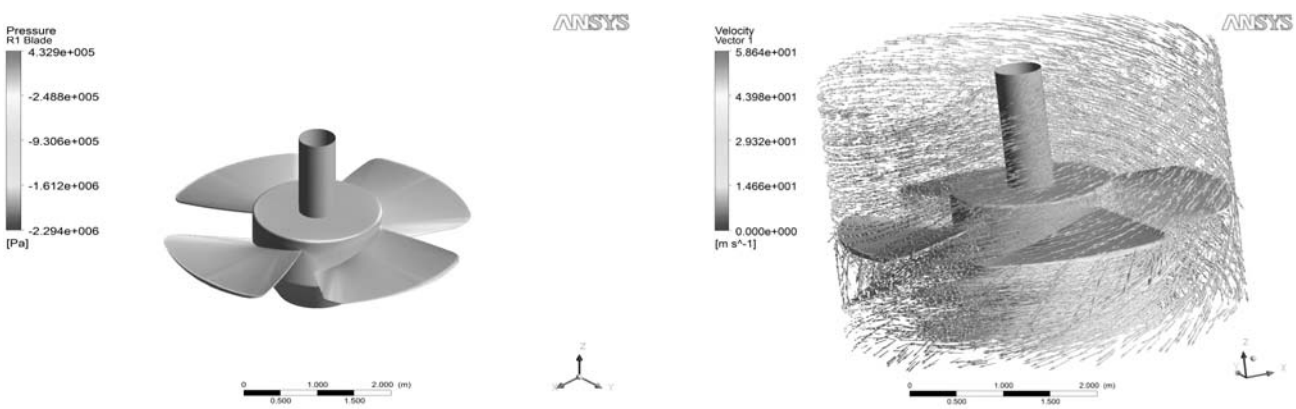

FIG. 8 PRESSURE CONTOURS ON TURBINE BLADES AND VELOCITY VECTORS

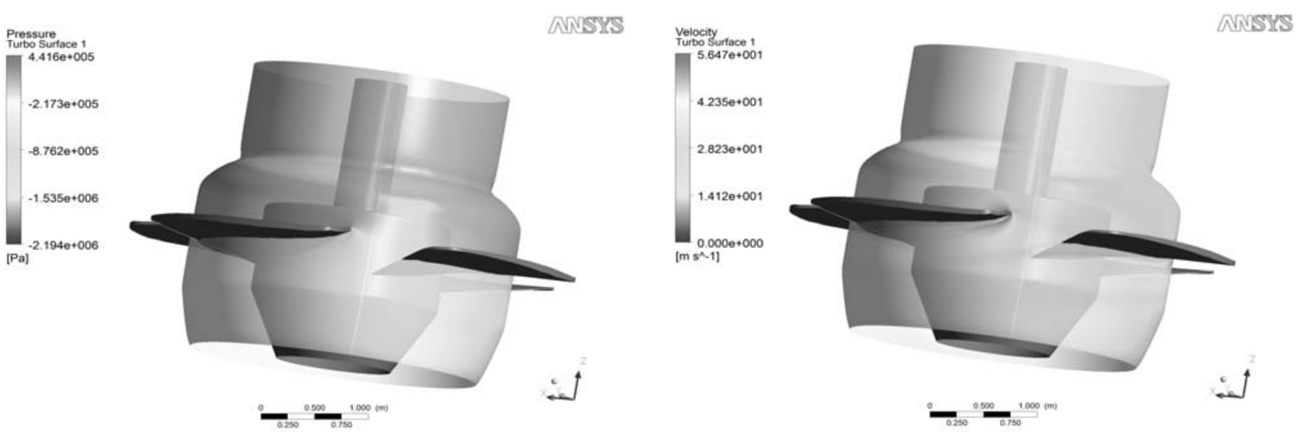

FIG. 9. PRESSURE AND VELOCITY CONTOURS ON 50% SPAN TURBO SURFACE

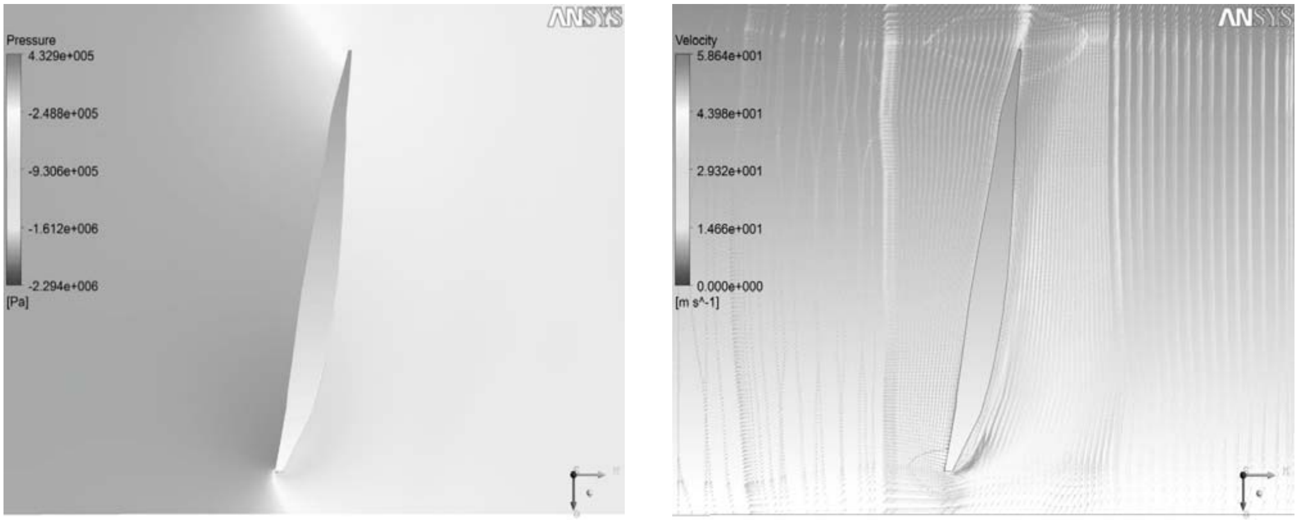

FIG. 10. PRESSURE CONTOURS AND VELOCITY VECTORS ON 25% SPAN

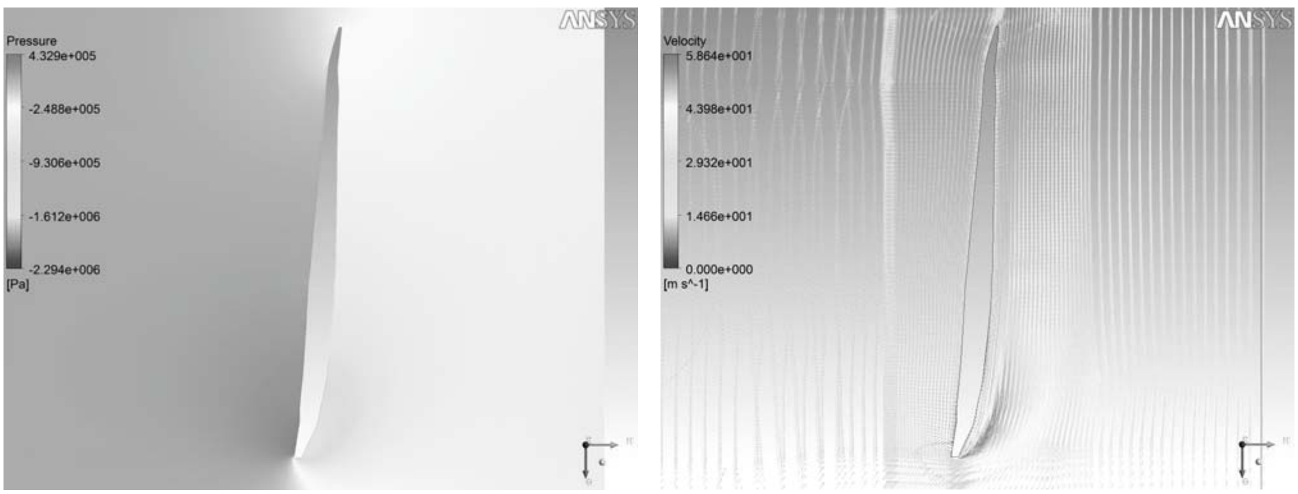

FIG. 11. PRESSURE CONTOURS AND VELOCITY VECTORS ON 50% SPAN

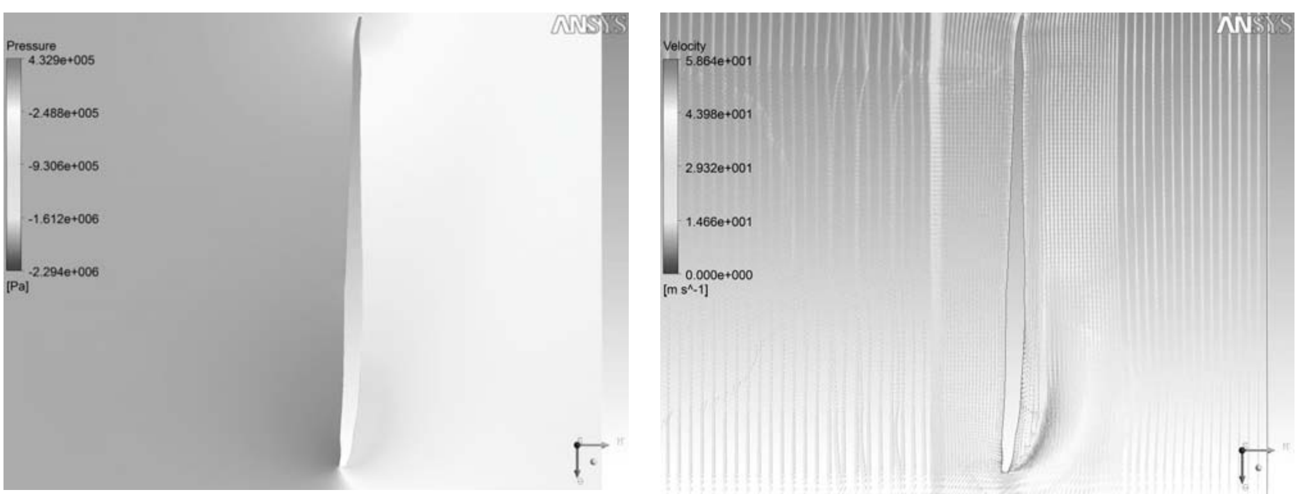

FIG. 12. PRESSURE CONTOURS AND VELOCITY VECTORS ON 75% SPAN

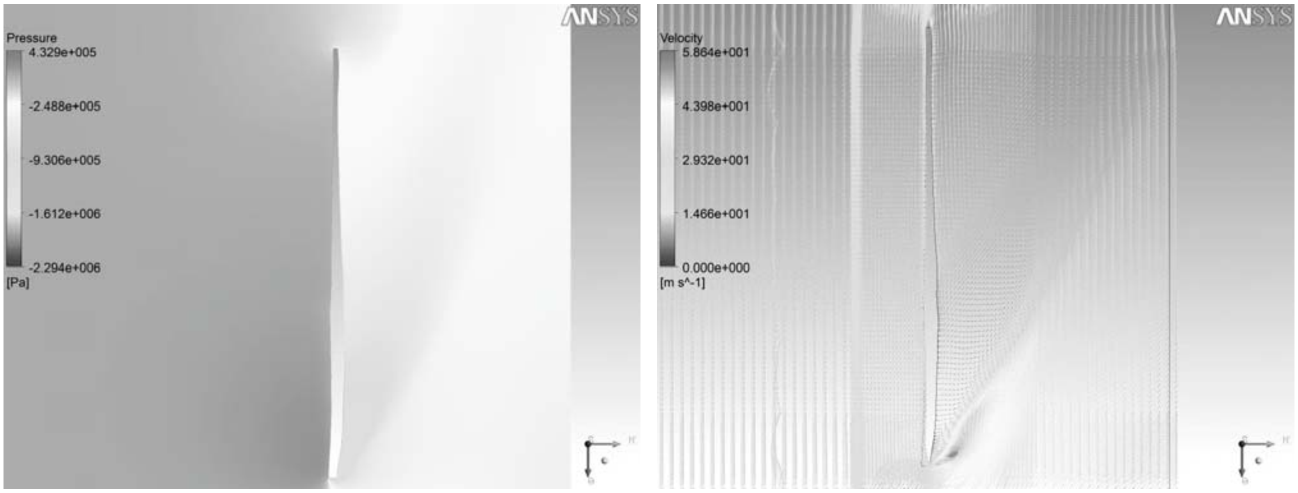

FIG. 13. PRESSURE CONTOURS AND VELOCITY VECTORS ON 97% SPAN

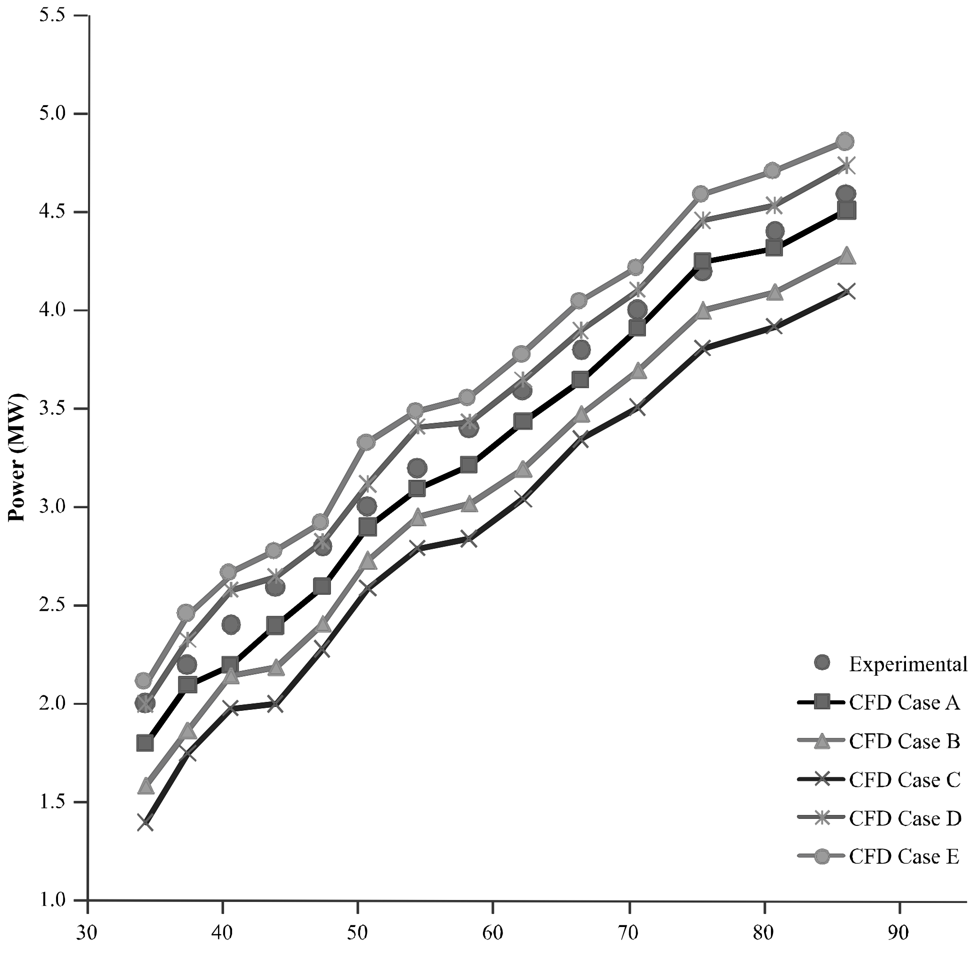

FIG. 14. COMPARISON OF EXPERIMENTAL AND CFD DATA FOR FIVE DIFFERENT CASES

Fig. 14 shows the results comparison of all five cases and it is evident that changes made in the blade geometry through orientation of blade have improved the turbine blades efficiency in terms of power output through CFD analysis. From results and comparison of all five cases, it is clearly indicated that blades used for Case-E have improved characteristics having 4.4% increases in the power output.

In a turbine blade profile, there are two sides of blade. One is pressure side and the other is suction side. Pressure side is high pressure side whereas suction side is low pressure side and due to this pressure difference, turbine blade rotates from pressure to suction side. Pressure difference of pressures between pressure and suction side depends on the profile of the blade. Better the profile, greater will be the pressure difference that will result higher pressure and efficiency of turbine. In our study as shown in Fig. 15, in Case-A pressure difference is comparatively intermediate (as colours suggest red on high pressure side while green and yellow on low pressure side) whereas Case-C with low pressure difference (one side red and on other side dominant yellow color) whereas case-E is with highest pressure difference among all cases (one side red other with dominant green color). This effect is further explained in Section 4.3 at different spans with pressure loading.

4.3 Pressure Loadings

Through CFD analysis, pressure loadings on the pressure side and suction side of blades were obtained and plotted

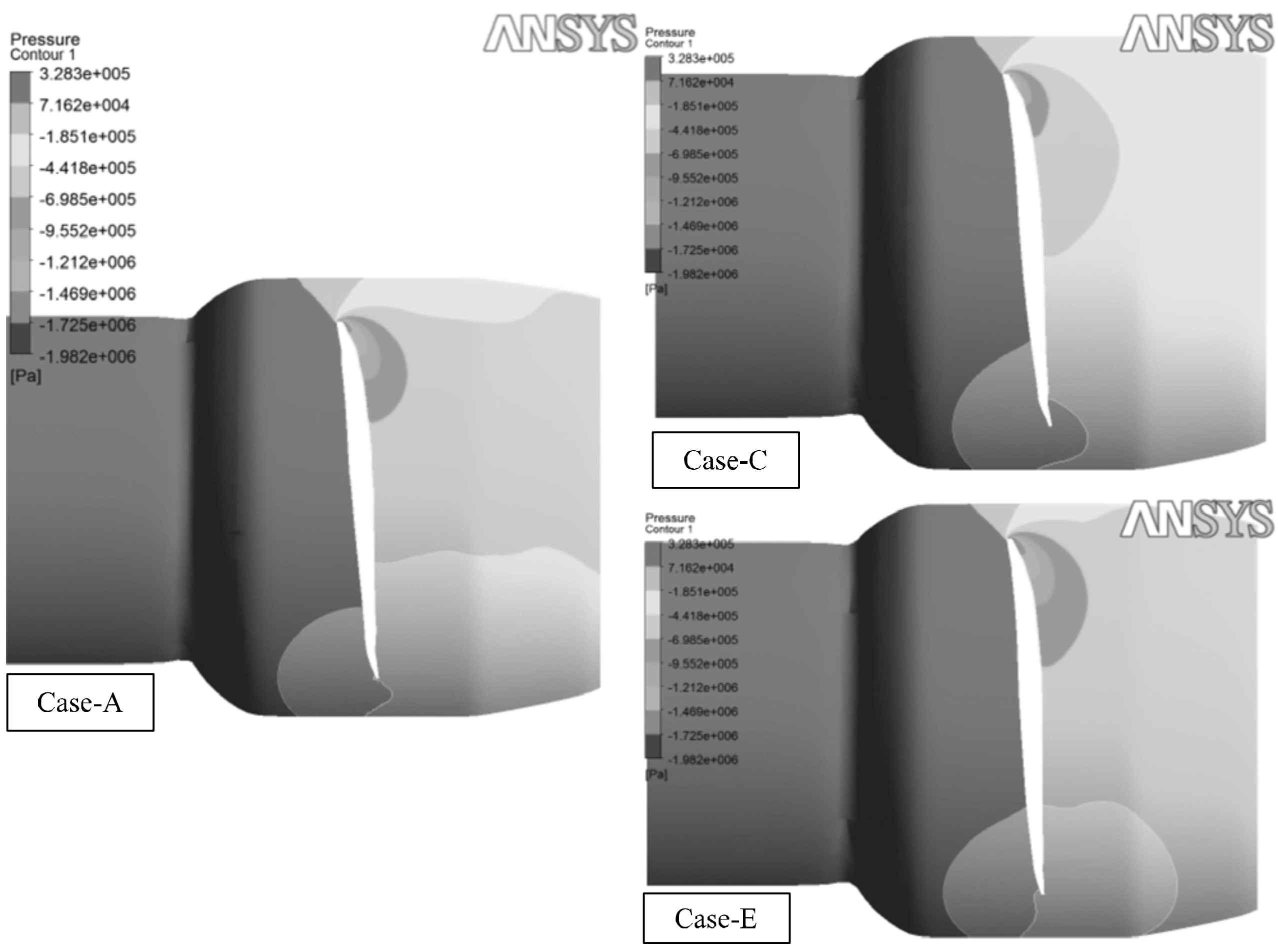

FIG. 15. PRESSURE CONTOURS AT 50% SPAN FOR CASE-A, C AND E

for Cases-A-E. Pressure loading of blade on pressure side and suction side for Case-A, Case-C and Case-E on 25, 50 and 75% span respectively are shown in Figs. 16-18. Increase in pressure difference indicates the improvement in the systems output. It is evident from the results shown that pressure difference in Case-E is much higher than compared to case-C with reference to original Case-A. This pressure difference actually impinges the blade runner to produce maximum power output. Pressure and suction sides are shown in Fig. 16.

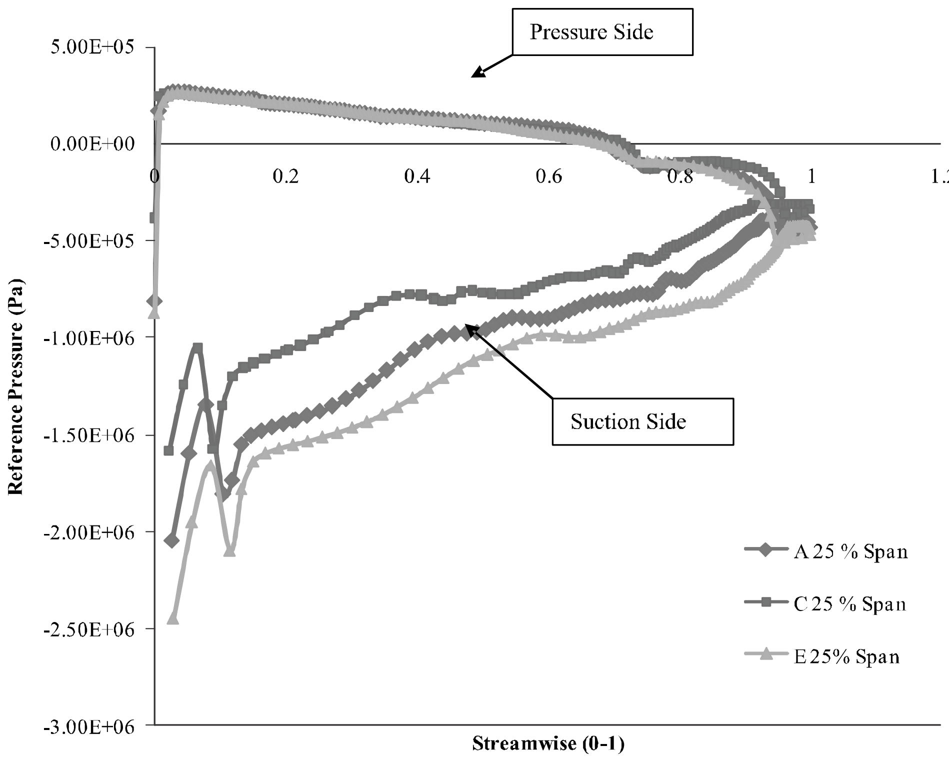

FIG. 16. PRESSURE LOADING ON BLADES AT 25% SPAN

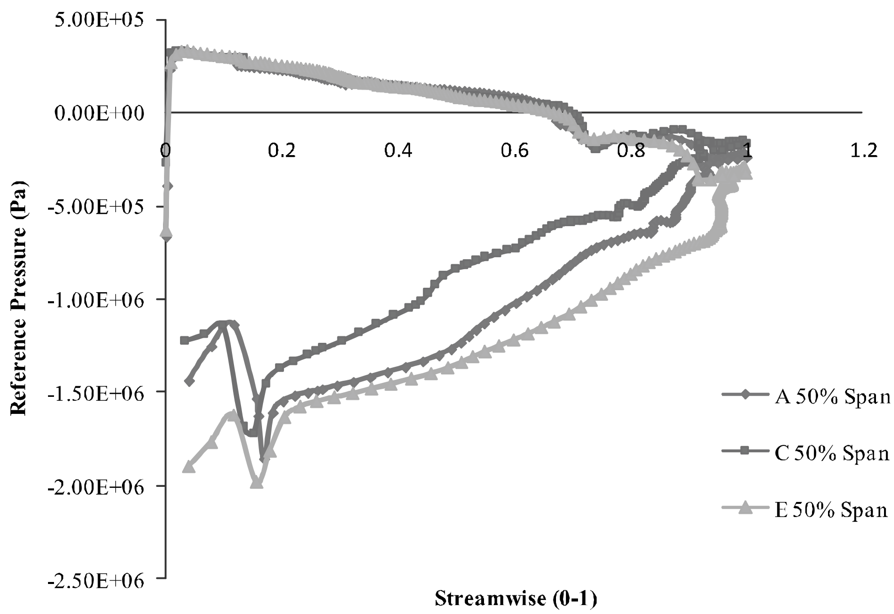

FIG. 17. PRESSURE LOADING ON BLADES AT 50% SPAN

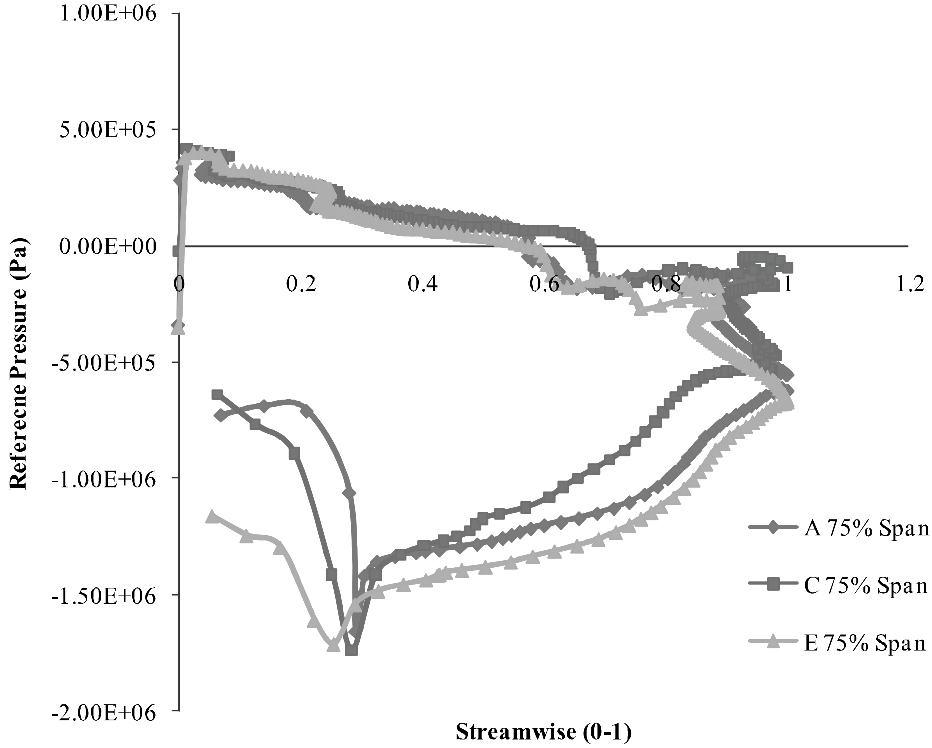

FIG. 18. PRESSURE LOADING ON BLADES AT 75% SPAN

5. CONCLUSIONS

As CFD results clearly indicate that the blade optimization through blade profile improvement can be achieved using the latest state of the art computational techniques instead of utilization of expensive tools like model/prototype testing. It is very important to note that the blade designs vary with the site data and for each new design, it is not worth to carry such expensive tools for model testing.

In our study, from results and comparison, it is clearly indicated that blades used for Case-E have improved characteristics with 4.4% increases in the power output. CFD results show that hydropower turbine unit parts like low head Kaplan turbine runner blades specially and other types of turbines in general can be optimized through the change of 3D geometries. Research optimization approach carried out can be utilized in future for the design and development of different types of hydro turbine components by using CFD.

In future research, CFD results obtained for the optimum blade profile can be used to compare with the same blade manufactured and experimentally tested. Furthermore, study on the tip clearance for the improvement in hydro turbines efficiencies can be carried out in future with the use of computational fluid dynamics technique.

ACKNOWLEDGEMENTS

The authors would like to thank University of Engineering & Technology, Taxila, Pakistan, for providing required resources to complete this project. Authors appreciate the guidance and support of Dr. Fahad Butt, Dr. Tariq Manzoor, and Mr. Atta-ur-Rehman, from HITEC University, Taxila, Pakistan.

REFERENCES

[1] Paish, O., "Small Gydro Power: Technology and Current Status", Renewable and Sustainable Energy Reviews, Volume 6, No. 6, pp. 537-556, UK, 2002.

[2] Zaigham, N.A., and Nayyer, Z.A., "Prospects of Renewable Energy Sources in Pakistan", Proceedings of Renewable-Energy Technologies and Sustainable Development, Volume 4, pp. 65-86, Pakistan, 2005.

[3] Huang, H., and Yan, Z., "Present Situation and Future Prospect of Hydropower in China", Renewable and Sustainable Energy Reviews, Volume 13, pp. 1652-1656, China, 2009.

[4] Ruchi, K., Vishnu, P., and Sushil, K., "CFD Approach for Flow Characteristics of Hydraulic Francis Turbine", International Journal of Engineering Science and Technology, Volume 2, No. 8, pp. 3824-3831, India, 2010.

[5] Celso, P., "Guide on How to Develop a Small Hydropower", The European Small Hydropower Association, Chapter 1, pp. 3-4, Belgium, 2004.

[6] Williamson, S.J., Stark, B.H., and Booker, J.D., "Low Head Pico Hydro Turbine Selection Using a Multi-Criteria Analysis", World Renewable Energy Congress, Volume 6, pp. 1377-1385, Sweden, 2011.

[7] Williams, A.A., Upadhyay, D.R., Demetriades, G.M., and Smith, N.P.A., "Low Head Pico Hydropower", World Renewable Energy Congress, Volume 6, pp. 1475-1480, Elsevier Science Ltd, Brighton, UK, 2000.

[8] Jose, M.F., Erik, R.T., Oscar, D.G., and Reynaldo, R.E., "CFD Performance Evaluation and Runner Blades Design Optimization in a Francis Turbine", ASME Conference Proceedings, Volume 1, pp. 2253-2259, Colorado, USA, 2009.

[9] Helena, M.R., and Mariana, S., "Hydrodynamic and Performance of Low Power Turbines: Conception, Modeling and Experimental Tests", International Journal of Energy and Environment, Volume 1, No. 3, pp. 431-444, Lisbon, Portugal, 2010.

[10] Peng, G., Cao, S., Ishizuka, M., and Hayama, S., "Design Optimization of Axial flow Hydraulic Turbine Runner", International Journal for Numerical Methods in Fluids, Volume 39, No. 6, pp. 533-548, USA, 2002.

[11] Santa, J.F., Blanco, J.A., Giraldo, J.E., and Toro, A., "Cavitations Erosion of Martensitic and Austenitic Stainless Steel Welded Coatings", 18th International Conference on Wear of Materials, Volume 271, Nos. 9-10, pp. 1445-1453, Colombia, 2011.

[12] Lipej, A., "Optimization Method for the Design of Axial Hydraulic Turbines", Proceedings of the Institution of Mechanical Engineers, Journal of Power and Energy, Volume 218, No. 1, pp. 43-50, UK, 2004.

[13] Prasad, V., and Khare, R., "CFD: An Effective Tool for Flow Simulation in Hydraulic Reaction Turbines", International Journal of Engineering Research and Applications, Volume 2, No. 4, pp. 1029-1035, India, 2012.

[14] Prasad, V., Gahlot, V.K., and Krishnamachar, P., "CFD Approach for Design Optimization and Validation for Axial Flow Hydraulic Turbine", Indian Journal of Engineering and Material Sciences, Volume 16, pp. 229- 236, India, 2009.

[15] Jingchun, W., Katsumasa, S., Kiyohito, T., Kazuo, N., and Joushirou, S., "CFD-Based Design Optimization for Hydro Turbines", Journal of Fluids Engineering, Volume 129, No. 2, pp. 159-168, USA, 2007.

[16] ANSYS 13.0 User's Manual Guide ANSYS Inc. Southpointe Canonsburg PA 15317, USA, 2010.