Acta Mechanica Slovaca **28 (4): 6 - 12,** December **2024**
https://doi.org/10.21496/ams.2024.026

Acta Mechanica Slovaca
ISSN 1335-2393
www.actamechanica.sk

Experimental and Numerical Investigation on Blade Sweep to Improve Performance of an OWC Turbine

S. M. K. Benslimane [1], K. Boualem [2], T. Yahiaoui [3,*], Antonio Mañino Ferrando [4], María Clavero Gilabert [5], Nemdili Ali[6]

[1] Laboratory of Maritime Science and Engineering (LSIM), Faculty of Mechanical Engineering University of Sciences and technology of Oran, Mohamed Boudiaf, Algeria
[2] University of Relizane, Department of Mechanics, Algeria
[3] Laboratory of Aeronautics and Propulsive Systems, Mechanical Engineering Faculty, USTO Oran, BP 1505 El Mnouar, Algeria
[4] University of Granada, Department of Structural Mechanics and Hydraulic Engineering, Spain, Granada
[5] University of Granada, Granada, department of Civil Engineering, Spain, Granada
[6] Laboratory of Rheology, Transport and Treatment of Complex Fluids, Hydraulic Department, USTO Oran, BP 1505 El Mnouar, Algeria

Abstract: The energy of sea waves is one of the renewable energies. The wells turbine uses the airflow produced by the pressure change inside the oscillating water column. A visible radial drift from the main flow occurs downstream of the rotor is noticed in the turbines wells with a constant chord length of the blade. Indeed, a blade chord linearly increasing with a radius is proposed in order to minimize this phenomenon and to increase the local coefficient of lift of the blade. In this context an investigated experimental and computational method are carried out to analyze the performances of two types of Wells turbines, one with a constant chord blade (rectangular-shaped blade) and the other with the blade's agreement increasing linearly (Swept blade). The results obtained show an improvement in the performance of the Wells turbine with a swept blade.

Keywords: energy conversion, Wells turbine blade., performance, NACA0018, blade with a rectangular plan form, swept.

Nomenclature

C	Blade chord length, (mm)
C_T	Torque coefficient
Rh	Hub Radius, (mm)
C_A	Input coefficient
Rt	Tip Radius, (mm)
Φ	Flow coefficient
Z	Number of blades
ρ	Density of air
S	Turbine solidity
V_A	Inlet axial velocity, (m/s)
Δp^*	Pressure drop across the rotor
U_t	Peripheral or tangential velocity (m/s)
Δp_0	The stagnation pressure drop across the turbine, (Pascal)
ω	Rotor angular velocity, (rd/s)
RPM	Rotational speed (Revolutions Per Minutes)
b	Blade span, (mm)
C	Blade chord and the medium blade chord in sweep blade, (m)

*** Corresponding author:** Tayeb Yahiaoui, ***E-mail address:*** *yahiaoui_tayeb@yahoo.fr*

η Turbine efficiency, (%)
T Torque force on shaft, (N.m)
Q Volume flow rate, (m^3/s)
α Attack flow angle, (rd)

1. Introduction

The burning of fuels has a great influence on the environment which obliges us to use renewable energies. One of the high potential renewable energy sources is certainly energy from sea waves and could be exploited by coastal countries [1]. According to recently published studies, ocean waves can generate approximately 30,000 TWh/year, sufficient to meet the global electricity demand if harnessed effectively [2]. The hybrid wells turbine's performance was improved with the application of passive flow control technology [3]. The hydrodynamic and conversion performance of a dual cylindrical OWC integrated within a caisson-type breakwater was investigated numerically [4].

Interesting devices designed for the extraction of wave energy are those based on the oscillating water column (OWC) principle [5], [6],.The converter is the Wells turbine, introduced by Wells DR.AA in 1976 [7],is a self-rectifying air turbine, with a constant blade chord length of symmetrical cross-section, arranged in a 90° stagger angle cascade, and has the specificity to turn in the same direction, when flow take two opposite directions. Thus, the mechanical energy produce, is converted into electrical energy with generator at the end of the device. [8], [9].

Several versions of the Wells turbine were carried out to improve the performance [10], [11], e.g., the contra-rotating, the biplane, the variable pitch, and with guide vanes, [12], [13], [14]. Other parameters such as humidity, temperature, and density affect a performance of Wells turbine [15], in real operating conditions, which were studied by some authors, such [13], [22-24].

Most of these research has used a turbine with air foil blades NACA 00XX with rectangular plan form.

Experimental investigations, and CFD analysis of the flow-field through the conventional Wells turbine blades have shown a stall zone at tip blade and becomes important when the flow increases, which significantly affects the local torque and therefore performance, [14],

In order to Improve the efficiency peak and stall characteristics of Wells turbine several prototypes of blades have been proposed, such as, the use of blades with two different types of profiles, one profile from the hub-to-mean radius, and another one from the mean to tip, [15, 16], and the swept tip blade on one side, [17], and on both sides, and also blade profile Increases linearly with the radius [18, 19, 20].

It is noted that visualization of the flow-field on the blades with a swept form showed a great influence on the separation of the boundary layer on the blades.

The distribution of the flow pressure around the modified blade, changes significantly at tip. The sweep of blade turbine is a promising design factor for improved performance, [21 and 5]. This study presents an experimental and numerical investigation showing the effect of sweeping the blade of a Wells turbine on the aerodynamic performance. The turbine consists of a hub of 192 mm in diameter, designed for fixing once, eight removable blades with a constant chord (blade with a rectangular plan form), and a second time, eight blades whose chord increases linearly with the radius (swept blade) In both configurations, diameter of turbine is 0.288 mm, and profile is NACA 0018.

Tests for unidirectional flow were performed, including the velocity at the input, and the output of the device, the circumferential pressure at the entrance, level, and output of the dispositive, and finally measuring the rotational velocity of the turbine. The power output is predicted numerically, by solving the 3-D Navier-Stokes equations with computational Fluid Dynamics (CFD) using a commercial Fluent Code, but they did not cover the stall condition and detailed simulation of the performance.

2. Materials and Methods

A combined system of the experimental and numerical simulation has been developed to carry this study. In order to analyze the performance of new design turbine, two types of blades based on the American Symmetric NACA 0018 profile changing the length of the cord was tested in the wind tunnel. While, the difficulty of finding a suitable coupler for the new design of the turbine, it was appropriate to numerically calculate the power produced, observing the experimental model, the boundary conditions and the RPMs resulting from the tests.

2.1. Turbine design

The turbine experimental model used for tests is presented in Fig.1, mainly consists of a hub whose diameter is 192 mm on which will set perpendicular and around its axis 08 removable blades.

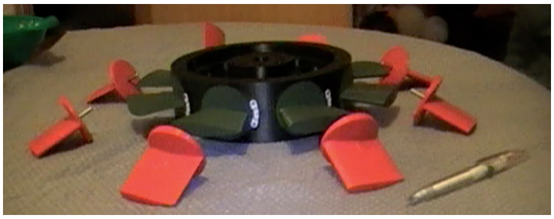

Figure 1: Turbine design

The fig.2 show two different blades form used in experimental study, first blade with rectangular plan (a) and second with a linear Swept (b) angles of λ = 29.20°.

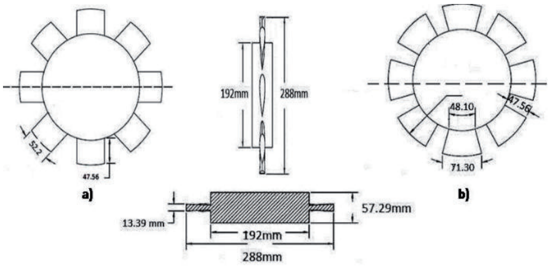

Figure 2: Geometric description of the turbine

The table 1 exposes the geometric characteristics of both blades:

Table 1: Geometric characteristics of the turbine

(a)	(b)
Wells Turbine with Rectangular Plan Form Blades	**Wells Turbine with Sweep Blades**
Hub Radius Rh = 96 mm	Hub Radius Rh = 96 mm
Tip Radius Rt = 144 mm	Tip Radius Rt = 144 mm
Hub-Tip Ratio Rh/Rt = 0.67	Hub-Tip Ratio Rh/Rt = 0.67
Number of Blades Z = 8	Number of Blades Z = 8
NACA 0018	NACA 0018
With constant chord length c = 52.20 mm	With sweep chord length c (mid) = 59.7mm
Solidity S = 0.55	Solidity S = 0.63

2.2. Experimental setup

Tests are conducted with the same conditions on the flow velocities generated by the wind tunnel of the IISTA Fluid Dynamics Laboratory (University of Granada). The wind tunnel has a test section 2.15 - 1.8 - 15 meters (width, height, length), the wind speed, up to 200 km/h, is controlled by frequency converter controlled by automaton, the power installed is 160 kW.

As shown in Fig. 3, the turbine was fitted within a uniform duct section, which had a length of 160cm in the flow direction. However, the system was mounted using standard ball type bearings. The flow duct was tapped with ports for pressure measurements at 03 locations in the axial direction, at each of which there were up to 04 ports equally distributed around the circumference of the duct. The circumferential ports were physically connected by air conduits, and were therefore combined into a single pressure signal, at each axial location.

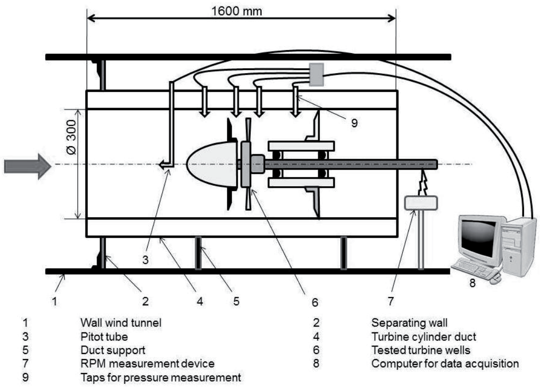

Figure 3: Device in Wind Tunnel

Indeed, a pressure scanner (i.e. piezo-resistive sensor) has been used over study area in order to characterize the pressure field (Fig.4) and to transform this pressure value into an electrical signal on a computer. Moreover, a Pitot-tube velocity measurement device was positioned just inside the hub of turbine.

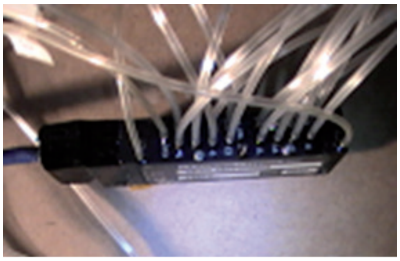

Figure 4: Pressure scanner connected to the taps

The tachometer is setting at end of device, to achieve the revolutions per minutes (RPM). We note, experimental works are carried out for a range of air inlet flow velocities and the normal experimental procedure involved establishing a steady incident air flow velocity.

2.3. Numerical method setup

A combined Fluent-Gambit models system is considered to calculate the power output.

The Geometry of the numerical model in three dimensional, is made according to the experimental model presented in Table 1. Moreover, the condition of the asymmetry of the turbine is used to simplify the mechanisms and allows us to mesh only a quarter of the overall domain of the turbine.

The boundaries conditions used are displayed in Fig. 6. The power output of turbine is got from the numerical simulation by solving the incompressible Navier-Stokes equations in a non-inertial reference frame rotating with the turbine using FLUENT 6.3.26.

The determination of the torque generated by the rotor of the turbine is directly related to the tangential forces acting on the blades of the turbine, an illustration of the force vectors is shown in Fig.5.

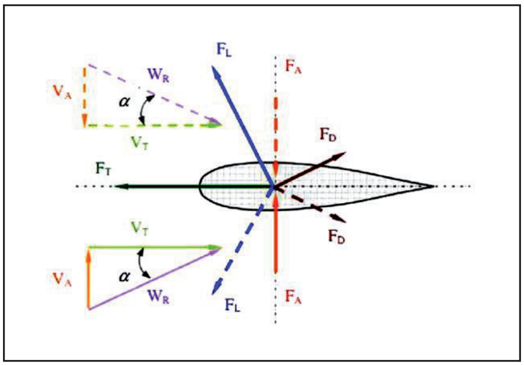

Figure 5: Forces acting on blades Turbine

Periodic boundary conditions are imposed on the neighbouring surfaces of the turbine. The model includes three domains. The "Inlet" and "Outlet" domains are stationary while the "Rotor" domain rotates with a certain angular velocity (RPM Data Experiment). Fluid domain is air at 25°C. The turbulence model used is "k-epsilon SST"[14].

The reference pressure is 1 atm at the inlet, and a normal speed those used in the wind tunnel, and at the outlet, a static pressure with a relative pressure at 0 atm. The frozen rotor interfaces connect the domains. Boundless boundary conditions are imposed on the walls of the estate. The type of analysis is "Steady state". A series of simulations were carried out, each time with an input speed change, and the angular velocity of the rotor (same conditions of the wind tunnel tests). The solution converged in all cases.

The geometry and grids are generated with the pre-processing Software Gambit. As shown in Fig.6, a hybrid mesh is created with Map-Hex mesh near the rotor and Tri Pave mesh in the other regions.

Figure 6: Geometry and Grid of the 1/8 Wells turbine model

3. Experimental and Numerical Analysis

The turbine performance under steady flow conditions is evaluated by turbine efficiency η, torque coefficient C_T and input coefficient C_A against flow coefficient Φ. The definitions of these parameters are as follows:

Pressure drop Δp^* across the rotor,

$$\Delta p^* = \frac{\Delta p_0}{\rho \cdot \omega^2 \cdot R_t^2} \tag{1}$$

Where, Δp_0 is the stagnation pressure drop across the turbine, taking the pressure values at the inlet and the outlet on the turbine tests, and ρ density of air.

Torque coefficient C_T, obtained from the numerical analysis for each inlet axial velocity V_A used in test is shown in Fig. 7.

$$C_T = \frac{T}{\frac{1}{2} \cdot \rho \cdot \left(V_A^2 + U_t^2\right) \cdot R_t \cdot Z \cdot b \cdot C} \tag{2}$$

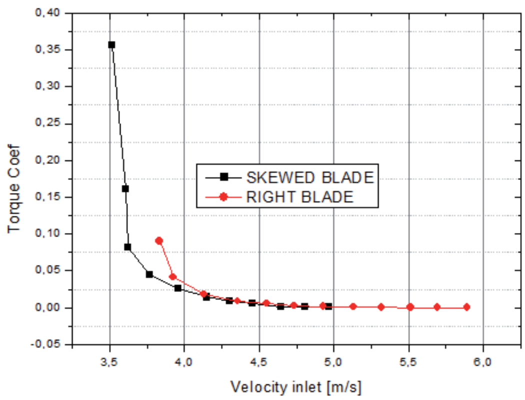

Figure 7: Torque coefficient C_T for each inlet axial velocity [m/s]

Input coefficient C_A:

$$C_A = \frac{\Delta p_0 \cdot \pi \cdot R_t^2}{\frac{1}{2} \cdot \rho \cdot \left(V_A^2 + U_t^2\right) \cdot Z \cdot b \cdot C} \qquad (3)$$

Flow coefficient $\boldsymbol{\Phi}$:

$$\Phi = \frac{V_A}{U_t} \qquad (4)$$

And efficiency η:

$$\eta = \frac{T \cdot \omega}{\Delta p_0 \cdot Q} = \frac{C_T}{C_A} \cdot \left(\frac{1}{\Phi}\right) \qquad (5)$$

Where:

Ut is the peripheral velocity with: $U_t = \omega . R_t$, ω, is the rotor angular velocity and R_t is the tip radius. Furthermore, V_A is the axial velocity normal to the plane of rotation, Z is the number of blades, b is the blade span, C is the blade chord and the medium blade chord in sweep blade, and Q is the volume flow rate.

Fig. 8 shows the difference between RPM turbine of rectangular plan form blades and sweep blades. It's observed that for the same Velocity inlet condition. The RPM of turbine of right blade is less than the RPM of turbine with sweep blade; this difference is due to the difference of geometry and solidity. This difference is constant when the velocity inlet is higher than 3.4m/s for the turbine with sweep blade, and 3.8 m/s for the turbine with right blade.

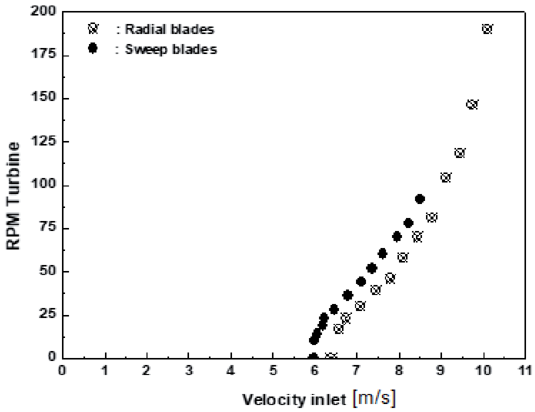

Figure 8: Comparison between RPM turbine Right blades and Sweep blades with Velocity of flow inlet (m/s)

The analysis of pressure drop across flow rate turbine function for the right blade and the sweep blade indicate that when the value of $\boldsymbol{\Phi}$ ($\boldsymbol{\Phi} < 0.1$) decreases, Δp^* is the same in the two turbine, and when $\boldsymbol{\Phi}$ increase the Δp^* of sweep is higher than the right one the graph shape for the both turbines is parabolic.

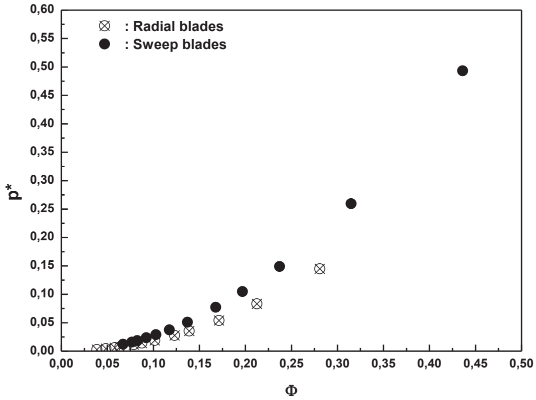

Figure 9: Pressure Drop Coefficient across Turbines (P*= f (Flow coefficient Φ))

Fig. 9, 10 and 11 present the experimental results of the sweep blade effect on the turbine performance. We note from the Fig. 7, it is clear that the value of C_T in the case of the turbine with sweep blade is higher than that in the case of the turbine with right blade in the whole range of attack angle α ($\alpha = \text{arctg } \Phi$).

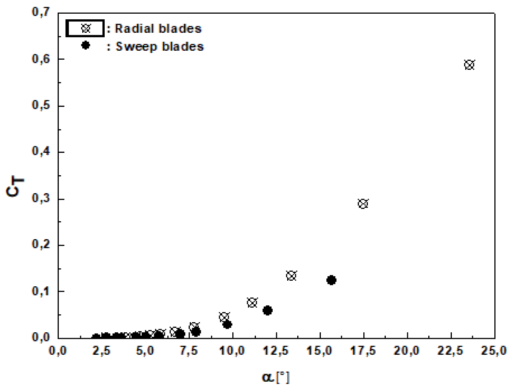

Figure 10: Torque Coefficient C_T across Turbines (C_T = f (Flow coefficient Φ))

Regarding input coefficient CA characteristics in Fig.11, although the CA$-\ \alpha$ also increases with sweep blade, Furthermore, Fig. 12 its peak efficiency increases with sweep blade than right blade from the value of $\Phi = 0.05$, the value in the case of the maximum efficiency of turbine sweep blade is approximately 62%. When compared to the turbine with right blade, it increases by approximately 10%. From Fig. 10, it can be noted that the CT slightly improved by using the swept blade.

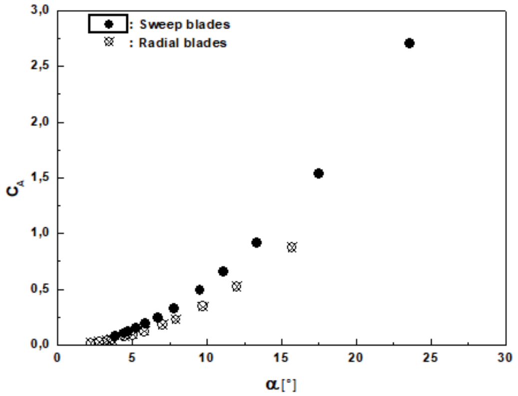

Figure 11: Input Coefficient C_A (C_A= f(Attack flow angle α [°]))

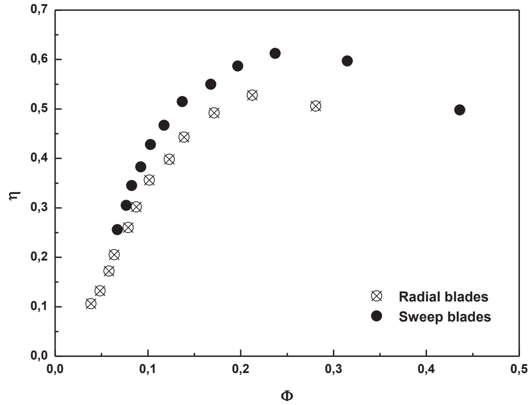

Figure 12: Efficiency (η = f (Flow Coefficient Φ))

According to the above facts, it is clear that the increase of peak efficiency and the postponement of stall point are achieved by swept blade. Consequently, the swept blade than square one is effective in enhancing the turbine performance.

4. Conclusion

In this study, an experimental investigation is carried out with the aid of combined system experimental-fluent in order to improve the performance of the Wells turbine for wave energy conversion. In this context, two form of blades turbines rectangular plan form blades and sweep blades are tested in the Wind Tunnel of the IISTA Fluid Dynamics Laboratory (University of Granada) using unidirectional airflow.

The analysis of the obtained results shows clearly that the characteristics of the Wells turbine with sweep blades are highest, the flow-field on the blades showed a great influence on the separation of the boundary layer and the distribution of the flow pressure around the blade changes significantly at tip, where the flow-field on the rectangular plan form blade have shown a stall zone at tip blade and becomes important when the flow increases, which significantly affects the local torque and therefore performance.

The sweep of blade turbine is a promising design factor for improved performance.

References

[1] T. Setoguchi and M. Takao, "Current Statut of self-rectifying Air Turbines for Wave Energy Conversion,"Energy Conversion and Management, Vol. 47, No. 15-16, 2006, pp. 2382-2396. N.

[2] Uddin, M.N.; Atkinson, M.; Opoku, F. A Computational Fluid Dynamics Investigation of a Numerically Simulated Wave Tank. Am. J. Mech. Eng. 2020, 8, 88–105.

[3] M. N. Uddin, M. Atkinson and F. Opoku, CFD Investigation of a Hybrid Wells Turbine with Passive Flow Control, J. energies, Vol. 16, 2023.

[4] C. Yang, C. Wan, X. Bai, T. Xu, L. Zhao, H. Chen, L. Johanning and T. E Baldock, Numerical investigation on the hydrodynamic and conversion performance of a dual cylindrical OWC integrated into a caisson-type breakwater, Ocean EngineeringVolume 305, 2024.

[5] T. Setoguchi, K. Kaneko and M. Inoue, "Determine of Optimum Geometry of Wells Turbine Rotor for Wave Power Generator," Proceedings of 3rd Symposium on Ocean Wave Utilz, Japan Agency for Marine-Earth Science and Technology (JAMSTEC), Tokyo, 1991, pp. 141-149. (in Japanese) .

[6] Gratton, T.; Ghisu, T.; Parks, G.; Cambuli, F.; Puddu, P.Optimization of Blade Profiles for the Wells Turbine. Ocean. Eng. 2018,169, 202–214.

[7] Opoku, F.; Uddin, M.N.; Atkinson, M. A Review of Computational Methods for Studying Oscillating Water Columns—The Navier-Stokes Based Equation Approach. Renew. Sustain. Energy Rev. 2023, 174, 113124.

[8] Numerical simulation of a simple OWC problem for turbine performance A. Moñino a , E. Medina-López , M. Clavero a , S.M.K. Benslimane / International Journal of Marine Energy 20 (2017) 17–32

[9] T. Setoguchi, K. Kaneko and M. Inoue, "Determine of Optimum Geometry of Wells Turbine Rotor for Wave Power Generator," Proceedings of 3rd Symposium on Ocean Wave Utilz, Japan Agency for Marine-Earth Science and Technology (JAMSTEC), Tokyo, 1991, pp. 141-149. (in Japanese) .

[10] S. RAGHUNATHAN and C.P. ABTAN. "Aerodynamic performance of a Wells air turbine", Journal of Energy, Vol. 7,

No. 3 (1983), pp. 226-230.
[11] Raghunathan, S. and Tan, C.P., Aerodynamic Performance of a Wells Air Turbine, J. Energy, 1981, Vol.7, No.3, pp.226-230.
[12] Raghunathan S. The Wells Turbine for Wave Energy Conversion [Journal] // Prog. Aerospace Sci. - 1995. - Vol. 31. - pp. 335-386.
[13] Gato, L.M.C. and Falcao, A.F.deO., "Aerodynamics of the Wells Turbine", Int. J. Mechanical Sci., 30, 6, 383-395 (1988).
[14] Gato, L.M.C. and Falcao, A.F.deO., "Performance of Wells Turbine with Double Row of Guide Vanes", JSME Int. J. (Ser. ii), 33, 2, 262-271 (1990).
[15] Medina-López, E. & Moñino Ferrando, A. & Clavero Gilabert, M. & del Pino, C. & Losada Rodríguez, M., 2016. "Note on a real gas model for OWC performance," Renewable Energy, Elsevier, vol. 85(C), pages 588-597.
[16] Inoue, M., Kaneko, K., Setoguchi, T. and Shimamoto, K., "Studies on Wells Turbine for Wave Power Generator (Part 4; Starting and Running Characteristics in Periodically Oscillating Flow)", Bull. JSME, 29, 250, 1177-1182 (1986b).
[17] Kaneko, K., Setoguchi, T. and Raghunathan, S., "Self-Rectifying Turbine for Wave Energy Conversion", Proc. 1st Offshore and Polar Engg. Conf., Edinburgh, UK, Aug., 11-16, 1, pp.385-392 (1991).
[18] M. Torresi, S.M. Camporeale, G. Pascazio, Detailed CFD Analysis of the Steady Flow in a Wells Turbine Under Incipient and Deep Stall Conditions, Journal of Fluids Engineering, Vol. 131 / 071103-1, July 2009.
[19] K. Takasaki, M. Takao, T. Setoguchi, Effect of Blade Shape on the Performance of Wells Turbine for Wave Energy Conversion International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol:8, No:12, 2014
[20] Shyjo Vivek Joseph John, G. Anil Kumar, Design and Analysis of 3D Blades for Wells Turbine,Shyjo Johnson, Sriram S Kumar, IJIRST –International Journal for Innovative Research in Science & Technology| Volume 1 | Issue 11 | April 2015
[21] Torresi M, Camporeale S, Pascazio G (2007) Experimental and numerical investigation on the performance of a Wells turbine prototype. In: Seventh European wave and tidal energy conference, Porto, Portugal, Sept, 2007, pp 11–14
[22] Raghunathan, S. The Wells air turbine for wave energy conversion. Prog. Aerospace Sci., 1995, 31, 335–386.
[23] Torresi M, Pranzo D, Camporeale S, Pascazio G (2011) Improved design of high solidity Wells Turbine. In: 9th European wave and tidal energy conference. Southampton, England, pp 5–9
[24] M. Webster., L.M.C Gato , 1998, The Effect of Rotor Blade Sweep On the Performance of the Wells Turbine ,The Eighth International Offshore and Polar Engineering Conference, 24-29 May, Montreal, Canada