CHEMICAL ENGINEERING TRANSACTIONS

VOL. 45, 2015

A publication of AIDIC
The Italian Association of Chemical Engineering
www.aidic.it/cet

Guest Editors: Petar Sabev Varbanov, Jiří Jaromír Klemeš, Sharifah Rafidah Wan Alwi, Jun Yow Yong, Xia Liu

ISBN 978-88-95608-36-5; **ISSN** 2283-9216

DOI: 10.3303/CET1545189

Characterisation of the Effects of Surface Modifications to Flow Across Rotating Cylinders using ANSYS CFD Modelling

Neil S.A. Lopez*, Brian K.G. Mara, Brian C.C. Mercado, Luigi A.B. Mercado, Jose M.G. Pascual

Mechanical Engineering Department, De La Salle University, 2401 Taft Ave, 1004 Manila, Philippines
neil.lopez@dlsu.edu.ph

Past and present studies have proven that rotating cylinders can generate significantly higher lift forces compared to airfoils. This force may be used to drive a wind generator for energy conversion and pollution reduction. Such, called a Magnus turbine, operates with low cut-in wind speeds, making it promising for urban power generation. Also, its self-stalling capability makes it safer to operate at strong wind speeds. However, the significant drag forces which accompany it have to be minimized. Analysing the whole system, one opportunity to improve its performance is to apply different surface modifications on the cylindrical rotor. In this study, a screening design of experiment was implemented to characterize the effects of different surface modifications to the lift and drag forces generated on a rotating cylinder. The experiments were run using the computational fluid dynamics simulation software ANSYS CFX. Results indicate that the effects demonstrate an exponential increase in drag reduction as the speed ratio is increased. The most significant reduction in drag comes from roughening the surface of the cylinder at high speed ratios while using bumps at lower speed ratios. Lastly, changing a regular to a frustum cylinder provides the most increase in lift. This study should provide leads for further research and inspire other process applications.

1. Introduction

The potential aerodynamic advantages of a rotating cylinder or a Magnus rotor inspired many engineers and scientists in the early twentieth century to search for practical applications. There were attempts to propel giant ships and improve flight aerodynamics using the Magnus rotor, as reported by Seifert(2012). Though promising, very few had reached meritorious status over the past century. The lack of established design information and aerodynamic modelling has hampered the progress of this technology.

Today, scientists and engineers are starting to look for efficient alternatives again as energy costs and climate concerns are rising. Most recently in 2010, as cited in Seifert (2012), E-Ship 1 of Enercon was equipped with four Magnus rotors to reduce fuel costs by 30 – 40 %. Another remarkable recent application, which is also the interest of this study, is for wind energy harvesting, as attempted by Murakami and Ito (2009) through their proprietary Magnus wind turbine.

A rotating cylinder traversing a fluid bears plenty of advantages against an airfoil-type rotor. First and foremost, it generates more lift force than an airfoil as reported by Sedaghat (2014). Sedaghat (2014) also reported lower cut-in wind speeds of 1 – 2 m/s, which should improve the capacity factor of the wind turbine. Lastly, and probably the most interesting feature is its self-stalling capability. At gusts of about 50 m/s and above, the Magnus rotor enters a neutral range, wherein the lift stops responding to the increase in wind speed. This was reported by Seifert (2012). This would make Magnus wind turbines safer and last longer than conventional airfoil-type machines.

However, the challenge is that a rotating cylinder also produces plenty of drag. This prohibits it from attaining higher lift-to-drag ratios in comparison to airfoil-type rotors. Nevertheless, there is potential to minimize this drag and further enhance lift through the application of surface modifications. Previous

Please cite this article as: Lopez N.S.A., Mara B.K.G., Mercado B.C.C., Mercado L.A.B., Pascual J.M.G., 2015, Characterisation of the effects of surface modifications to flow across rotating cylinders using ansys cfd modelling, Chemical Engineering Transactions, 45, 1129-1134 DOI:10.3303/CET1545189

studies have shown that introducing different surface modifications can be beneficial to the aerodynamics of a rotating cylinder.
According to Seifert (2012), the surface roughness relatively increases the torque produced compared to a smooth surface. In an experiment, three samples were tested namely a smooth, a wooden and a sanded cylinder. Results showed that the lift coefficient of the sanded cylinder was the highest among the rest.
In another study, effects of hexagonal patterns both protruding and indented were investigated by Butt and Egbers (2013). The hexagonal pattern that was pressed outwards at an angle of 90° displayed the highest drag reduction. Similarly, Srivastav (2012) pointed out in his experiment conducted on an aerofoil that the outward dimple also produced lesser drag than the inward dimple.
In a study made by Huang (2010), it was noted that adding helical grooves reduced drag by 20 % when in a subcritical range of the Reynolds number. The effects of the grooves though became negligible when the roughness ratio of k/D = 0.012 or higher.
Lastly, Sun et al. (2012) made a study about the various effects of different cylindrical shapes to the performance of the Magnus turbine. A frustum-shaped cylinder posed a power coefficient of 30 % which was the highest at a tip speed ratio of 1.8. In terms of torque coefficient, the frustum shaped cylinder was also the highest.
However, it is yet to be known if there exists an interaction between the different modifications and which one is best. Using a full-factorial design of experiment, the present work will collectively investigate all the aforementioned modifications from literature, and determine if lift and drag behaviour may be further enhanced by combining a particular set of modifications. It will also determine if a particular modification is superior and more worthwhile than the rest, narrowing down the scope for future researches in the area. The current study will take off from this premise, through the use of computer modelling and simulation tool ANSYS CFX and statistical software JMP.
The use of computer modelling such as CFD and simulation has been proven to be worthwhile in the pursuit of product and process characterization and optimization. Similar with the methodology of this study, past researches have asserted its validity and functionality in various applications. Cuervo, et al. (2014) had performed optimization on the setting up of conditions for dust explosion tests by combining experimental results with that of the CFD simulation results. Rosa et al. (2014) also used CFD modeling in optimizing the anaerobic sequencing batch reactor where the model generated accurate results. Bubbico et al. (2014) simulated the dispersion of cold nitrogen cloud using the ANSYS Fluent code.
On a different perspective, one of the most important first steps of process integration is product design. Process integration principles have been used in the past to investigate wind turbines. Becker et al. (2014) studied the effect of materials, as well as moisture and temperature, to the structural integrity of a wind rotor blade. Lastly, Barbera et al. (2013) studied the applicability of state of the art condition based maintenance practices to wind turbine generators.

2. Theoretical considerations

Fluid flow across rotating cylinders is central to this study. Like an airfoil, a rotating cylinder traversing a fluid experiences lift and drag forces. The direction of lift is always perpendicular to the flow, while drag is always parallel to the flow.
Lift on a rotating cylinder, which is also referred to as the Magnus force, is generated via the Bernoulli Effect. As fluid travels across the rotating cylinder, a pressure gradient is created perpendicular to the flow direction. This pressure gradient creates the lift or Magnus force. The lift force is a function of the ratio of the cylinder’s rotational speed and free stream velocity, unlike for an airfoil which is dependent on the angle of attack.
Drag comprises of two components: friction and pressure drag. For bluff bodies such as a cylinder, the pressure drag is significantly greater than the frictional component. Pressure drag is a result of flow separation. However, for non-separated flows, the drag is mainly due to frictional losses.
Lift and drag forces on a rotating cylinder are a function of speed ratio, according to Sedaghat (2014). It is defined as the ratio of the cylinder’s tangential velocity at the outer surface to the bulk velocity of the incoming air. Lastly, lift to drag ratio is simply the ratio between the lift and drag forces acting on the rotating cylinder.

3. Methods

Three-dimensional models of the modified cylinders inside a wind tunnel configuration were developed using SolidWorks software. Subsequently, the CFD models were developed using the ANSYS CFX platform. Detailed development and validation of the model were discussed by the same authors in

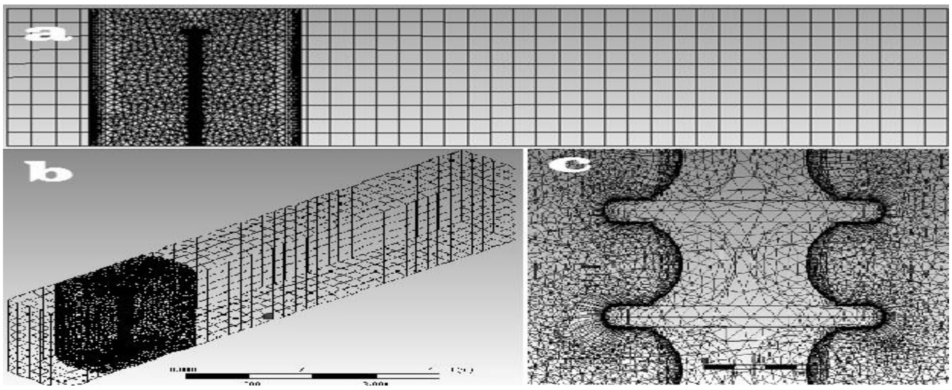

Figure 1: Screenshot of final fluid domain mesh configuration. a) right side view; b) perspective view; and c) zoomed-in on the cylinder to show inflation layers

separate papers in 2014. Mara et al. (2014a) discusses the development and validation of the CFD and meshing model used in the study, while Mara et al., (2014b) discusses the selection process for the turbulence model employed. The CFD model developed simulates an isolated rotating cylinder inside a wind tunnel configuration. The unstructured mesh utilized inflation layers near the cylinder surface with a first node distance of 0.8 mm. A higher resolution mesh was utilized near the cylinder wall to ensure effects of the modifications are captured. It should be noted that on the more physically complicated cylinders, the model utilized a multi-zone meshing approach to save on computing resources. Global and local mesh settings are summarized in Table 1 for easy replication of the model. A screenshot of the mesh cross-section is shown in Figure 1. The Eddy Viscosity Transport Equation turbulence model, a modified version of the k-ω model, was adopted in the CFD model. Models in the k-ω family are reliable in predicting flow separation and can maximize the benefit of high resolution meshes because they do not use wall functions. Besides, turbulence models using wall functions like k-ε are detrimental to our purpose because they will not be able to correctly model the boundary layer interaction between the rotating cylinder and the flow. Lastly, roughness was introduced into the model by means of sand grain roughness height and geometric roughness height, which were made equal in the model. Defining the geometric roughness height is needed when flow transition and separation is expected, and most especially when the shape of the roughness element is not spherical like a sand grain.

For this screening design of experiment (SDOE), a full-factorial DOE was created using JMP statistical software. A full-factorial design gives the researchers the benefit of a high resolution DOE. Since all the experiments were ran on simulation software, the researchers were not constrained much by resources and could afford a full-factorial DOE which comprised of 180 runs (~720 simulation h). The control variables in the DOE were non-continuous as it was only desired to determine the contribution of each modification as it is turned on or off. The complete list of control variables and their corresponding levels in the full-factorial design can be seen on Table 2.The default rotor is a plain hydraulically smooth cylinder with uniform diameter across its length.

The experimental results were analyzed using JMP software, wherein a polynomial model was fitted on the data. The goodness of fit of the model was obtained and the main and interaction effects were measured and compared using scaled estimates from JMP. As defined from JMP documentation, scaled estimates measure how much a main effect or interaction affects a response when its value is increased or decreased by a unit, regardless of the scale used in the experiment.

4. Results and discussion

Prediction equations containing all main, second and third degree interaction effects were fitted on the raw data. A good fit was obtained with an average R^2 of 0.978. A different equation was derived for lift, drag and lift-to-drag ratio (LDR) at each speed ratio tested. Each equation had 20 terms and 36 observations in it. A total of 15 prediction equations were created from the simulation raw data. These fitted equations were used to compare the contributions of each surface modification and analyze their interactions.

By plotting the scaled estimate of each main and interaction effect on lift, drag and LDR per speed ratio (see Figures 2, 3, and4), the contributions of each were compared. An interesting observation was that the significant effects vary in low and high speed ratios.

Table 1: Summary of mesh settings developed using ANSYS CFX Mesher

Location	Meshing Strategy	Local Settings	
Global	Tetrahedral	Element size	0.0625 m
		Relevance	60
		Relevance Center	Fine
Local (Cylinder Surface)	Prism (Inflation layers)	Method	First Aspect Ratio
		First node distance	0.0008 m
		First aspect ratio	20
		Growth rate	1.2
		Maximum layers	1000
Local (Near-cylinder zone)	Tetrahedral	Element size (on zone wall)	0.1 m
		Zone diameter	2.25 m
Local (Zones away from cylinder)	Hexahedral	Element size	0.25

4.1 Lift enhancement

Converting the cylinder shape into a frustum consistently yielded the highest lift through all speed ratios. A dimple was more preferable than a bump at lower speed ratios but the bump became more effective beyond a speed ratio of 3.5. It is also notable that a rough surface paired with straight grooves incurred a negative effect on lift. With regards to lift enhancement, the main effects were more significant than the interactions. It can be observed that applying grooves to the frustum cylinder reduces its lift enhancement effects.

4.2 Drag reduction

Significant effects demonstrate an exponential-like trend in drag reduction as the speed ratio increases. Roughening the surface of the cylinder was the most effective drag reducer beyond a speed ratio of 3.5. Below a speed ratio of 3, introducing bumps was most effective. However, having a parabolic trend, its effect started to decrease beyond a speed ratio of 3. Similarly, the main effects are more significant than the interactions. It is notable that although insignificant individually, the frustum cylinder and dimples combination resulted to significant drag reduction at high speed ratios (> 3.5).

By measuring wall shear stress on the surface of the cylinder, it was determined that no flow separation occurred on all simulations, and thus majority of the drag were in the form of frictional drag. Wall shear stress plots of the designs with the highest drags are shown in Figure 3. A wall shear stress of zero would have indicated the start of flow separation. According to Finnemore and Franzini (2006), if the tangential velocity of the cylinder becomes greater than twice the free stream velocity, a ring of fluid is dragged around the cylinder and the stagnation point is moved away from the cylinder surface.Frictional loss in the boundary layer increases with wall shear stress, which is directly correlated to the velocity gradient across the boundary layer. A rough surface, instead of creating a velocity gradient with the help of viscous forces in the fluid, possibly drags the whole boundary layer with it, thus effectively reducing, if not eliminating, the velocity gradient in the boundary layer. On the other hand, a bump may be visualized as a large sand grain or roughness element, which should similarly explain its significance in reducing drag at lower speed ratios. The Reynolds number in this study was from 3.8×10^4 to 7.6×10^4.

4.3 Lift-to-drag ratio

The relationship between LDR and speed ratio is parabolic. Bumps gave the highest LDR's across majority of the speed ratios as a consequence of having both good lift enhancement and drag reduction properties. The interaction of a frustum cylinder with dimples also yielded good LDR's, due to the same reason. However, caution must be taken when interpreting these results as a high LDR may be obtained even with just minimal lift, as long as the equivalent drag is also smaller. Other modifications, such as dimples on a regular cylinder and increasing surface roughness, did not yield good LDR responses because they either had good lift enhancement but poor drag reduction or vice versa. It is also observed that the presence of grooves shifts the peak of the LDR curve to a lower speed ratio.

Table 2: Control variables with corresponding levels in the full-factorial DOE

Factor	Cylinder Shape	Dimple or Bump	Surface Roughness	Grooves
Levels	Regular	Circular Dimple	Smooth	Helical
	Frustum	Circular Bump	Rough	Straight
		None		None

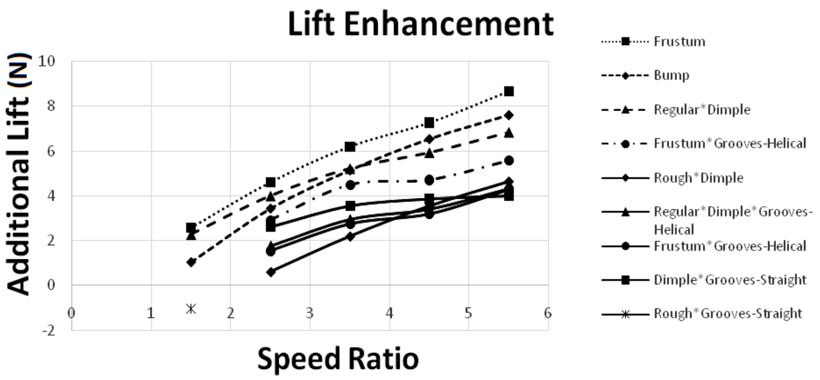

Figure 2: Lift enhancement from top modifications based on scaled estimates from JMP

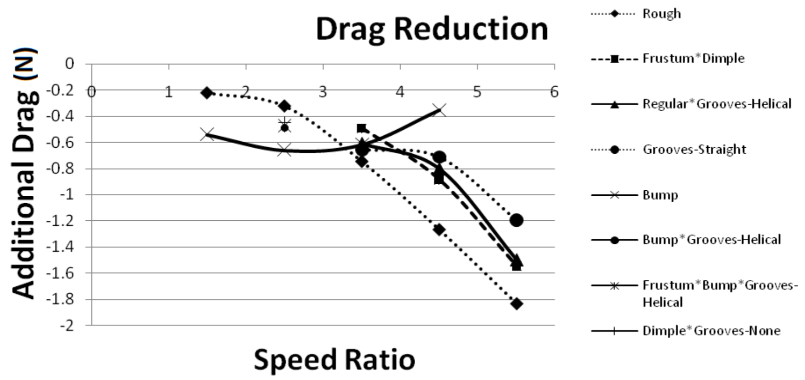

Figure 3: Drag reduction from top modifications based on scaled estimates from JMP

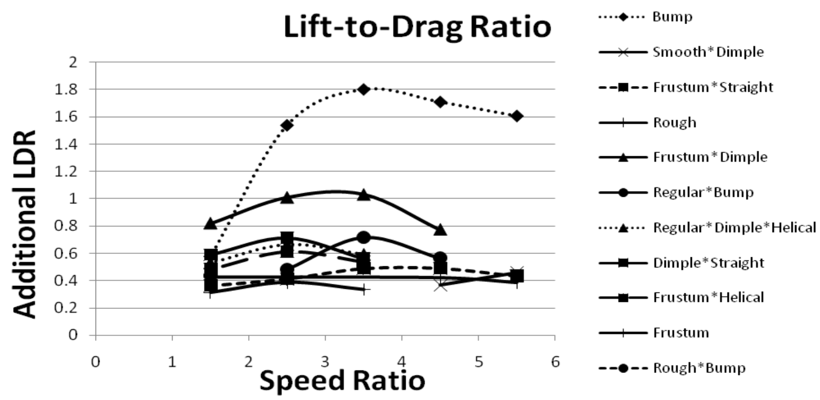

Figure 4: Additional LDR from top modifications based on scaled estimates from JMP

5. Conclusion and future work

Using computer modelling and simulation, the authors were able to execute a full-factorial DOE to compare and analyze the effects of different surface modifications to the lift and drag generated on a rotating cylinder. Overall, it is possible to obtain higher lift and lesser drag by employing surface modifications. It was also observed that the main effects are more significant than the interactions. The use of bumps and a frustum cylinder are most promising in improving the performance of a Magnus rotor. It is important to take note that the insights derived from this study are highly dependent on the dimensions of the modifications tested. A second DOE utilizing continuous control variables, which takes off from the findings of this first

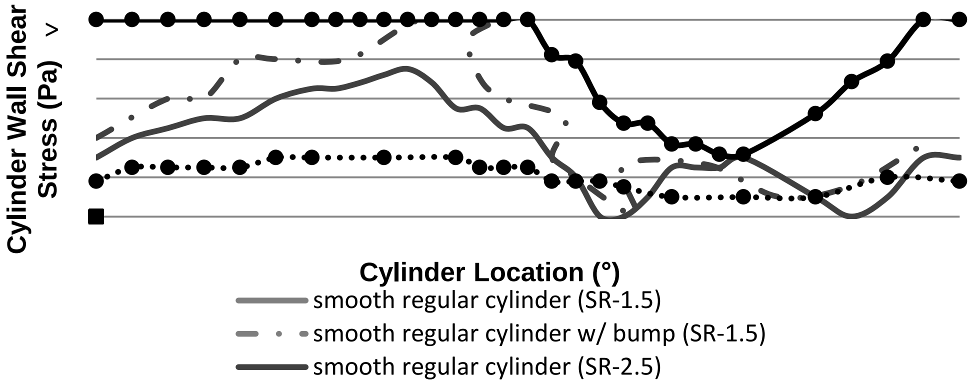

Figure 5: Wall shear stress plot of the best drag reducing modifications compared to a regular cylinder

DOE, is underway as of writing by the same authors and aims to capture the effect of varying the size of the modifications. With regards to finding the optimum surface modification, the criterion to be used is also arguable. Considering only LDR could sacrifice the lift force and eventually the energy produced in a wind turbine. It is also arguable if equivalent importance should be given to both lift and drag during optimization. This will be tackled in an upcoming paper by the authors.

References

Barbera L., Guerrero A., Crespo A., Gonzalez-Prida V., Guillen A., Gomez J., Sola A., 2013, State of the art of maintenance applied to wind turbines, Chemical Engineering Transactions, 33, 931-936.

Becker W., Valeva V., Petrova T., Ivanova J., 2014, Technical damage in wind rotor blade under static load at environment conditions, Chemical Engineering Transactions, 42, 91-96.

Bubbico R., Falleni G., Mazzarotta B., 2014, CFD simulation of the release of ln2 inside a tunnel, Chemical Engineering Transactions, 36, 325-330.

Butt U., Egbers C., 2013, Aerodynamic characteristics of flow over circular cylinders with patterned surface, International Journal of Materials, Mechanics, and Manufacturing, 1(2), 121-125.

Cuervo N., Murillo C., Dufaud O., Bardin-Monnier N., Skali-Lami S., Remy J.F., Auzolle P., Perrin L., 2014, Combining CFD simulations and PIV measurements to optimize the conditions for dust explosion tests, Chemical Engineering Transactions, 36, 259-264.

Finnemore E.J., Franzini J.B., 2006, Fluid Mechanics with Engineering Applications, 10th Ed, McGraw-Hill, New York, United States of America.

Huang S., 2010, Cylinder drag reduction by the use of helical grooves, Proceedings of the HYDRALAB III Joint User Meeting., 3, 115-118.

Mara B.K., Mercado B.C., Mercado L.A., Pascual J.M., Lopez N.S., 2014a, Development and validation of a CFD model using ANSYS CFX for aerodynamics simulation of Magnus wind rotor blades, Proceedings of the 7thIEEE HNICEM Conference, Puerto Princesa, Philippines, DOI: 10.1109/HNICEM.2014.7016231

Mara B.K., Mercado B.C., Mercado L.A., Pascual J.M., Lopez N.S., 2014b, Preliminary turbulence model validation for flow across rotating cylinders using ANSYS CFX, Proceedings of the 7th IEEE HNICEM Conference, Puerto Princesa, Philippines, DOI: 10.1109/HNICEM.2014.7016266

Murakami N., Ito J., 2009, Magnus type wind power generator, US Patent 7504740 B2.

Rosa L., Pederiva L., Maurina G., Beal L., Torres A., Sousa M., 2014, CFD analysis of the effect of baffle plates on the fluid flow in an anaerobic sequencing batch reactor, Chemical Engineering Transactions, 38, 133-138.

Seifert J., 2012, A review of the Magnus effect in aeronautics, Progress in Aerospace Sciences, 55, 17-45.

Sedaghat A., 2014, Magnus type wind turbines: Prospects and challenges in design, Renewable Energy, 62, 619-628.

Srivastav D., 2012, Flow control over airfoils using different shaped dimples, International Proceedings of Computer Science & Information Tech, 33, 92-98.

Sun X., Zhuang Y., Cao Y., Huang D.,Wu G., 2012, A three-dimensional numerical study of the Magnus wind turbine with different blade shapes, Journal of Renewable and Sustainable Energy, 4(6), 063139, DOI: 10.1063/1.4771885.