25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-1115
INFLUENCE OF CROSSWINDS ON PROPELLERS PERFOMANCE

Nícolas Lima Oliveira
Lucas Machado Rocha
Patricia Habib Hallak
Manuel A. Rendón
Universidade Federal de Juiz de Fora, Rua José Lourenço Kelmer, sn, Juiz de Fora, MG
nicolas_limaoliveira@yahoo.com.br; lucas.machado@engenharia.ufjf.br; patricia.hallak@ufjf.edu.br; manuel.rendon@ufjf.edu.br
Abstract. *Propellers are providing throughout history a notorious contribution to the development and construction of various engineering systems, such as airplanes, helicopters, drones, ships, submarines and even land vehicles. Specifically for aircrafts, this device is responsible for providing thrust, converting the power generated from an engine. This work aims to study the performance of propellers through computational fluid dynamics (CFD), considering the influence of crosswinds. After validating the CFD model, two different small propellers were used in the simulations, comparing the results without crosswinds from the reference experimental results. Three simulation conditions were used, varying the intensity of the crosswind and the propeller advance ratio. After the simulations, the perception is that crosswinds decreases propeller efficiency for higher advance ratio values, whereas it had no negative effect on static condition.*

Keywords: *Propeller efficiency, CFD, crosswinds.*

1. INTRODUCTION

The use of propellers for flying vehicles began in the second half of the eighteenth century, at first for propulsion of dirigibles. At this time, there were few researches about this subject and little was known about its principles (Bass, 1983). Since then, several studies have been carried out to understand the functioning of this element. These were essential for the development of early aircraft in the early nineteenth century, and to this day propellers are used in propulsion systems, from the simplest to the most complex of them.

In general, the main objective of these methods is to evaluate the performance of the propeller in terms of thrust, torque, power and efficiency. There are currently in literature different methods for propeller design and analysis, based on different levels of desired sophistication.

By tracing a brief history, the first methods were originated from analytic studies, beginning with the momentum theory developed by Rankine (1865) and Froude (1911) that follows the basic principle of the operation of a propeller, which is to supply a certain amount of movement to the air that passes through it. Further, a major contribution to the development of propellers theories came from the work of Betz (1919). He developed, in order to incorporate the effects of the circulation to the wings of the planes, the lifting-line theory. Later, Glauert (1935) presented a methodology that incorporates two basic models, the momentum theory with the blade element theory, called blade element momentum theory (BEMT). Along with Theodorsen (1948), proved that it is possible to obtain the distribution of the coefficient of support in the blades for different levels of loading, by analyzing the helical vortex sheet in a distant wake of the propeller, when considering the contraction of it.

Nowadays there are models that consider viscous, three-dimensional and incompressible or compressible flows. The simulations of this nature are called Computational Fluid Dynamics (CFD) and are used along with computational resources. As it simplify complex problems and allows the simulation of some problems that mathematically do not have exact analytic solution, this work used the CFD to analyze the behaviour of propellers that have crosswinds incidence, a very common situation during a flight, which can change the performance and efficiency of this device.

2. MODEL VALIDATION

Specific algorithms used in CFD deal with the inherent problems of the discretization of the Navier Stokes equations, as the non-linearity of the advective term, the coupling of terms of pressure and velocity, the interaction between the fluid and the moving structure, insertion of turbulence models that contemplate the various temporal and spatial scales as a function of the Reynolds parameter and also the compatibility of interfaces between stationary and rotating domains, either by direct topological changes in the mesh or by geometric simplifications (Beaudoin and Jasak, 2008). Due to its complexity, CFD requires appropriate computational resources and demands a validation of the model to be adopted in the desired simulation. Therefore, the student version of the software ANSYS was used in this work. Specifically for the the

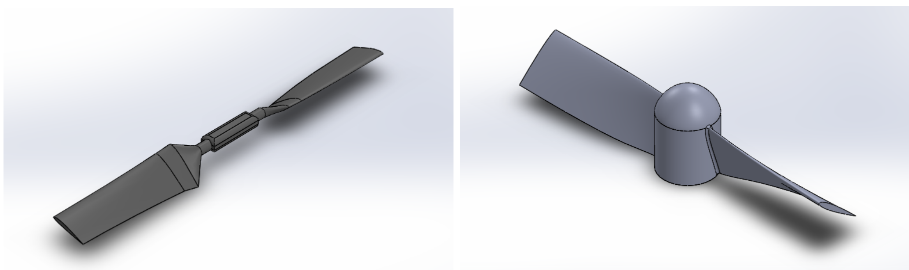
CFD problem, the ANSYS / CFX is a platform that uses an advanced solver with pre-processing and post-processing. Can be applied to transient and stationary problems, laminar or turbulent flow, for subsonic, transonic or supersonic situations (Ansys Inc, 2012).

Various numerical techniques were employed in this research. The discretization of the partial differential equations was fulfilled by the finite volume method, and the convective term was treated by the high resolution algorithm, suggested by Barth and Jespersen (1989), for the upwind scheme. To consider the axis rotation transmitted to the propeller, the relative movement is that of rotation and a moving reference frame was adopted. For coupling between pressure and velocity, a non-staggered mesh method was used, where control volumes are identical for all transport equations, based on the non-staggered method of Rhie and Chow (1983), and adapted by Majumdar (1988). In the treatment of interfaces, due to its simplicity and applicability, the General Grid Interface (GGI) technique (Galpin *et al.*, 1995) (Beaudoin and Jasak, 2008) (Belamri *et al.*, 2005) (Jasak and Beaudoin, 2011) was used, adopting the Frozen Rotor model. That treats the flow from the fixed component to the rotary by changing the reference system without making averages, which allows local characteristics of the flow, such as recirculations and shock waves, to be transported through the interface.

In order to validate further results obtained from ANSYS / CFX, simulations were performed for the DA4002 (Brandt and Selig, 2011) and Purdue (Witkowski *et al.*, 1989) propellers. The results in terms of efficiency, thrust coefficient and power coefficient were compared with those of the respective references.

2.1 Experimental results comparison

The DA4002 propeller is a two blades propeller with fixed-pitch of 0.17145 m and diameter of 0.2286 m, as shown in figure 1.(a). The airfoil used throughout its extension is the SDA1075. The simulations considered 2000 rpm, according to the literature, and used the k-ω turbulence model. The mesh contains 259150 tetrahedral elements, a cylindrical rotational domain with diameter of 25,146 cm and height of 9,144 cm, and a cubic stationary domain with 182,88 cm edge. In addition to the Frozen Rotor condition adopted in the interface, a no-slip wall was applied to the propeller geometry faces.

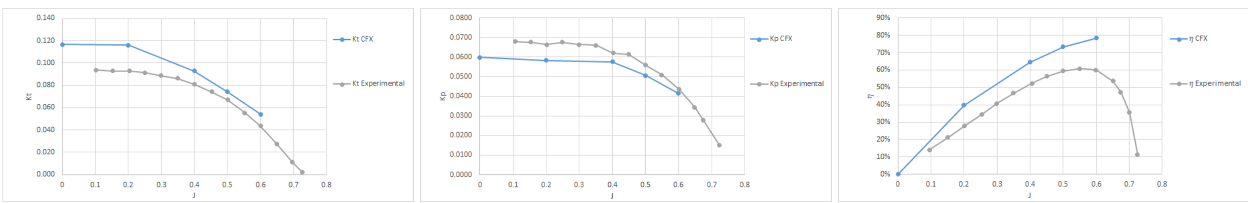


(a) DA4002 geometry (b) Purdue geometry

Figure 1: Geometry of the propellers simulated

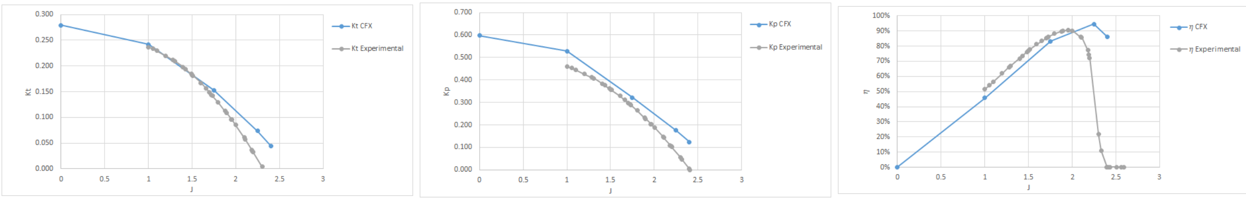
The Purdue propeller is also a two blades propeller with fixed-pitch, but has a 0.3048 m diameter and pitch of 45.4^o. The geometry representation of this propeller is at figure 1.(b). It has the NACA 0010 airfoil used throughout its extension, considering a 2880 rpm for the simulations and using the k-ϵ turbulence model. The mesh contains 259150 tetrahedral and almost the same characteristics adopted for the DA4002 mesh, varying the dimensions of the rotational domain to a 33.528 cm diameter and 12.192 cm height, and the dimensions of the stationary domain to a 243.84 cm edge cube.

Through the variation of advance ratio (J) , it is possible to extract from ANSYS / CFX the results of interest for both propellers. Figure 2 shows the results for the DA4002 propellers and figure 3 shows the Purdue results. From those, it is possible to affirm that the model adopted in the CFD simulation is representative in comparison to the experimental result.



(a) Kt (b) Kp (c) η

Figure 2: DA4002 propeller results



(a) Kt (b) Kp (c) η

Figure 3: Purdue propeller results

2.2 Mesh Information

- DA4002

The total number of mesh elements used in ANSYS/CFX was of 259150 tetrahedra. For the stationary domain, the mesh was generated using Table 1 specification in ANSYS/Mesh

Item	Value
Size Function	Adaptive
Center Relevance	Fine
Element Size	Default
Transition	Slow
Initial Speed Seed	Default
Span Angle Center	Fine

Table 1: ANSYS/Mesh specification for DA4002 propeller stationary domain.

The parameters for the rotating domain are displayed in the Table 2 and is represented in Figure 4.

Item	Value
Size Function	Adaptive
Center Relevance	Moderate
Element Size	Default
Transition	Slow
Initial Speed Seed	Default
Span Angle Center	Fine

Table 2: ANSYS/Mesh specification for DA4002 propeller rotating domain.

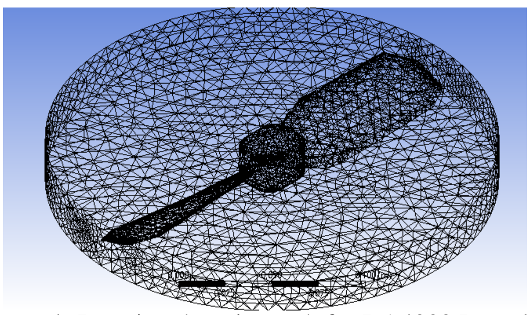


Figure 4: Rotating domain mesh for DA4002 Propeller

- Purdue

The total number of mesh elements used in ANSYS/CFX was of 494607 tetrahedra. For the stationary domain, the mesh was generated using Table 3 specification in ANSYS/Mesh

Item	Value
Size Function	Adaptive
Center Relevance	Fine
Element Size	Default
Transition	Slow
Initial Speed Seed	Default
Span Angle Center	Fine

Table 3: ANSYS/Mesh specification for Purdue propeller stationary domain.

The parameters for the rotating domain are displayed in the Table 4 and is represented in Figure 5.

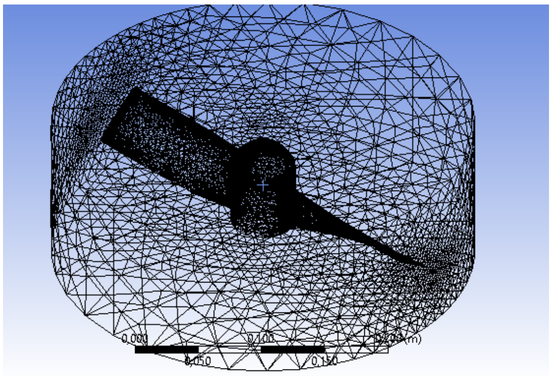


Figure 5: Rotating domain mesh for Purdue Propeller

Item	Value
Size Function	Proximity and Curvature
Max Face Size	5e-2m
Mesh Defeaturing	Sim
Defeature Size	8e-5m
Growth Rate	1,2
Min Size	1e−3m
Max Tet Size	0,1m
Curvature Normal Angle	18^0
Proximity Min Size	1e−3
Num Cell Across Gap	3
Proximity Size Function Source	Faces e edges

Table 4: ANSYS/Mesh specification for DA4002 propeller rotating domain.

3. CROSSWIND INFLUENCE

The objective of this section is to evaluate the influence of crosswinds on propellers performance. The meshes used in this section correspond to those used in the previous section. The idea of this section is to establish a qualitative study of the influence of this condition on propeller performance.

The methodology used can be summarized as:

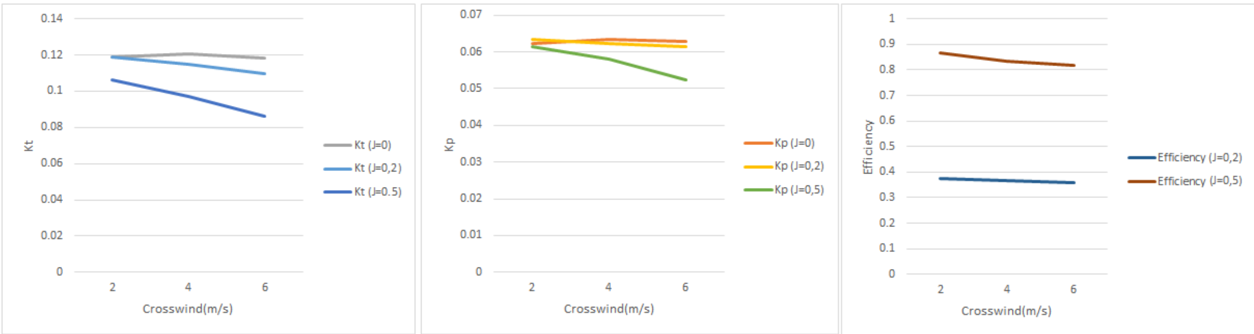
- The boundary conditions were changed to consider crosswind interference. In this case, was included one side inlet in the stationary domain. Faces that do not receive textit inlet condition, receive textit open condition, as in the previous section;
- Different values of crosswind (V_t) were adopted and two situations were considered: static, referring to the situation of the propeller without speed of advance, and dynamic, with two speeds of advance. Advance Ratio were extracted from the efficiency curve and correspond to the highest efficiency value. One of them corresponds to the one of greater efficiency, obtained from the previous section. The second speed is an intermediate speed value;
- In order to evaluate the intensity of the crosswind in relation to the flight conditions, a reference speed (V_R) is adopted, which corresponds to the speed of higher efficiency.

For each study, the turbulence models used were:

- DA4002 : k-ω;
- Purdue: k-ϵ.

3.1 DA4002

Table 5 shows the values of the crosswind (V_t) and the values of advance ratio J and V_J adopted for the DA4002 and Purdue propellers. The results are graphically represented on Figure 6 and 7.

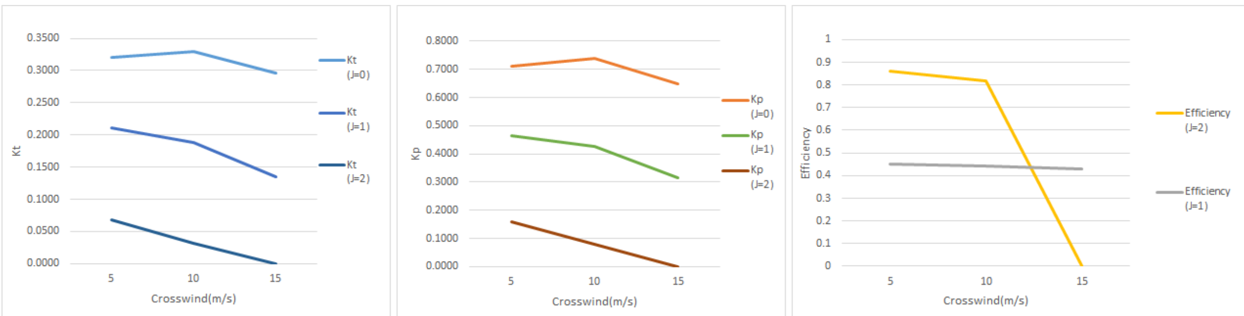


(a) (b) (c)

Figure 6: Crosswind influence on (a) thrust coefficient (b) power coefficient (c) efficiency - DA4002

3.2 Purdue

Table 5 shows the values of the crosswind (V_t) and the values of advance ratio J and V_J adopted for the DA4002 propeller. The results are graphically represented on Figure 7.



(a) (b) (c)

Figure 7: Crosswind influence on (a) thrust coefficient (b) power coefficient (c) efficiency - Purdue

DA4002					Purdue				
Case	$V_t(m/s)$	J	$V_J(m/s)$	$(V_t/V_R)\%$	Case	$V_t(m/s)$	J	$V_J(m/s)$	$(V_t/V_R)\%$
Static	0	0	-	-	Static	0	0	-	-
	2	0	-	52		5	0	-	15
	4	0	-	104		10	0	-	30
	6	0	-	157		15	0	-	46
	8	0	-	210		20	0	-	61
Dynamic	2	0,2	1,524	52	Dynamic	5	1	14,63	30
	4	0,2		104		10	1		46
	6	0,2		157		15	1		61
Dynamic	2	0,5	3,810	52	Dynamic	5	2	29,26	30
	4	0,5		104		10	2		46
	6	0,5		157		15	2		91

Table 5: Crosswind and advance ratio value for DA4002 and Purdue propellers.

3.3 Results analysis - DA4002 Propeller

According to obtained results for crosswind influence of DA4002 propeller, presented in Figure 6, it's possible to conclude:

- Analysis of V_t influence on efficiency

  The references values shall be η = 0,9 for $J = 0,5$ and $\eta = 0,4$ for $J = 0,2$, obtained from simulations without crosswinds. It's possible to realize:

  - There is a slight drop in efficiency for the two flight conditions. Apparently, this drop in efficiency for is approximately constant for the values of crosswinds studied.
  - Under the conditions studied, there is no loss of stability.

- Static case

  The adopted reference values were:$K_{T0} = 0,14$ and $K_{P0} = 0,068$. From Figure 2 it's possible to note that values were slightly different from those of reference.

- Simulation with advance ratio $J = 0,2$

  Adopted reference values were $K_T = 0,12$ e $K_P = 0,064$.

  - Thrust coefficient presents a linear fall in all considered air speeds. For $V_t = 157\% * V_R$, $K_T = 0,1098$
  - The power coefficient value remains almost constant.

- Simulation with advance ratio $J = 0,5$

  he adopted reference values were: $K_T = 0,11$ e $K_P = 0,063$.

  - The thrust coefficient drops about 25%, in a almost constant rate, between the air speeds studied.
  - The power coefficient drops about 17%, in a almost constant rate, between the air speeds studied.

3.4 Results analysis - Purdue Propeller

According to obtained results for crosswind influence of Purdue propeller, presented in Figure 7, it's possible to conclude:

- Analysis of V_t influence on efficiency

  The references values shall be η = 0,95 for $J = 2$ and $\eta = 0,5$ for $J = 1$, obtained from simulations without crosswinds. It's possible to realize:

  - There is a drop in efficiency for the two flight conditions.
  - For efficiency $J = 1$, the drop is small for all cases.
  - The efficiency drop for $J = 2$ is about 10% for $V_t = 46\% * V_R$. For $V_t = 91\% * V_R$, a sharp fall in the efficiency value to 0% is observed. This phenomenon can be attributed to the detachment of the boundary layer.

- Static case

  The adopted reference values were:$K_{T0} = 0,2889$ and $K_{P0} = 0,6176$. From Figure 3 it's possible to note that values were slightly different from those of reference.

- Simulation with advance ratio $J = 1$

  he adopted reference values were: $K_T = 0,2458$ e $K_P = 0,5473$.

  - The thrust coefficient drops about 45%, in a almost constant rate, between the air speeds studied.
  - The power coefficient drops about 47%, in a almost constant rate, between the air speeds studied.

- Simulation with advance ratio $J = 2$

  he adopted reference values were: $K_T = 0,1071$ e $K_P = 0,2301$.

  - The thrust coefficient falls steadily in the studied ranges up to 0 %
  - The power coefficient follows the tendency of the thrust coefficient.

4. CONCLUSIONS

In Section 2 of this article was performed a verification of a CFD model using experimental results for two different propellers. In section 3, the same CFD model was used to predict crosswinds influence on propeller efficiency.

The results indicated that the efficiency of the propellers decreases with the increase of the crosswinds, as expected. Unlike other methods, the CFD is a tool with great versatility, able to obtain good results in nontrivial situations.

5. ACKNOWLEDGEMENTS

Authors would like to thank FAPEMIG and EMBRAER for funding the resources of the research project TEC-APQ-03593-17, about what the present work was developed. It's also important to thank CAPES for the masters scholarships and PPGMC-UFJF, PEC-UFJF and MAC-UFJF for the support provided.

6. REFERENCES

Ansys Inc, 2012. "Ansys cfx-solver modeling guide". *ANSYS CFX Release*, Vol. 14, p. 153.

Barth, T. and Jespersen, D., 1989. "The design and application of upwind schemes on unstructured meshes". In *27th Aerospace sciences meeting*. p. 366.

Bass, R., 1983. "An historical review of propeller developments". *The Aeronautical Journal*, Vol. 87, No. 867, pp. 255–267.

Beaudoin, M. and Jasak, H., 2008. "Development of a generalized grid mesh interface for turbomachinery simulations with openfoam". In *Open source CFD International conference*.

Belamri, T., Galpin, P., Braune, A. and Cornelius, C., 2005. "Cfd analysis of a 15 stage axial compressor: Part I - Methods". In *ASME Turbo Expo 2005: Power for Land, Sea, and Air*. American Society of Mechanical Engineers, pp. 1001–1008.

Betz, A., 1919. "Schraubenpropeller mit geringstem energieverlust". *Gottinger Nachrichten*, pp. 193–213.

Brandt, J. and Selig, M., 2011. "Propeller performance data at low reynolds numbers". In *49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*. p. 1255.

Froude, R.E., 1911. "The acceleration in front of propeller". *Transactions of the Institution of Naval Architects*, Vol. 53, pp. 139–171.

Galpin, P., Broberg, R. and Hutchinson, B., 1995. "Three-dimensional navier stokes predictions of steady state rotor/stator interaction with pitch change". In *Proceedings of 3rd Annual Conference of the CFD Society of Canada, Banff, AB, Canada*. Vol. 3.

Glauert, H., 1935. "Airplane propellers". In *Aerodynamic theory*, Springer, pp. 169–360.

Jasak, H. and Beaudoin, M., 2011. "Openfoam turbo tools: From general purpose cfd to turbomachinery simulations". In *ASME-JSME-KSME 2011 Joint Fluids Engineering Conference*. American Society of Mechanical Engineers, pp. 1801–1812.

Majumdar, S., 1988. "Role of underrelaxation in momentum interpolation for calculation of flow with nonstaggered grids". *Numerical Heat Transfer*, Vol. 13, No. 1, pp. 125–132.

Rankine, W.J.M., 1865. "On the mechanical principles of the action of propellers". *Transactions of the Institution of Naval Architects*, Vol. 6, pp. 13–39.

Rhie, C. and Chow, W.L., 1983. "Numerical study of the turbulent flow past an airfoil with trailing edge separation". *AIAA journal*, Vol. 21, No. 11, pp. 1525–1532.

Theodorsen, T., 1948. *Theory of propellers*. McGraw-Hill Book Company.

Witkowski, D.P., Lee, A.K. and Sullivan, J.P., 1989. "Aerodynamic interaction between propellers and wings". *Journal of Aircraft*, Vol. 26, No. 9, pp. 829–836.