Study of the meshing effect on the numerical results of a NACA2415 airfoil wind turbine

Z. DRISS[a*], W. BARHOUMI[a], T. CHELBI[a], A. KAFFEL[b], M.S. ABID[a]

[a] Laboratory of Electro-Mechanic Systems (LASEM), National School of Engineers of Sfax (ENIS), University of Sfax, B.P. 1173, Road Soukra km 3.5, 3038 Sfax, TUNISIA
[b] University of Maryland College Park, MD 20742, USA
*Email : zied.driss@enis.rnu.tn

Résumé :

Dans ce travail, nous sommes intéressés à l'étude de l'effet du maillage sur les résultats numériques développés à l'aide d'un code commercial de dynamique des fluides numérique (CFD) basé sur la résolution des équations de Navier-Stokes et du modèle de turbulence du type k-ε standard. Ce modèle numérique est utilisé pour l'étude de la structure aérodynamique autour d'une éolienne à axe horizontal avec un profil de type NACA2415. Les résultats numériques développés sont comparés avec les résultats expérimentaux trouvés à l'aide d'une soufflerie aérodynamique pour choisir le modèle numérique le plus adéquat.

Abstract :

In this work, we are interested on the study of the meshing effect on the numerical results developed using a commercial CFD code based on the resolution of the Navier-Stokes equations in conjunction with the standard k-ε turbulence model. The developed numerical method is used to study the aerodynamic structure of a horizontal axis wind turbine with a NACA2415 airfoil type. The developed numerical results are compared with experimental results conducted on an open wind tunnel to choose the adequate numerical model.

Mots clefs : Eolienne, NACA2415, Soufflerie, Modélisation, Maillage, CFD.

1 Introduction

To prepare a truly sustainable development, the community recommends to increase the share of renewable resources for electricity generation. The production of electricity by wind turbines is playing a major role. In this context, many scientists have examined the effects of wind turbines parameters design. For example, Hirahara et al. [1] developed a unique and very small wind turbine with a diameter of 500 mm and a small aspect ratio for wide use in urban space. The basic performance was tested for various free stream and load resistance. The airflow around the turbine was investigated using a particle image velocimetry (PIV). Wright and Wood [2] showed that the acceleration and deceleration of the rotor at speeds below its controlled maximum speed for a range of wind speeds were calculated and compared with data. Schreck and Robinson [3] showed that wind turbine blade aerodynamic phenomena can be broadly categorized according to the operating state of the machine, and two particular aerodynamic phenomena assume crucial importance. At zero and low rotor yaw angles, increasing rotation determines blade aerodynamic response. At moderate to high yaw angles, dynamic stall dominates blade aerodynamic. The main goal of the Mirzaei et al. [4] investigation was to understand the flow field structure of the separation bubble formed on NLF-0414 airfoil with glaze-ice accretions using CFD and hot-wire anemometry and comparing these results with previous researches performed on NACA 0012 airfoil. Hu et al. [5] showed that Coriolis and centrifugal forces play important roles in 3D stall-delay. At the root area of the blade, where the high angles of attack occur, the effect of the Coriolis and centrifugal forces is dominant. Thus, it shows apparent stall-delay phenomenon at the inner part of the blade. However, by increasing the Reynolds number, the separation position has a stronger effect than by increasing the Coriolis and centrifugal forces. Sicot et al. [6] investigated the aerodynamic properties of a wind turbine airfoil. Particularly, they studied the influence of the inflow turbulence level (from 4.5% to 12%) and of the rotation on the stall mechanisms in the blade. A local approach was used to study the influence of these parameters on the separation point position on the suction surface of the airfoil, through simultaneous surface pressure measurements around the airfoil.

Despite the numerous papers already devoted to this subject, studying the aerodynamic characteristics of the horizontal axis NACA2415 airfoil type wind turbine is still needed. The present work is concerned to study the meshing effect on the numerical results developed using the software "SolidWorks Flow Simulation". The developed numerical results are compared with experimental results conducted on an open wind tunnel to choose the adequate numerical model.

2 Computational Domain

Using the software "SolidWorks", the computational domain is presented in figure 1. It is defined by the interior volume of the wind tunnel blocked by two planes: the first one is in the tranquillization chamber entry and the second one is in the exit of the diffuser. The considered wind turbine is a horizontal axis with a NACA2415 airfoil type placed in a test section of the wind tunnel (Figure 1). This turbine consists of three adjustable blades of a length L=110 mm and width C=45 mm. The wedging angle is measured between the rotation plane of the wind turbine and the chord; it's equal to β=30°.

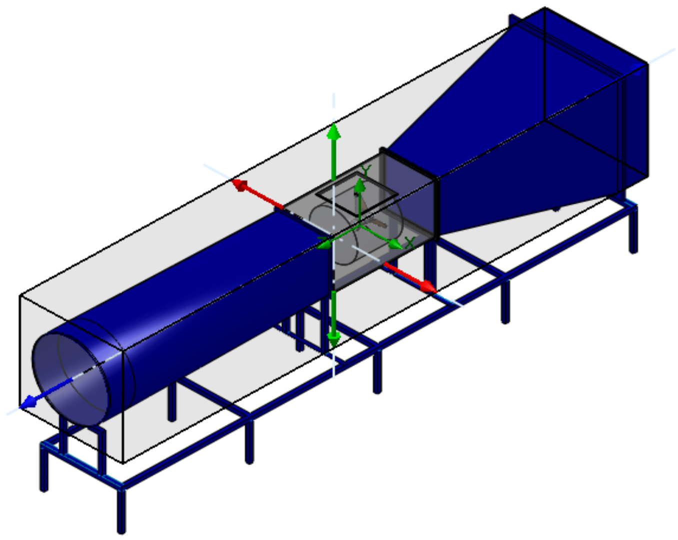


FIG. 1 – CAD model of the wind tunnel.

3 Boundary conditions

The boundary conditions of our application are presented in figure 2. It consists on the velocity inlet equal to V=3 m.s^{-1}. In these conditions, the frequency of the speed variator is equal to f=50 Hz. The pression of the air flow through the drive section is made at the atmospheric conditions. For this reason, the pressure outlet is set equal to p=101325 Pa. Around the wind turbine, a rotating area was considered with an angular velocity equal to Ω=42 rad.s^{-1}. The choice of these conditions is very important and requires a lot of precision for an exact description of the problem in order to get satisfactory results [7-9].

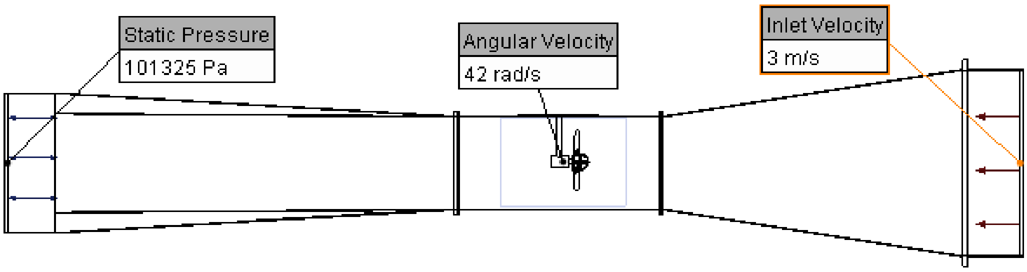
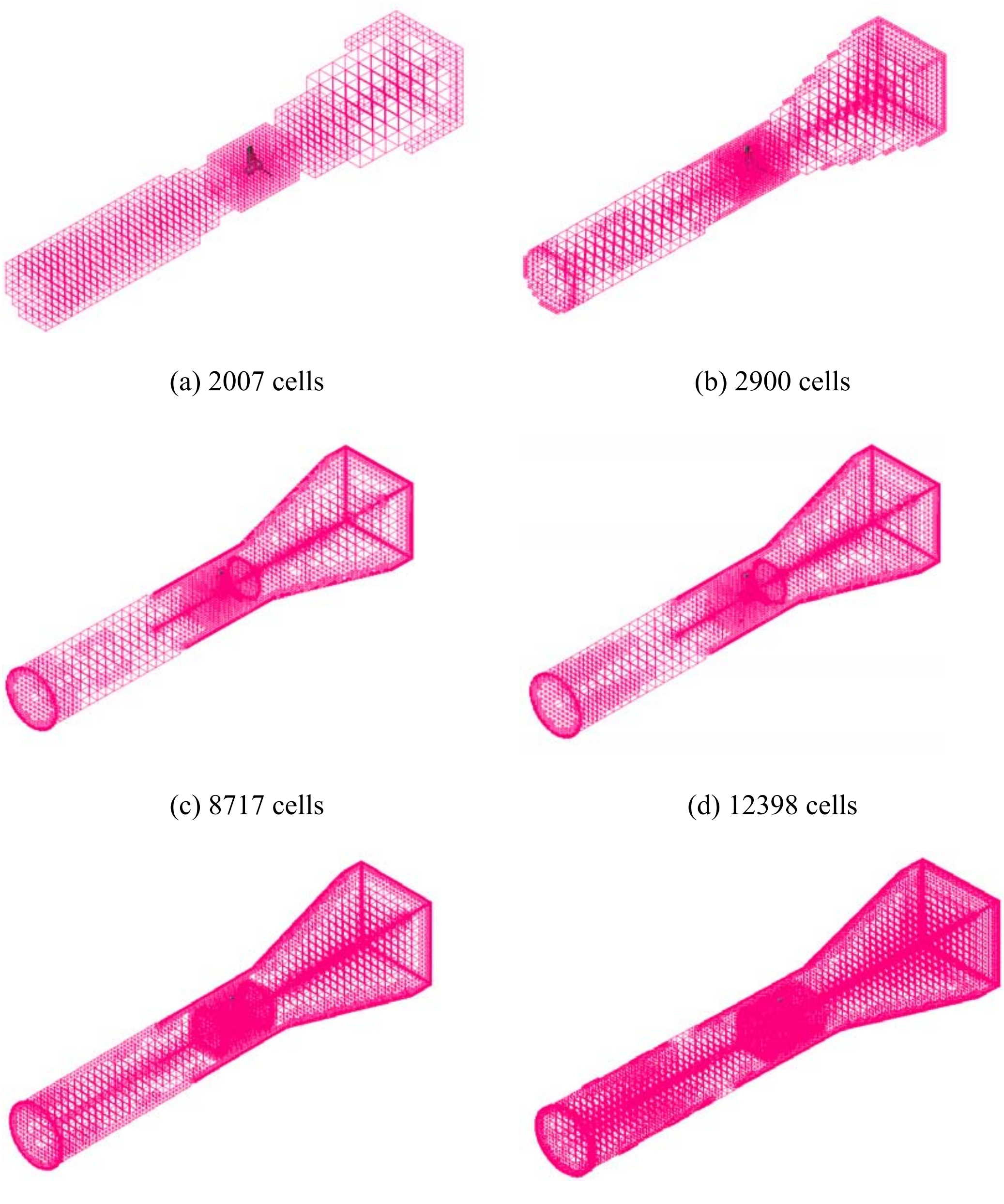


FIG. 2 – Boundary Conditions.

4 Meshing

The goal of this section is to demonstrate various meshing capabilities of Flow Simulation allowing us to better adjust the computational mesh to the problem at hand. Although the automatically generated mesh is usually appropriate, intricate problems with thin or small, but important, geometrical and physical features can result in extremely high number of cells, for which the computer memory is too small. In such cases Flow Simulation options allow us to manually adjust the computational mesh to the solved problem's features to resolve them better. The geometry can be resolved reasonably well. However, if we generate the mesh and zoom in a thin region, we will see that it may still unresolved. In order to resolve these regions properly, we will use the Local Initial Mesh option. The local initial mesh option allows us to specify an initial mesh in a local region of the computational domain to better resolve the model geometry and flow peculiarities in this region. The local region can be defined by a component of the assembly, disabled in the Component Control dialog box, or specified by selecting a face, edge or vertex of the model. Local mesh settings are applied to all cells intersected by a component, face, edge, or a cell enclosing the selected vertex. The local mesh settings do not influence the basic mesh but are basic mesh sensitive: all refinement levels are set with respect to the basic mesh cell. To refine the mesh only in a specific region and avoid excessive splitting of the mesh cells in other parts of the model, we apply a local initial mesh at the component surrounding this region. The component is created specially to specify the local initial mesh. The settings on the Narrow Channels tab control the mesh refinement in the model's flow passages. Characteristic number of cells across a narrow channel box specifies the number of initial mesh cells (including partial cells) that Flow Simulation will try to set across the model's flow passages in the direction normal to solid/fluid interface. If possible, the number of cells across narrow channels will be equal to the specified characteristic number. Otherwise it will be close to the characteristic number. If this condition is not satisfied, the cells lying in this direction will be split to satisfy the condition.

In this application, we are interested on the study of the mesh resolution's effect. In fact, we are going to change the size of the mesh and compare the results with the values of the velocity collected from the test section obtained experimentally. In particular, we have chosen to study six meshes. The first case to be treated corresponds to a cell size of 200 mm. The second case corresponds to a cell size of 100 mm. The third case corresponds to a cell size of 20 mm. The fourth case corresponds to a cell size of 15 mm. The fifth case corresponds to a cell size of 10 mm. The latter one corresponds to a cell of 5 mm (figure 3). In these cases, the number of cells is respectively equal to 2007, 2900, 8717, 12398, 20185 and 54086 cells; which corresponds to a coarse mesh in the first case and a refined mesh in the sixth case. Figure 4 shows the meshing on the two transverse planes defined by x=0 mm and z=0 mm.

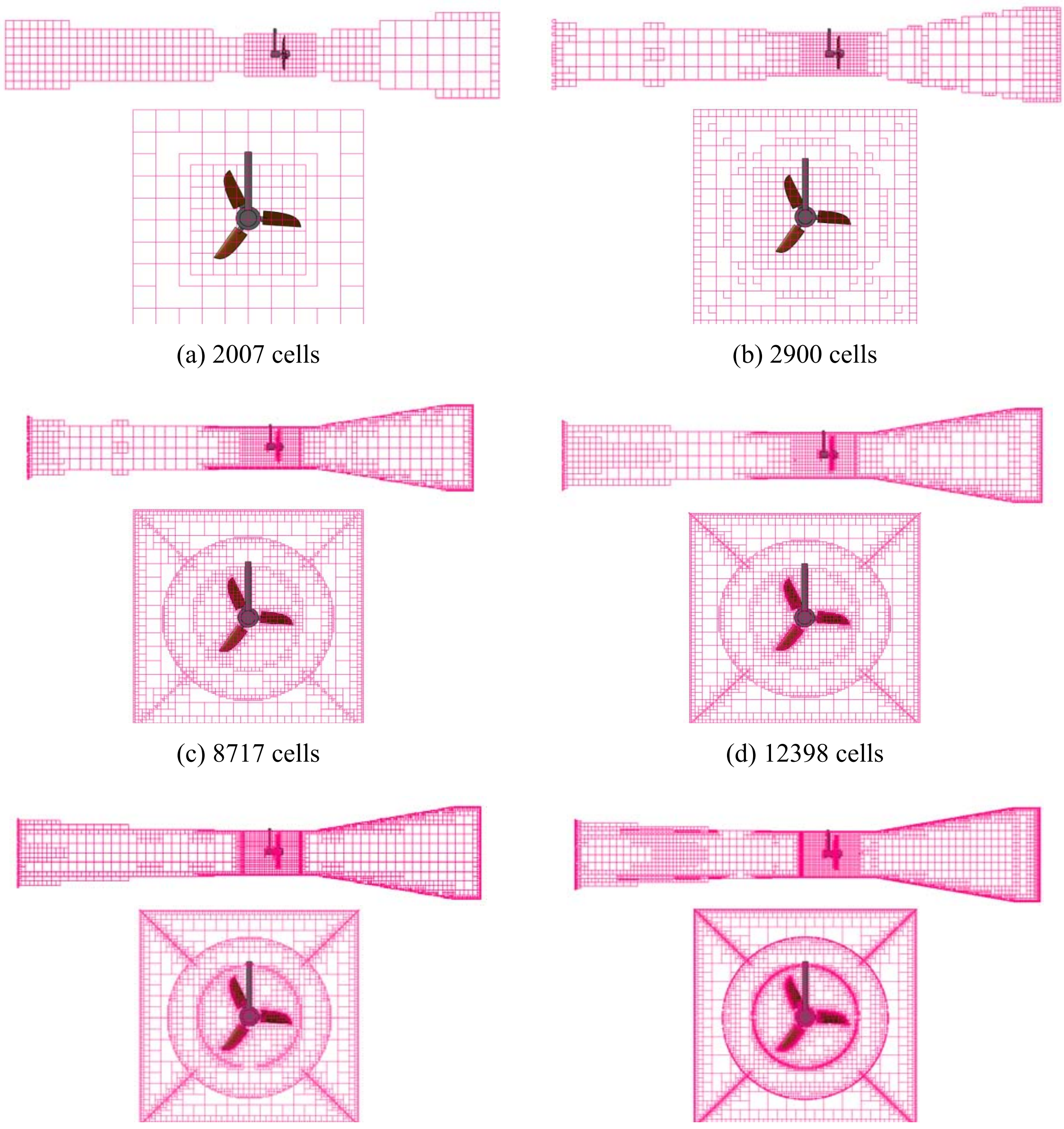


(a) 2007 cells (b) 2900 cells

(c) 8717 cells (d) 12398 cells

(e) 20185 cells (f) 54086 cells

FIG. 3 – 3D view of the meshing.



(a) 2007 cells (b) 2900 cells

(c) 8717 cells (d) 12398 cells

(e) 20185 cells (f) 54086 cells

FIG. 4 – 2D view of the meshing.

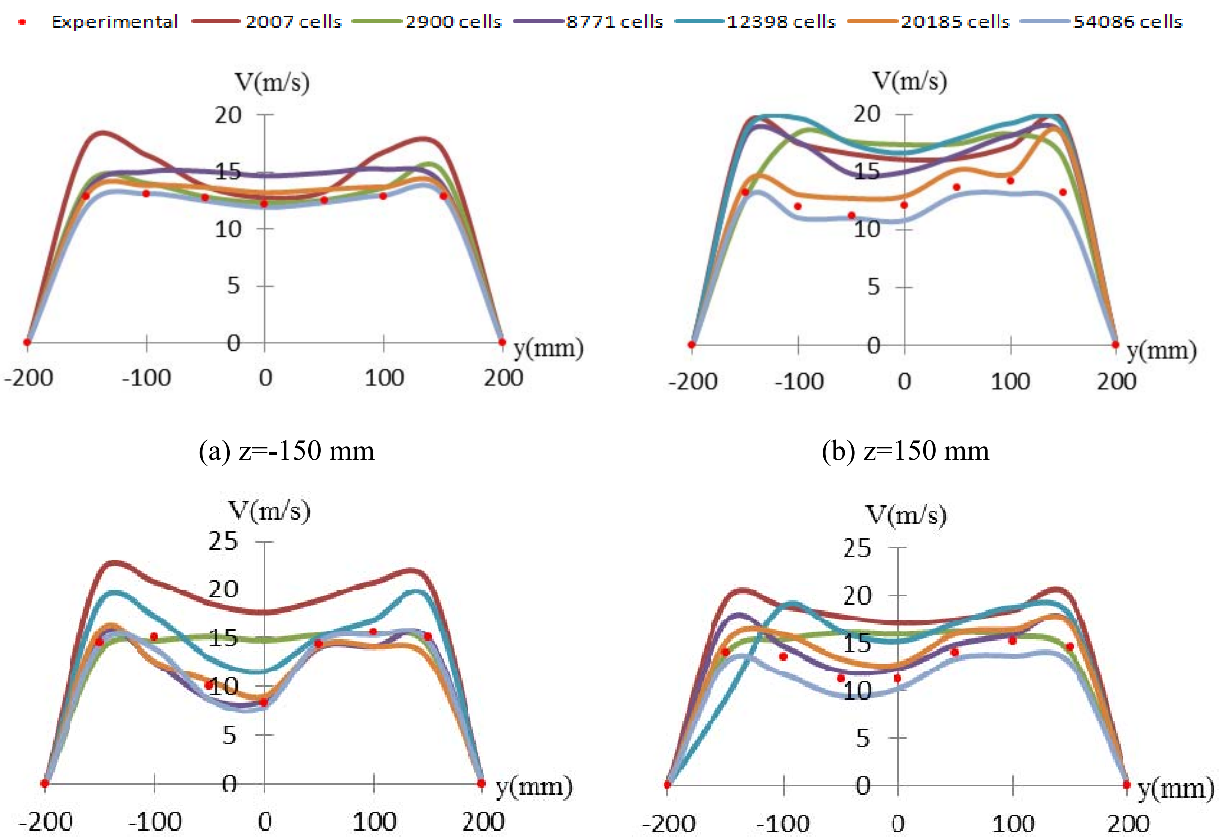
5 Comparison between numerical and experimental results

In this work, computer investigations are carried out to study the flow field developing around a NACA2415 airfoil wind turbine. The Navier-Stokes equations in conjunction with the standard k-ε turbulence model are considered. These equations are solved numerically to determine the local characteristics of the flow [10-12]. The models tested are implemented in the software "SolidWorks Flow Simulation" which uses a finite volume scheme. The numerical results are compared with experiments conducted on an open wind tunnel to validate the numerical results. This will help improving the aerodynamic efficiency in the design of packaged installations of the NACA2415 airfoil type wind turbine. Table 1 presents the resolution time and the velocity value measured in the test section for the treated cases. According to these results, we note that the velocity value obtained for the fifth case is the closest to the experimentally measured value for the point repered by the intersection of the planes defined by x=50 mm, y=50 mm and z=100 mm. Also, it has been noted that the time resolution increases with the decrease of the size of mesh cells. Later in the work, it is proposed to use the fifth mesh. This choice leads us to a better result with regards to the precision and the spent time.

Figure 5 shows the different profiles of the averge velocity for different cells size. It presents the superposition of the numerical results gathered from the software "Solid works Flow simulation" and the experimental results taken by the aneometer. The velocity profiles are chosen for different directions situated in the test section. The considered directions are defined by the intersection of the plane x=0 mm with the planes z=50 mm, z=100 mm, z=150 mm and z=-150 mm. The different velocity profiles seem to have the same appearance. However, the velocity values depend on the cell size. Indeed the greater the cell size gets the more the gap between numerical and experimental results is large. The best result regarding precision and time is found to be a cell of 5 mm size.

Table 1 – Mesh selection criterium

Cases study	Cells number	Resolution time (h:m:s)	Velocity ($m.s^{-1}$)	
			Numerical	Experimental
1	2007 cells	0:1:50	17.47	13.94
2	2900 cells	0:2:00	16.18	
3	8771 cells	0:9:32	15.15	
4	12398 cells	0:13:41	14.88	
5	20185 cells	0:21:37	15.97	
6	54086 cells	1:07:27	13.34	



(a) z=-150 mm (b) z=150 mm

(c) z=50 mm (d) z=100 mm

FIG. 5 – Velocity profiles.

5 Conclusion

In this work, we are interested on the study of the meshing effect on the numerical results of a horizontal axis wind turbine with a NACA2415 airfoil type. Particularly, we have changed the cells size and we have compared the numerical results with the values of the velocity collected experimentally from an open wind tunnel to choose the adequate numerical model. The different velocity profiles seem to have the same appearance. However, the velocity values depend on the cell size. Indeed the greater the cell size gets the more the gap between numerical and experimental results is large. The best result regarding precision and time is found to be a cell of 5 mm size. In the future, we intend using the particle image velocimetry laser (PIV) system to determine the local characteristics.

Références

[1] H. Hirahara, M.Z. Hossain, M. Kawahashia and Y. Nonomura, Testing basic performance of a very small wind turbine designed for multi-purposes, Renewable Energy 30 (2005) 1279-1297.
[2] A.K. Wrigh and D.H. Wood, The starting and low wind speed behaviour of a small horizontal axis wind turbine, Journal of Wind Engineering and Industrial Aerodynamics 92 (2004) 1265-1279.
[3] S.J. Schreck and M.C. Robinson, Horizontal Axis Wind Turbine Blade Aerodynamics in Experiments and Modeling, IEEE Transactions on Energy Conversion 22 (2007) 61-70.
[4] M. Mirzaei, M.A. Ardekani and M. Doosttalab, Numerical and experimental study of flow field characteristics of an iced airfoil, Aerospace Science and Technology 13 (2009) 267-27.
[5] D. Hu, O. Hu and Z. Du, A study on stall-delay for horizontal axis wind turbine, Renewable Energy 31 (2006) 821-836.
[6] C. Sicot, P. Devinant, S. Loyer and J. Hureau, Rotational and turbulence effects on a wind turbine blade: investigation of the stall mechanisms, Journal of Wind Engineering and Industrial Aerodynamics 96 (2008) 1320-1331.
[7] Driss Z, Damak A, Karray S, Abid MS. Experimental study of the internal recovery effect on the performance of a Savonius wind rotor. Research and Reviews: Journal of Engineering and Technology 1 (2012) 15-21.
[8] Damak A, Driss Z, Abid MS. Experimental investigation of helical Savonius rotor with a twist of 180°, Renewable Energy 52 (2013) 136-142.
[9] Driss Z, Mlayeh O, Driss D, Maaloul M, Abid MS. Numerical simulation and experimental validation of the turbulent flow around a small incurved Savonius wind rotor, Energy 74 (2014) 506-517.
[10] Z. Driss, G. Bouzgarrou, W. Chtourou, H. Kchaou and M.S. Abid, Computational studies of the pitched blade turbines design effect on the stirred tank flow characteristics, European Journal of Mechanics B/Fluids 29 (2010) 236-245.
[11] Z. Driss, G. Bouzgarrou, H. Kchaou and M.S. Abid, Computer simulation of the laminar flow in stirred tanks generated by the proximity impellers of a mono and double screws type with simple and modified profiles, Mechanics & Industries 12 (2011) 109-121.
[12] Z. Driss, S. Karray, W. Chtourou, H. Kchaou and M.S. Abid, Study of mixing structure in stirred tanks equipped with multiple four-blade Rushton impellers, Archive of Mechanical Engineering 59 (2012) 53-72.