Design and Performance Investigation of a Hydraulic Mini Turbine Based on Renewable Energy Production System

Layth H. Jawad

Department of Mechanical Techniques, Kerbala Technical Institute, Al-Furat Al-Awsat Technical University, 56001 Kerbala, Iraq

laihasan@yahoo.com

Submission date:- 23/5/2019	Acceptance date:- 30/6/2019	Publication date:- 17/7/2019

Abstract

Nowadays great interest of renewable energy production systems since last few years. The renewable energy sources such as a hydro, solar and wind have been rapidly growing, especially the hydropower energy due to the increasing of predicted scarcity of fossil fuels and the environmental issues. Kaplan turbine is a reaction type of hydropower which is one of the primary sources of renewable energy. In this study, the aim of this work is to design and performance investigation of a Kaplan turbine runner for determining the power output and efficiency based on different configurations of turbine wheel. The main characteristics of a micro Kaplan turbine primary proposed design configurations was performed to increase the use of renewable power in rural areas and reduce the dependence on fossil fuels. The computational fluids dynamics (CFD) was used to simulate the proposed design for further analysed and improve both the power output and efficiency using ANSYS CFX17.1.The performance of a micro Kaplan turbine has been predicted such as power output and efficiency. Furthermore, the effect of blades number showed a significant influence on the turbine's power output and efficiency. Generally, improvement of proposed design configurations are presented for understanding the flow field in a Kaplan turbine runner.

Keywords: CFD; Hydraulic Micro Kaplan Turbine; Renewable Energy; Power Output; Efficiency

1. Introduction

Hydropower is a green, clear, and sustainable source of renewable energy that produces cheaper electricity and lowers emissions it is generated mainly from hydroelectric dams. The most primarily available renewable power source and most efficient of hydropower to produce electricity because of its high energy [1]. Install of hydraulic turbines have to be suitable in the hydroelectric power plants, depending on the discharge of the site and head, In order to obtain greater efficiency and power output. Commonly the hydraulic turbines available categorised into two type's impulse and reaction turbines. The reaction hydraulic turbine in which of some of amount available hydraulic energy of water was converted into kinetic energy before it reaches to the runner of the turbine. While in case of all hydraulic energy converted into kinetic energy before the water strikes on the runner of the turbine was called impulse turbine, table 1 show that the properties of different turbine types [2].

Table 1 Classification of the turbines

Type	Head	Flow Rate	Specific Speed
Pelton Turbine	High	Low	Low
Francis Turbine	Medium	Medium	Medium
Kaplan Turbine	Low	High	High

Figure 1 shows the low head hydropower plant of Kaplan turbine that the water comes through scroll casing into guide vanes in the radial direction in which the flow make a right angle turn and axially enters the runner. Kaplan turbine is applicable for a low head range of (2-4) m [3].

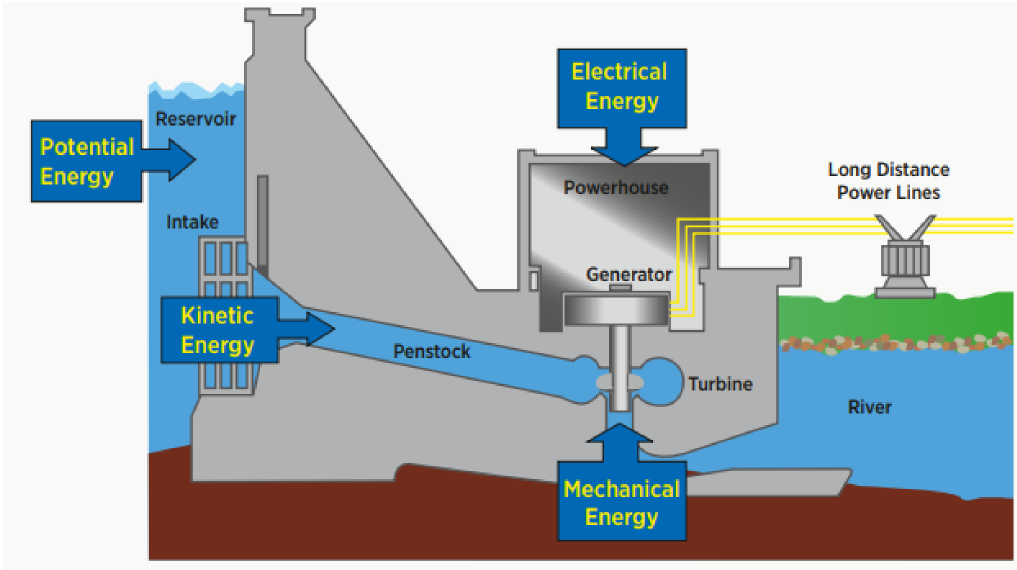


Figure 1 Low head hydropower plant.

Generally, the hydropower schemes classified to cover abroad range of power output as in table 2. Pico hydro scheme is a smallest scale may not require any damming and require water as run of river which generate no more than a few kilowatts. Supply high levels of demand Greater amount of power generation connected to the grid require a larger scale of turbine which it require damming to regulate the water flow and create storage capacity [4].

Table 2 Power output classification of hydropower scheme

Classification	Power output
large	>100 MW
medium	10-100 MW
small	1-10 MW
mini	100kW-1 MW
Micro	5-100 KW
Pico	< 5 KW

Development of design method numerically was presented for axial hydraulic turbine with very low head and validated with experimental data. The results show that the hydraulic efficiency was affected by rotational turbine speed and runner blade angle [5].Computer aided flow visualization was used to study the Cavitation Phenomena on the Kaplan runner blade for a modified turbine. Results showed that cavitation phenomena was reduced, power output and performance were increased for the modified Kaplan turbine [6]. Development of new software applied for feasibility studies of a micro hydraulic turbine performance in Cameroon. The application of that software for a micro hydro power plant was successfully to estimate the demand of power [7].

Many publications in last years dedicated on hydropower plants because of easy to design and minimum cost for fabrication turbines [8].In rural and hilly areas the power generation has a rapid growth of hydro power turbines thus its connect to a grid supply as a demand requirements for different load [9, 10].Therefore, the Hydropower turbines require further study to achieve highest power and efficiency. Hence, it is important to optimize the runner blade design to extract more water energy for improving power generation [11]. Nowadays, development of producing electricity by using hydropower plant the most suitable in rural area, it is effective and cheap [12]. Computational fluid dynamic was used to simulate the flow on blade of a hydro turbine, the results showed the efficiency was affected by pressure distribution on blade turbine [13, 14]. Suitable hydro turbine was designated depending on non dimensional design parameters [15].

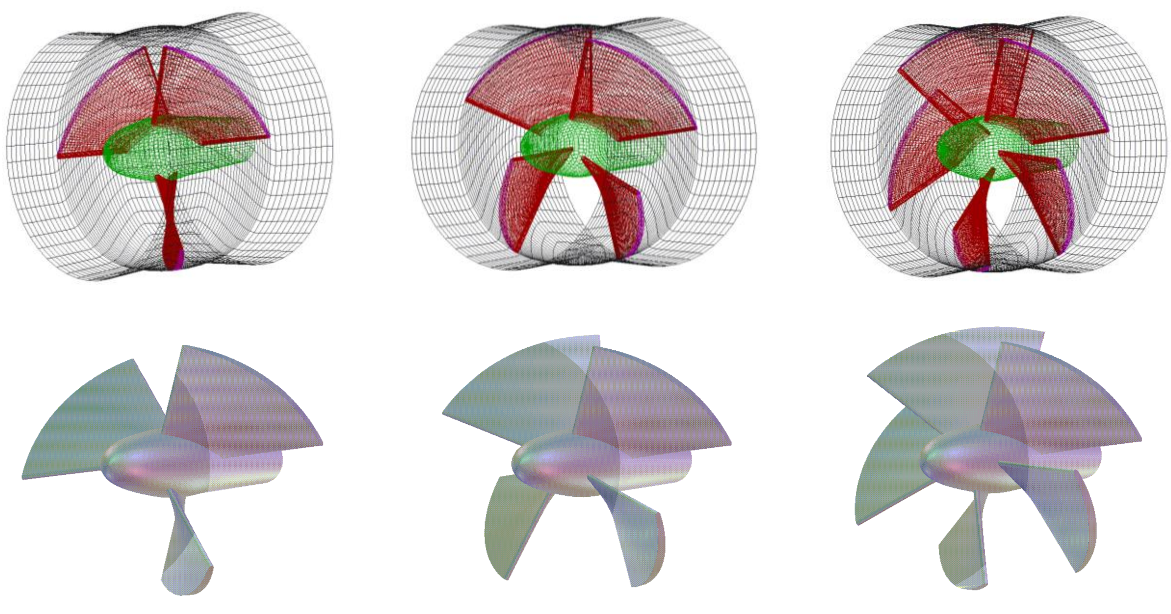
In this paper, the design of a micro hydraulic turbine was made to work in a small amount of power to provide energy for a number of houses. Numerical investigation characteristics of flow through low head Kaplan turbine runner was carried out to predict and improve the overall performance of a mini hydro turbine.

2. Specifications & Design

The supposed main geometry dimensions of the micro Kaplan turbine types for different blade number are given in Table 3. Figures 2 shows the design geometry was created using (ANSYS BladeGen) The variation of the beta angle (β) along the runner was established by a spline curve for every layer (hub, mid, and shroud) that can be configured and modified in the program .The blade profile chosen for runner was a general NACA (National Advisory Committee for Aeronautics) profile. The turbine studied was an axial turbine without inlet guide vane stage model. The CFD computations for the design were performed on the geometries. Initial Graphics Exchange Specification (IGES) parts, were defined via ANSYS BladeGen for all geometry surfaces such as inlet, exit, and periodic boundaries.

Table 3: Axial Kaplan Turbine Features by ANSYS BladeGen.

Radial Turbine Dimensions	Runner
Inner Radius	112.5 mm
Outer Radius	450 mm
Shroud Tip Clearance (% span)	95%



3 Blades **4 Blades** **5 Blades**

Figure 2: Types of micro kaplan turbine runner

3. Methodology of Computational Fluid Dynamic

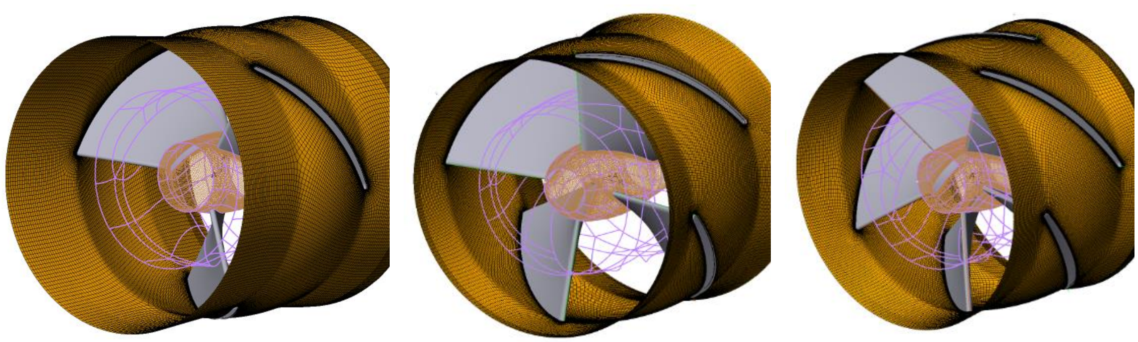
3.1 Mesh Generation

Numerical study was carried out to simulate the extracted fluid domain with the blade in the middle for runner passage by using computational fluid dynamics Software (ANSYS CFX 17.1). The finite element hexahedral mesh was generated on the runner blade surface mesh and it's consisted of eight nodes. The structured hexahedral elements for computational grid Runner blade passage were generated using ANSYS Turbogrid in multi-block environment. Sufficient fine mesh were created around the runner blade, clearance tip, shroud, and hub walls with acceptable limits of mesh angle, Table 3 shows the mesh data of water for runner passage.

Table 3 Mesh Data of Water

No. of Blades	No. of Nodes	No. of Elements
3	**672706**	**637316**
4	**622668**	**588802**
5	**599831**	**566748**

The accuracy of solution and convergence depends on grid size, initially a coarse mesh was used to study the boundary conditions and solver settings. In general, the finer mesh size is a compromise between the accuracy and computational time of results. Hence, a study of grid independence test was carried out to ensure the accuracy of numerical solution. Therefore, a fine element size was reported in this paper for the computational fluid dynamic solutions, figure 3 shows the computational domain with surface grid.

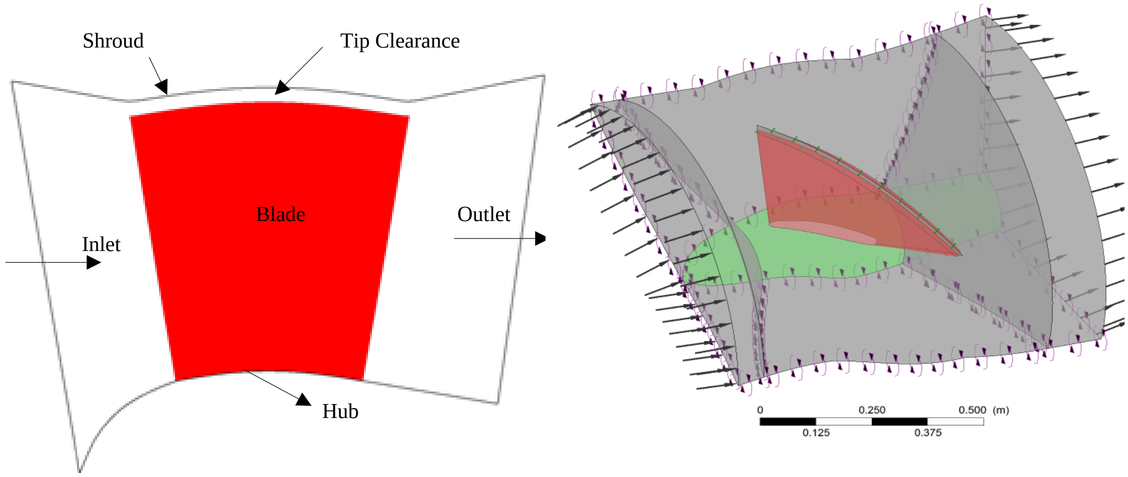


3 Blades **4 Blades** **5 Blades**

Figure 3: Computational domain with surface grid

3.2 Fluid Flow Modelling

The performance prediction of the designed axial Kaplan turbine runner blade configurations has been analysed using 3D flow, steady numerical simulations, and k-ω-SST turbulence model of the full water passage. The flow domain of fluid includes the stationary parts (inlet, outlet, and shroud), the rotating parts (hub and runner blade) and the periodic parts are defined for the right and left boundaries as in figure 4 shows the computational flow as in meridional plane and fluid domain. The flow rate was applied to inlet boundary condition and the static pressure was applied to outlet boundary condition.



(a) Meridional Plane **(b) Flow Domain**

Figure 4 Computational fluid flow domain

4. Results and Discussion

The axial flow Kaplan turbine performance has been discussed in detail to identify the optimum design. The results of performance parameters were assessed for different design such as blade number, flow rate, and turbine speed. The numerical results compared with other researchers experimental results. There is very good agreement of the computed results with available literatures results. Convergence of the simulation is assumed when the residuals of mass, momentum and turbulence reaches to less than 1 x 10 -5 .The performance prediction has been computed at constant dimensions of different axial Kaplan turbines runner with variable flow rate. Figure 5 shows that the output power increase with increasing the flow rate at constant rotational speed (300 rpm) for different blade number of Kaplan turbine runner, the highest output power values can be seen for 5 and 4 blade runner turbine and smallest values for 3 blade runner turbine .Figure 6 shows the comparison of turbine efficiency curves with flow rate for different blade number. The efficiency of the runner turbine increases as seen the high efficiency for 3 blade runner turbine due higher loss pressure. The computational results shows that the optimum turbine efficiency and output power are obtained for 4 blade turbine runner.

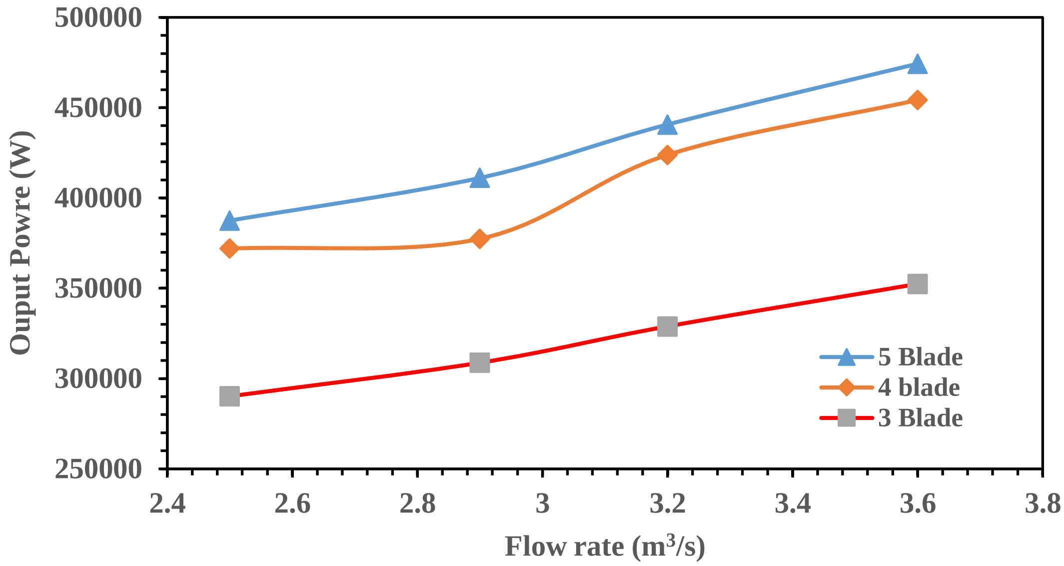


Figure 5 Output power of hydro turbine with various flow rate for different blade number

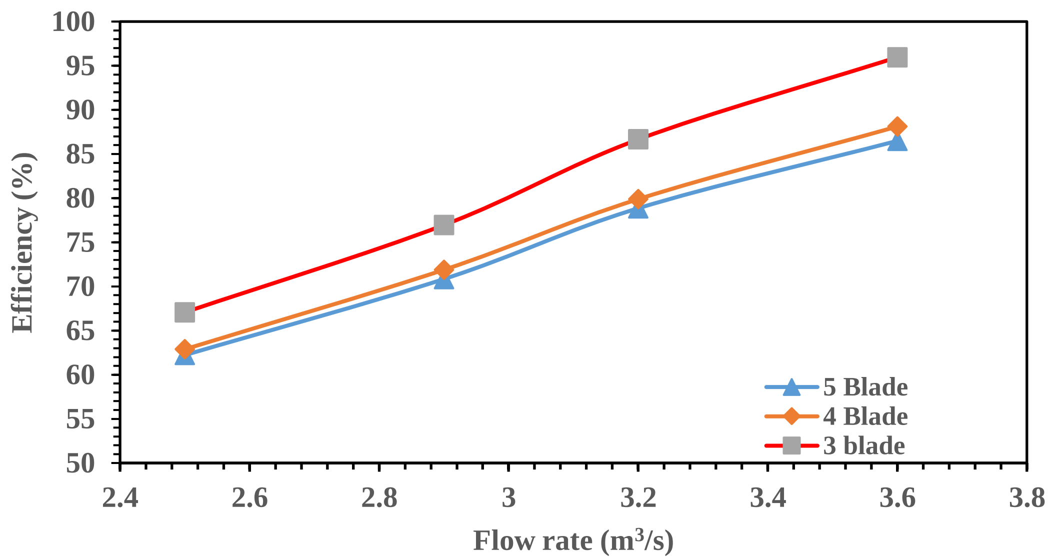
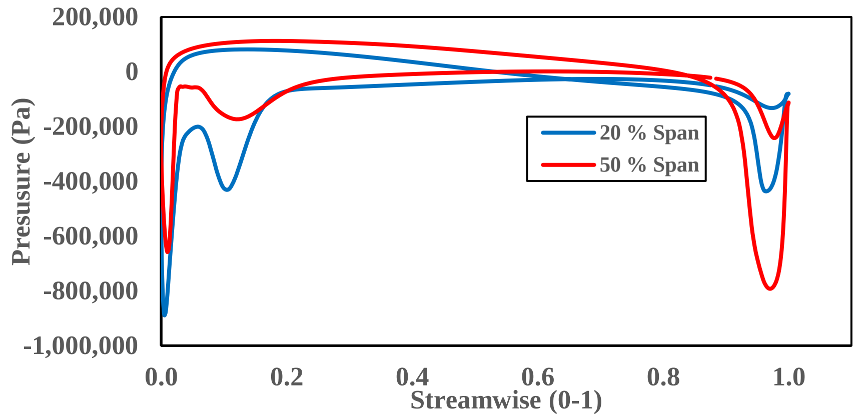
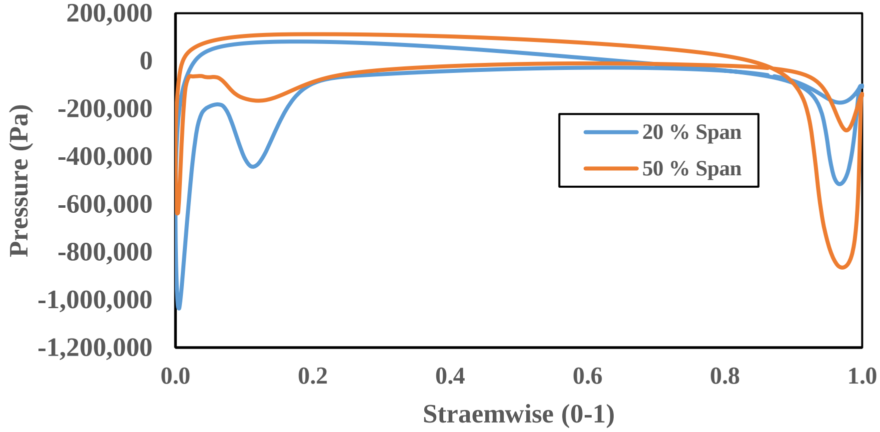


Figure 6 Efficiency of hydro turbine with various flow rate for different blade number

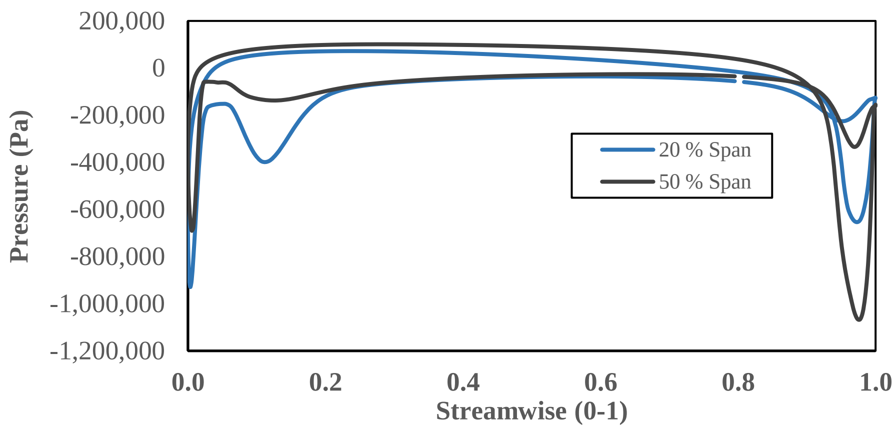
Figure 7 shows the runner blade loading throughout the runner passage at 20 % and 50 % for each design. In order to study blade loading nearby trailing edge to ensure that the flow direction not forced reverse. The reverse direction flow can be seen at trailing edge due to the pressure on the pressure surface becomes smaller than that on suction surface. Therefore, the reversal flow can be avoided in design of runner blade to reduce the losses turbine performance.



(a) Five blades



(b) Four blades



(c) Three blades

Figure 7 Runner blade pressure loading

Figures 8, 9, and 10 show the pressure and velocity vectors contour at constant span 50 % for the runner turbine 3, 4, and 5 blade as an example of computational fluid dynamic results. At the design point no separation flow can be seen which a major source of losses. The velocity vectors of fluid flow through the runner turbine blade passage becomes uniform and normal behaviour. It is clearly seen that there was some of backflow vortex at leading edge and trailing edge of blade at suction surface for different configuration of turbine blade which was important factor to modify the design in order to optimise the blade geometry. In addition, the effect of pressure contours formation on the mid span runner blade at suction surface indicates negative pressure values at trailing edge of the runner blade due to dynamic action of the blade for all designed types. Cavitation take place when the vapour pressure higher than local static pressure, it can be seen an occurrence of cavitation in contours plot.

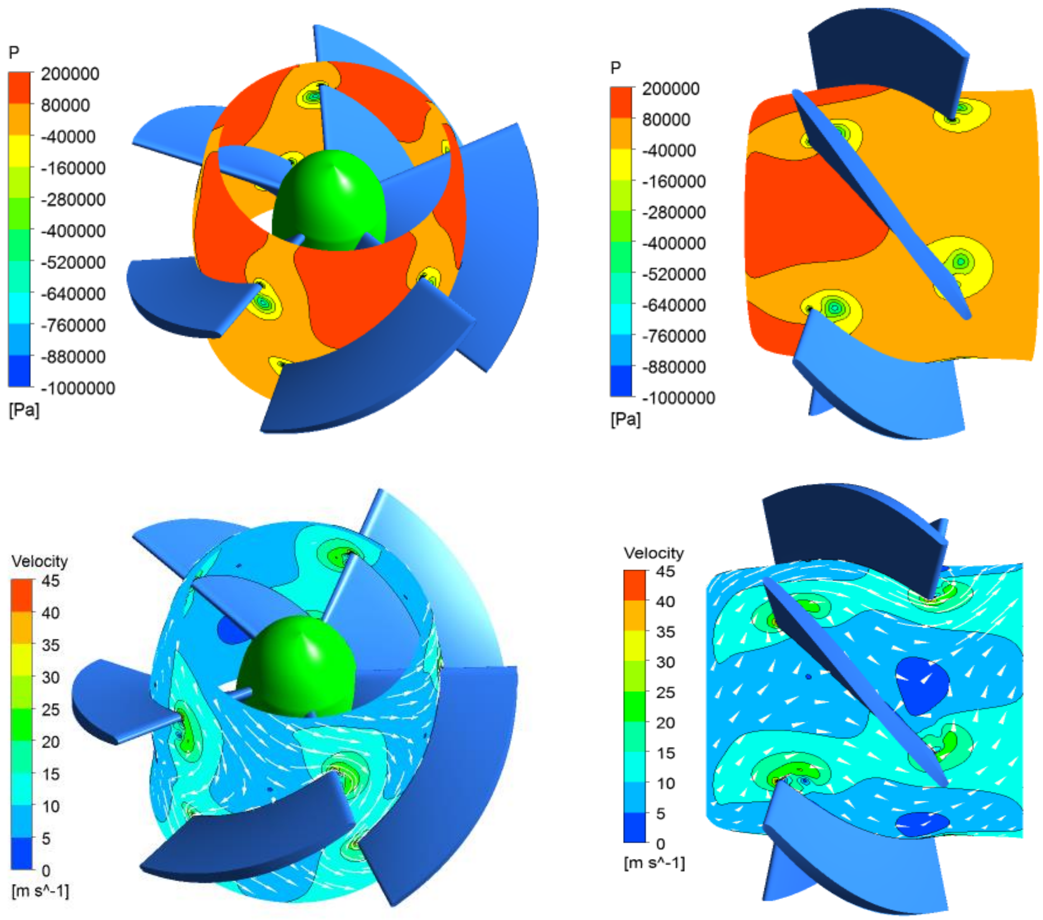


Figure 8 Pressure and velocity contour for 5 blade runner turbine

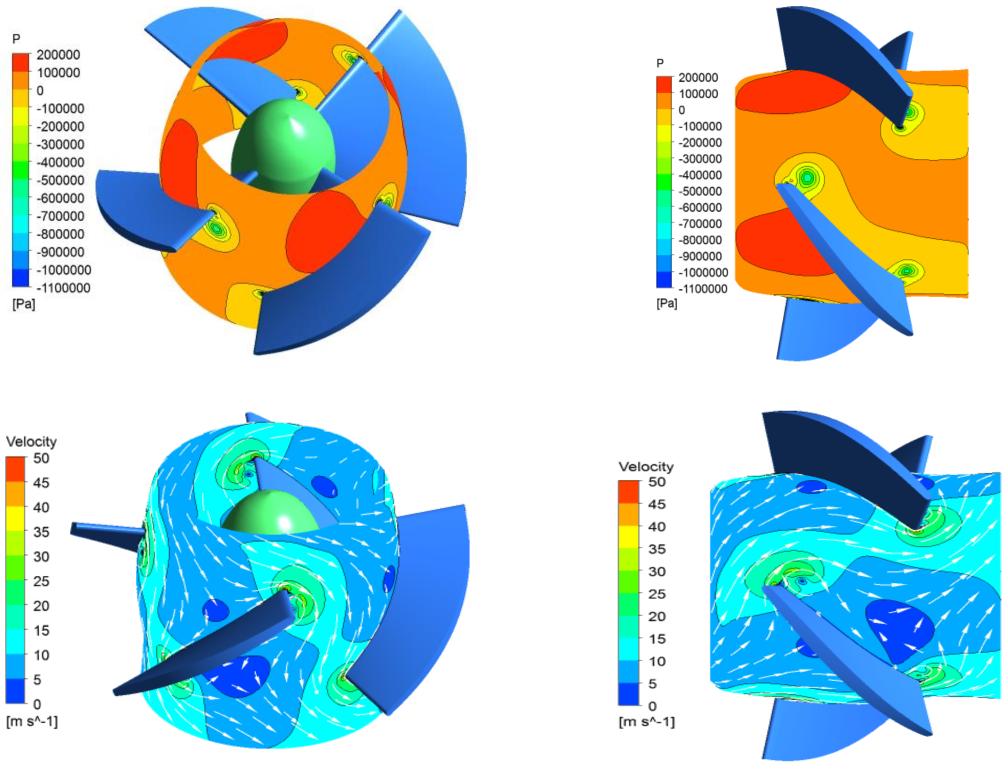


Figure 9 Pressure and velocity contour for 4 blade runner turbine

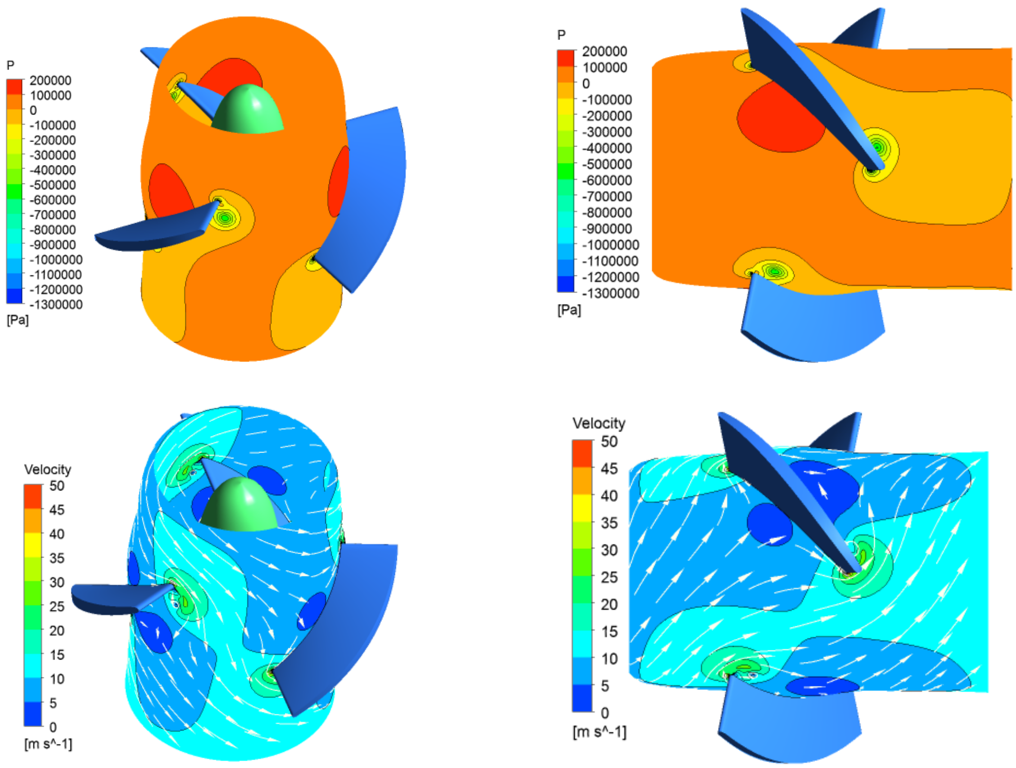


Figure 10 Pressure and velocity contour for 3 blade runner turbine

5. Conclusions

Global energy demand would be significantly increased on hydropower as a source of renewable energy to reduce the fossil fuels which has impact on the environment. Any hydropower requires more and more development of turbine design which is the most significant part of hydro plant, in order to study the hydrodynamic of water flows behavior. These days emerging computational fluid dynamics codes are widely being used to make design of turbine a step ahead to predict the output power, efficiency and performance parameters. This study presented interesting and important numerical results of hydraulics flow within a turbine runner under different blade number. Achievement wide ranges of output power and efficiency were in the 4 blade case, hence it is recommended. The slightly highest efficiency and smallest output power were in the 3 blade case, the efficiency in 5 blade case was slightly lower than that of 4 blade case due to when blade number increased leads to increase the friction. However, increasing number of blades is always not possible and there should be a maximum possible number of blades for a particular size of a runner. The numerical results show the distribution pattern for pressure and velocity contours variation plots between hub to shroud, power output and efficiency affected by the flow rate. Hence, it is concluded that the numerical codes method can be used to optimize the design parameters with less time and low cost in order to study the flow regime within turbine runner. Finally, the performance of a Kaplan hydraulic turbine has been predicted by numerical simulation with minimum time and money.

Conflicts of Interest

The author declares that they have no conflicts of interest.

References

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

[2] Pati, S. Fluid Mechanics and Hydraulic Machines. New Delhi: Tata McGrow Education Private Limited, 2013.

[3] Timo Flashopler. Design of the runner of a Kaplan Turbine for small hydroelectric power plants. Tampere university of Applied Sciences, pp. 21-40, 2007.

[4] Arthur Williams and Stephen Porter. Comparison of hydropower options for developing countries with regard to the environmental, social and economic aspects. Proceedings of the International Conference on Renewable Energy for Developing Countries, 2006.

[5] M.H.Sotoude, HaghighiS.M.MirghavamiS.F.ChiniA.Riasi. Developing a method to design and simulation of a very low head axial turbine with adjustable rotor blades". Renewable Energy, Vol. 135, Pp.266-276, 2019.

[6] Andrej Podnar, Matevž Dular, Brane Širok and Marko Hočevar (2019) Experimental Analysis of Cavitation Phenomena on Kaplan Turbine Blades Using Flow Visualization**.** Journal of Fluids Engineering, V. 141 Issue 7, 2019.

[7] Elie Bertrand Kengne, and etal. Performance in Feasibility Studies of Micro Hydro Power Plants. New Software Development and Application Cases in Cameroon. Energy Procedia, Vol. 157, pp. 1391-1403, 2019.

[8] Shahram Derakhshan, Ahmad Nourbakhs. Experimental Study of Characteristic Curves of Centrifugal Pumps Working as Turbines in Different Specific Speeds. Experimental Thermal and Fluid Science. Vol.32, pp. 800–807, 2008.

[9] O. Paish. Micro-hydropower: Status and Prospects. Proceedings of the Institution of Mechanical Engineers, Part A. Journal of Power and Energy, 2002.

[10] John Twidell, Tony Weir. 2006. Renewable Energy Resources. 2nd Edition by Taylor and Francis, Newyork, USA.

[11] S. L. Dixon. Fluid Mechanics and Thermodynamics of Turbomachinery. 5th Edition, Elsevier Butterworth–Heinemann, 2005.

[12] A. H. Elbatran and etal. Hydro Power and Turbine Systems Reviews. Jurnal Teknologi (Sciences & Engineering), pp. 83–90, 2015.

[13] Weerapon Nuantong. Flow Simulations on Blades of Hydro Turbine. International Journal of Renewable Energy, Vol. 4, No. 2, 2009.

[14] L. H. Jawad and etal. Numerical study on the effect of interaction vaned diffuser with impeller on the performance of a modified centrifugal compressor. *Journal of Mechanics*, vol. 30, pp. 113-121, 2014.

[15] Dixon, S. L. Fluid Mechanics, Thermodynamics of Turbomachinery. Elsevier, Oxford, UK, Chapters, 2005.

فحص تصميم وأداء توربين هيدروليكي صغير المستندة إلى نظام إنتاج الطاقة المتجددة

ليث حسن جواد

قسم التقنيات الميكانيكية، معهد كربلاء التقني، جامعة الفرات الأوسط التقنية، 56001 كربلاء، العراق

laihasan@yahoo.com

الخلاصة

في الوقت الحاضر اهتمام كبيرة لانظمة إنتاج الطاقة المتجددة منذ السنوات القليلة الماضية. شهدت مصادر الطاقة المتجددة مثل الطاقة المائية والطاقة الشمسية وطاقة الرياح نمواً سريعاً، لا سيما الطاقة الكهرومائية بسبب تزايد ندرة الوقود الأحفوري والقضايا البيئية. توربين كابلان هو أحد انواع انتاج الطاقة الكهرومائية ومن المصادر الرئيسية للطاقة المتجددة. في هذه الدراسة، يهدف هذا العمل إلى تصميم وفحص أداء عجلة توربين كابلان لتحديد ناتج الطاقة وكفاءتها استنادًا إلى الانواع المختلفة لعجلة التوربينات. الخصائص الرئيسية الأساسية للتصاميم المقترحة لتوربين كابلان الصغير اجريت وذلك لزيادة استخدام الطاقة المتجددة في المناطق الريفية وتقليل الاعتماد على الوقود الأحفوري. تم استخدام ديناميك الموائع الحسابية (CFD) لمحاكاة التصاميم المقترحة للتعرف على مزيد من التحليل وتحسين كلاً من القدرة الخارجة والكفاءة باستخدام ANSYS CFX17.1. وقد تم التنبؤ بأداء توربين كابلان الصغير مثل القدرة الخارجة والكفاءة. علاوة على ذلك، أظهرت النتائج ان عدد الريش يؤثر بشكل كبير على اداء التوربينات وكفاءتها. بشكل عام قدمت تحسينات للتصاميم المقترحة وذلك لفهم مجال الجريان داخل عجلة توربين كابلان.

الكلمات الدالة: ديناميك الموائع الحسابية، توربين كابلان هيدروليكي صغير، الطاقة المتجددة، القدرة الخارجة، الكفاءة.