Mechanics and Mechanical Engineering
Vol. 20, No. 4 (2016) 579–593

# Development of a CFD model for propeller simulation

Mateusz Stajuda
Maciej Karczewski
Damian Obidowski
Krzysztof Jóźwik
*Lodz University of Technology*
*Institute of Turbomachinery*
*mateusz.stajuda@p.lodz.pl*

Received (29 September 2016)
Revised (6 November 2016)
Accepted (11 December 2016)

The article presents a development of numerical model for a single propeller simulation and comparison of obtained results with experimental data available from a test campaign in scale 1:1. Described simulation is a steady state computation taking advantage of Multiple Reference Frame model implemented in Ansys CFX. The paper includes an analysis of rotating domain thickness influence on numerical values of thrust and power. The results indicate that this type of simulation may be sensitive to the sizing of rotating domain especially when disc solidity is low, or when the number of blades is 2, a frequent situation in all electric flight vehicles. The analysis shows that performing simulations, using one domain sizing, for a number of flight scenarios requiring analysis of a few rotational speeds can produce unintuitive results. Therefore, it is suggested to calibrate the model, preferably by experimental results.

*Keywords*: propeller, CFD, electric airship, UAV, manned flight, experiment.

## 1. Introduction

The emergence of new all–electric aircraft concepts has resulted in an increased interest in propellers design and optimization. New vehicle designs are presented by technological startups, but also by major aviation companies, believing in a development of this market sector. Promising concepts were presented for example by Agusta Westland and E–volo. The Project Zero and Volocopter are presented in Fig. 1a and 1b. Many other electric aircrafts are to be presented in the oncoming years. The growth in the market of electric aircrafts is encouraged by European Union Clean Sky 2 program [1], which goal is to reduce the fuel consumption, carbon dioxide and nitric oxides emissions and to decrease the noise footprint of the air fleet. Created all electric or hybrid aircrafts are characterized by a much simpler design, in terms of mechanical components, than contemporary helicopters and often higher number of propulsion elements. The use of electric motor instead of a fuel powered engines offers the possibility of relatively easy changes in rotational speed of the propellers, which allows the designers to trim the aircraft via the rotational speed variation. This, in turn, changes the requirements which have to be met by the propeller design, as it should be efficient for a wide range of rotational speeds.

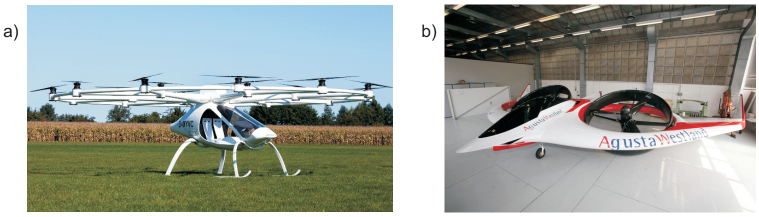

**Figure 1** Examples of all-electric aricarafts: a) Volocopter by E-volo [3], b) Project Zero by Agusta Westland [2]

The rise of momentum theory, widely used for the general design of propeller blades, dates to the beginning of $20^{th}$ century with the works of Betz [4] and Glauert [5]. Betz initiated a general theory of flow through rotating disc, while Glauert enriched it with terms accounting for momentum loss, including the tip loss factor. The tests on different blade geometries and blade configurations were performed throughout 1950s, 60s, 70s and 80s, for example in NACA research centers by Gessow [6], Hartman and Biermann [7] or Harrington [8]. A blade shape, including the chord distribution, planform, twist and pitch, was the source of interest. The studies also included ducted and coaxial configurations. The experimental research provided necessary data for theoretical development in the field of propeller design, leading to a creation of analytical methods such as Blade Element Theory (BET), Blade Element Momentum Theory (BEMT) or Free Vortex Method (FVM). These methods are fast and may be used for the design and performance validation of the propellers [9]. Despite their reliability and quickness, they always require input parameters from experiment to produce accurate results, which make them difficult to use in order to precisely assess new designs without a complementary experimental study. Generally, a computational fluid dynamics (CFD), offering a wide range of modelling capabilities is the most precise design validation method. The generality of CFD codes and necessity of introducing turbulence closure in Navier–Stokes equations make it difficult to create a proven computational model able to produce coherent and reliable results, especially when highly unsteady flows, as in propeller simulations, are expected. The CFD simulation reliability depends on many aspects, including computational domain sizing, mesh size and quality as well chosen solution type (steady state or transient) and physical formulation (type of equation solved, additional wall formulations and corrections). A validated numerical model offers possibility of a much cheaper and more thorough analysis than experiments. The most reliable information on propeller performance can be delivered by full–scale tests, which are very demanding and expensive, therefore they nowadays are performed by mainstream aircraft manufacturers having sufficient funds for such campaigns. The majority of papers concerning the topic of propellers, such as those of Juhasz et al. [10] or Syal and Leishman [11] relies on the experimental tests described, among others, in sources [6, 7, 8]. The articles concerning validation of new propeller design based on full scale tests for a manned VTOL flight were not found.

The aim of this paper is to describe the development of numerical model used for simulations of a single propeller. The procedure of domain sizing, meshing approach and numerical setup is well described. The article presents a study on the influence of rotating domain size on the calculated thrust and power magnitude for a discreet model prepared in ANSYS CFX. A short study of an interface type is also performed. The obtained results are compared with full scale experiment results to draw conclusions about an appropriate domain sizing.

## 2. Method of CFD Modelling

The modelling approach presented in this paper is a result of authors' experience gained during the work in the field of numerical simulations and propeller designs. The goal was to assess the importance of model parameters on obtained results. This can lead to a reliable and versatile simulation setup which may be used for tests of different propeller designs and optimization of the blade shape without the need for expensive, profound experimental studies.

### 2.1. *Simulation approach*

A steady state simulation taking advantage of Multiple Reference Frame (MRF) approach is the model used by the authors [12]. This model allows one to connect a rotating domain containing the propeller blade and its proximity with a complementary stationary domain representing the volume of air further from the propeller. The two domains are connected using domain interfaces of an appropriate type, which transfer data between stationary and rotating frames This method of simulation introduces certain approximation, as the simulation is stationary and the blade itself does not rotate with respect to the surrounding domain. The method is, however, industrially accepted for simulating propellers and provides reliable results.

### 2.2. *Computational domain*

The domain (Fig. 2a) consisted of rotating propeller disc (marked in blue), motor (marked in red) and stationary surrounding domain. Only half of the physical domain was computed as a two–bladed propeller was analysed thus allowing for rotational periodicity. The simulation did not include the test bench elements, present during experimental campaign. The propeller blade was placed in the centre of the computational domain (Fig. 2 b). The surrounding domain size was selected on the basis of numerical experiments performed in previous computational campaigns [13]. The domain radius was five times bigger than the radius of the propeller and domain height was 10 times larger than the diameter of the propeller. Such sizing ensured that the boundary conditions were sufficiently far from the propeller disc.

## 2.3. *Mesh*

The mesh for the simulation was generated in two different mesh generators. For the rotating domain, where precise solution was crucial, a structural hexahedral mesh in ICEM CFD was created. The size of the mesh elements was chosen to provide acceptable mesh quality and to allow one to fully solve the boundary layer at the blade, which in case of ANSYS CFX, requires y+ value below 2 [12]. The propeller mesh consisted of around 200 elements chord–wise for capturing the pressure changes along the blade chord, which have important influence on the propeller thrust and around 300 elements span–wise, which offer much lower resolution, however sufficient to account for pressure gradients expected in this direction. A mesh clip along the blade chord is presented in Fig. 3. The mesh was also refined in the region of leading edge and behind the trailing edge, where the blade wake was expected. The final mesh of the rotating domain comprised of 8 million nodes.

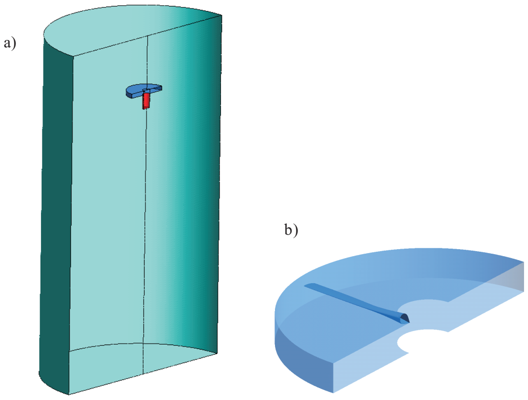

**Figure 2** a) General view of the computational domain, b) view of the rotating domain

The mesh in the surrounding domain was chosen to be the hybrid one. Hexahedral elements were used in the propeller disc proximity for better shape and size fit of elements on both sides of the interface (Fig. 4). Further from the propeller, a tetrahedral mesh was created. It allowed for a standard growth rate of 1.2, high quality and reasonable aspect ratio of the elements. A use of tetrahedral mesh also gives possibility of easy changes in a surrounding domain, for example introducing test bench elements, without a need for changes in the meshing strategy. The mesh in a surrounding domain was created using ANSYS Meshing, which requires less labour and allows one to generate the mesh of more complicated geometrical models than structural mesh created with the use of ICEM CFD.

The sizing in the regions of blade induced wake path, near the propeller blade tip and in the areas where the prop induced eddies dissipate by exchanging energy with the quiescent surroundings was the most important in the outer mesh. For this purpose a growth rate from the hexahedral mesh surfaces was decreased from default 1.2 to 1.05 and sizing using body of influence was added (see Fig. 5). The outer mesh was defined to include as little nodes as possible, to decrease the need for computational power and minimize the computational time needed to converge the solution. The total number of nodes in outer domain, while maintaining expected sizing and good elements quality, was below 3 million.

## 2.4. *Mesh independence*

A mesh independence and rotating disc thickness study were performed for this computational model. Both tests included three variants, as they allowed one to capture the trend and draw conclusions. Mesh independence study was only performed for the rotating propeller domain. The number of nodes varied from 5 to 12 million, as the number of elements along the blade span and blade chord was changed. The sizing of the first layer, therefore y+ value, remained relatively constant. Results of the study showed that the difference between observed parameters for the least and the most dense meshes were below 1%. Consequently, the mesh with the lowest number of nodes was chosen. Attempts in further decrease of the mesh size resulted in low quality of the mesh elements, therefore these meshes were not used.

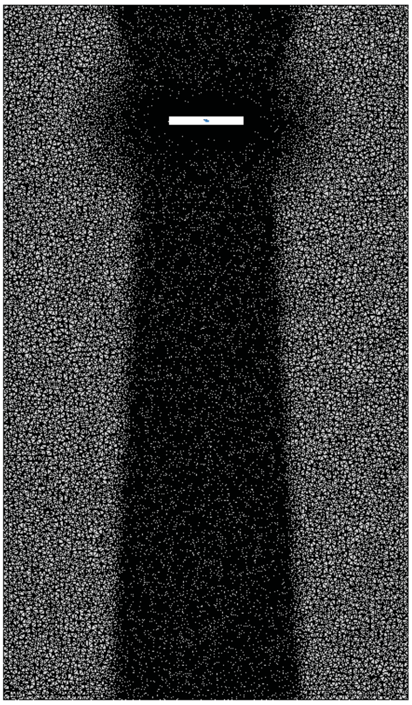

**Figure 3** Mesh clip inside the rotating domain

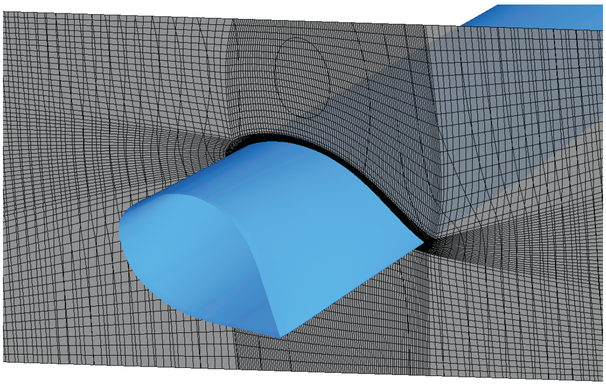

**Figure 4** Mesh structure in the proximity of propeller domain; white region is where the propeller mesh is inserted

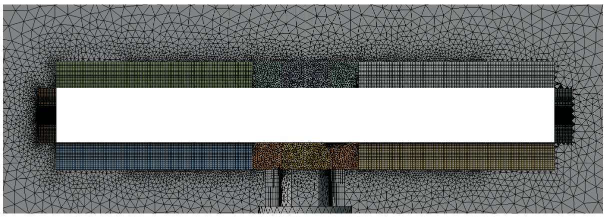

**Figure 5** Mesh clip of the stationary surrounding domain

## 2.5. *Numerical model setup*

The Reynolds–Averaged Navier-Stokes simulation prepared for propeller analysis was treated as fully turbulent. In the simulation, the Shear Stress Transport turbulence model was used. The SST turbulence model formulation in ANSYS CFX provides robustness and ease of application as it allows for automatic wall treatment, i.e. changing from full solution to boundary layer modelling depending on grid resolution in the model [12].

For proper thrust and power computation, the exact pressure distribution and possible separations on the blade were very important, therefore full boundary layer resolution was desirable with y+ value below 2 on the whole propeller. Due to the aspect ratio limits and quality of resulting elements, the zone where y+ was below 2 was limited to the outer part of the blade, where majority of the thrust is generated and majority of the power is consumed. The simulation was performed for absolute pressure of 101325 placeStatePa. The temperature was set to 20 $^o$C. The fluid entering the domain had low turbulence intensity (<1%). The total energy model was used in simulations, which ensured that the compressibility effects were taken into account.

The multiple frame of reference (MFR) model used for this simulation introduces certain simplifications, due to which its application is not straightforward. In steady state simulation, the domain does not physically move, but only the velocity resulting from domain rotation is added in the appropriate nodes. This leads to a situation, where the propeller disc domain is numerically rotating, while the blade remains stationary with respect to the surrounding domain. For this reason, the modelling of wake development downstream the blade differs along with the rotating disc thickness. Thus each MFR model requires thorough study of disc thickness influence on the resultant values of thrust and power.

### 2.6. *Disc thickness study*

The disc thickness study appears to be very important for the MRF model in the two bladed propeller simulation due to limited mixing of the trailing vortices. The vorticity field shed by a propeller with such a low number of blades is thus under the strong influence of the quiescent surroundings. A rapid transition from a rotating frame of reference to a stationary one may result in an unphysical numerical diffusion of the conservative quantities. In this sense, the MRF introduces a certain simplification, as in steady state simulation, the domain motion is simulated by adding appropriate velocity components as source terms in N-S equations, while the domain and, therefore, mesh remain in the constant position. To transfer the fluxes of conservative parameters between rotational and stationary domains, the possible interface options are as follows: frozen rotor, which directly transfers the data between adjacent cells through the interface and stage, which performs averaging of conservative quantities over the control surfaces building the interface and then transfers the resulting values over. In case when the blade chord is only a small part of the domain, a frozen rotor interface is recommended option [12]. Even though the interfaces provide correct data transfer for basic flow quantities, they may cause a loss of information and changes in certain complex flow structures induced by the propeller as described above. Therefore, the influence of rotating domain sizing and resulting interface placement were assessed, as disc thickness changes the flow characteristic around the propeller and alters the interface placement.

The disc thickness study described in this paper consisted of 4 thickness values, ranging from 2 blade chords (2 BC) up to 4 blade chords (4 BC). The disc thickness is increased symmetrically i.e. the same height is added above and below the blade, using the rotating domain of 2 BC height as a starting point. The results obtained with changing disc thickness showed significant variability, confirming the importance of this parameter for obtaining reliable results.

## 3. Results

The described numerical model was used to simulate the propeller at three different rotational speeds – 0.84 of nominal, nominal, and 1.16 of nominal speed. For each RPM, four rotating domain sizes were tested. The obtained results provided numerical parameters, such as thrust and power, but also additional information which were used to assess the simulation quality.

An assessment of resultant y+ value on the blade was the first step for solution validation. Fig. 6a shows the y+ parameter value in a range up to 10, indicating that the whole blade is included in this range. The y+ contour up to two (Fig. 6 b), shows in which regions the boundary layer was fully solved. Presented values were obtained for the highest 1.16 nominal rotational velocity. It is visible that the y+ parameter is below 2 for the majority of the blade area. The value is higher for the blade inboard part, due to a meshing compromise described earlier. The separate study aimed at obtaining the y+ value smaller than 2 for the entire blade confirmed that, despite increased computational resources, no further improvement in thrust nor power was achieved. The area average value of y+ parameter over the whole blade is 1.27.

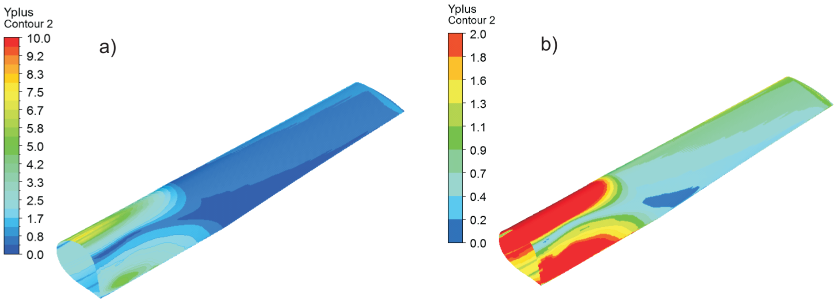

**Figure 6** Y+ parameter contour on the propeller blade: a) with range up to 10; b) with range up to 2

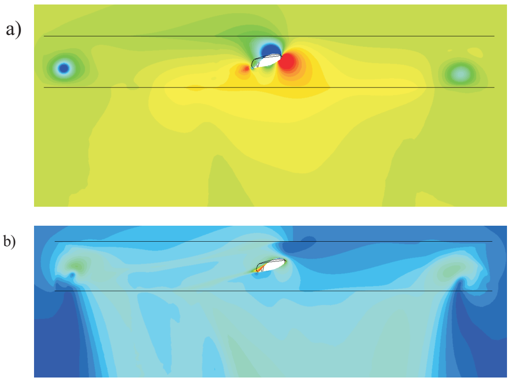

**Figure 7** a) Pressure field near the interfaces (black lines); b) velocity field near the interfaces

Velocity and pressure continuity at the interfaces were another important parameters which had to be assessed. The contours showing pressure and velocity transfers through interfaces are shown in Fig. 7a and 7b. The interfaces proved to be coherent and ensure physical data transfer. Even though the same level of accuracy was observed for smaller disc thickness (2 BC), the results differed significantly. This indicated that not only the integrity of fluxes across interfaces is to be assessed when using the MRF model.

The converged simulations were next used to calculate thrust and power values for four rotating domain sizes at three rotational speeds. The results showed high dependence of thrust and power on the rotating domain sizing. The experimental data was used for a comparison with obtained numerical results and recommendation of an appropriate disc thickness. The results for different rotating domain sizing are shown in terms of thrust and power differences from the experimental values.

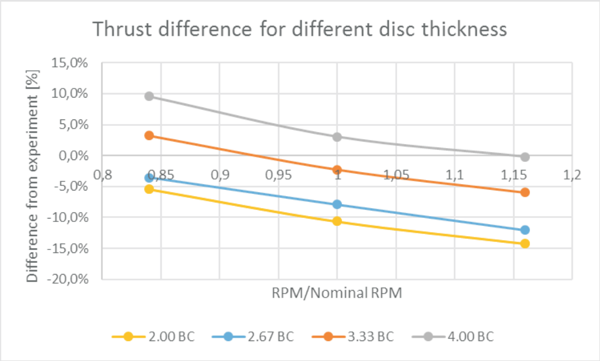

**Figure 8** Thrust difference from experiment for different rotating domain sizing

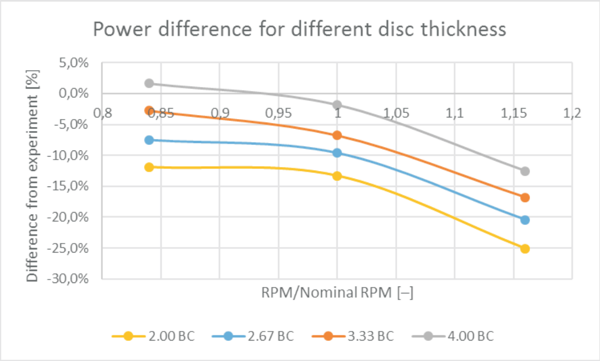

**Figure 9** Power difference from experiment for different rotating domain sizing

The graph in Fig. 8 presents the difference between numerical and experimental values of thrust for three rotational velocities. Fig. 9 shows a similar dependence for differences between experimental and numerically obtained values in terms of power. The figures indicate that an increase of disc thickness results in an increase of power and thrust generated by the blade. Considering the thrust, a very close match for experimental values can be found in the range of the applied domain sizing. The corresponding values of power are lower than the experimental ones, with the difference increasing along the rotational speed increase.

Due to the thrust and power variation for different rotating domain sizing, an analysis of thrust and power distribution along the blade for different disc thickness was conducted. In Fig. 10a–10c thrust distributions along the blade for different disc thickness are presented. A difference in the thrust value is visible along the whole blade span, with the biggest variation for the outboard blade region especially between the cases with lower domain span (2 and 2.67 BC) and the two with larger rotational domain sizing (3.33 and 4 BC). The highest difference in thrust is observed for 1.16 of nominal RPM, where the air velocity and, therefore, forces generated on the blade surfaces are the highest. In general, the higher the disc thickness, the bigger the thrust generated by a blade portion is. This confirms the assumption that this mesh parameter has an impact on pressure distribution forming around the blade sides. In the absence of the intensified flow mixing, normally present when numerous blade aerodynamic trails are encountered, the rate of dissipation of energy carried by vortex sheets is hindered. The location of rotating–stationary interface seems to have an upstream influence on the base pressures and thus has to be moved further away to allow sufficient room for loss mechanisms to unfold.

Fig. 11 from a) to c) presents power distributions along the blade in a similar convention as in Fig. 10. The power distribution analysis also indicates that the amount of power produced by the blade is bigger for each blade section with increasing disc thickness. Interestingly enough, in case of power, however, higher differences are observed in the inboard blade part. Higher differences are also visible for the region close to the blade tip where especially the 2 BC case seems to underperform.

## 4. Discussion

The analysis presented in this paper shows that CFD is highly sensitive to numerical model design, especially when applying simplifications, such as steady state model for a propeller. In case of MFR model, the disc thickness and, therefore, interfaces placement plays a crucial role. The mistakes done in this aspect may lead to significant differences in results, despite the overall good mesh quality and attained solution convergence level.

The curves presented in Figures 8 and 9 have a very similar character for different disc thickness. This proves the consistency of the method used in this paper. The difference in obtained values of power and thrust implies that the numerical model of propeller simulation should be calibrated for a given rotational speed and probably blade geometry. The thrust and power variation from experimental results for the same rotational speed, reached up to 15%. Such a difference may lead to important mistakes when the blade characteristics is used for a design of an aircraft.

The results show, that the smallest disc height, which, intuitively, could be assumed as the most correct, as the smallest volume of air around the propeller is rotated, produces much lower power and thrust values.

It may be noticed that for reproduction of the experimental results, the disc thickness should grow along with the rotational speed of the propeller. This may result from the fact that the pressure distribution on a blade depends on the vortex sheet created by the propeller. If the disc is too thin and an interface is placed too close to the propeller, the vortex may not be correctly simulated, which influences the pressure distribution.

Figures 8 and 9 show that differences from experimental values either in terms of thrust or power are present for the given disc thickness. According to the results, the choice of disc thickness for which the value of power and thrust will be identical with the experiment is not possible. While the error made in case of 0.84 of nominal RPM and nominal RPM is not high, it becomes more significant for the highest rotational speed.

It should be, however, noted that the experimental results were obtained with a test bench, which was not included in the simulation. The influence of this structure could lead to disturbances, which would, in turn, result in higher power consumption for the same value of thrust. Due to this fact, the simulation results may show that for a propeller working in a flow field not altered by the test bench elements, the same thrust as in experiment would be obtained for lower power, or for the same power, higher thrust value could be attained.

The thrust distribution along the blade shows that the differences are visible along the entire blade. There are, however, regions, where the difference is more noticeable: mainly on the outboard portion of the blade and near the tip.

An analysis of power distribution indicates that very high amount of power is generated near the tip of the blade, which results from significant energy consumption by the forming tip vortex. The complementary effect of thrust decrease due to tip leakage is also visible on thrust distributions. In case of power, higher difference for changing domain sizing is visible in an inboard part of the propeller, while the outboard part is almost identical for the three highest domain sizes. The difference between the three highest velocities is also visible at the tip of the propeller, where for higher disc thickness, higher power is generated.

The study considered only equidistant rotating domain size changes. The importance of domain growth on the inflow side or outflow side could also be established for further analysis. This could give an answer whether for the MRF approach, the distance of interface from the blade is more important for the inflow or outflow side.

## 5. Conclusions

This article presents the development of numerical model for a propeller simulation, gives detailed description of the performed simulation and presents its validation using the experimental data. The created CFD model using multiple reference frame formulation allows one to precisely simulate the behaviour of the propeller using steady state approach. It is more efficient than transient simulation, which is more time consuming and requires far greater computational resources.

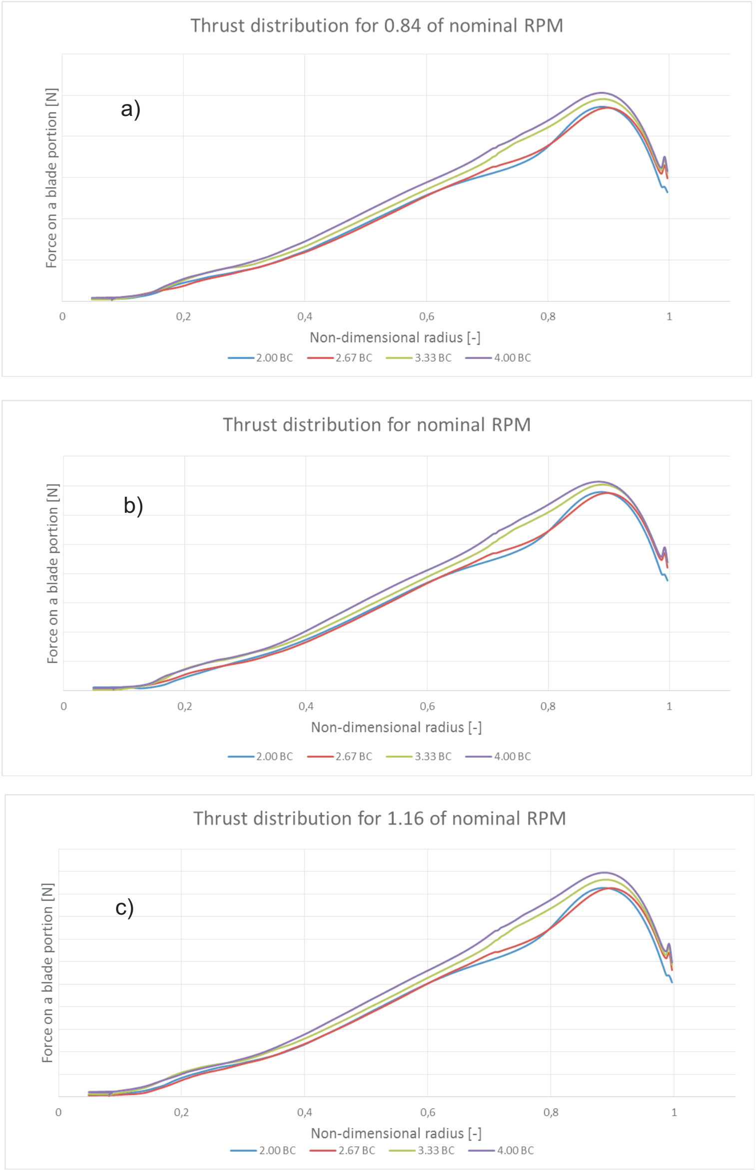

**Figure 10** Thrust distribution for different rotating domain sizing: a) 0.84 of nominal RPM; b) nominal RPM; c) 1.16 of nominal RPM

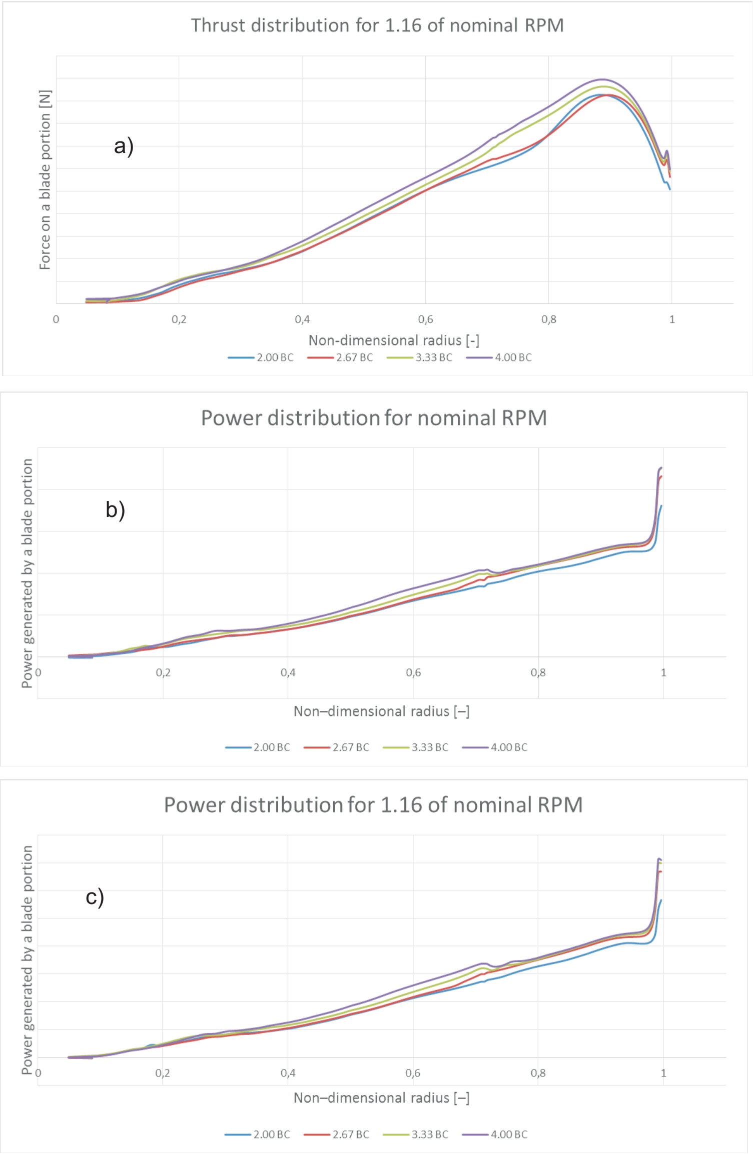

**Figure 11** Power distribution for different rotating domain sizing: a) 0.84 of nominal RPM; b) nominal RPM; c) 1.16 of nominal RPM

The study shows that the model is sensitive to the rotating domain sizing, therefore it should be properly calibrated for the given blade and rotational velocity. The sensitivity for two–bladed propeller may come from important propeller wake, concentrated on a relatively small interface area, compared to the overall interface extent.

The difference in results for the given disc thickness changes with the rotational velocity. When attempting to reproduce the experimental results, for higher rotational velocities, higher disc thickness is required. The need for the size increase is a result of propeller wake growth with the rotational velocity. A part of the differences may also come from altering the inflow conditions, which happen when disc thickness is changed. The study of a non-symmetric disc changes could give an answer to the observed behaviour.

The comparison of numerical and experimental results shows that different disc thickness should be used for different rotational velocities. There is no general formula presenting the optimal thickness, therefore further research in this area is required to provide dependence from rotational velocity, blade geometry or other parameters. The choice of optimal disc thickness should be possible for a certain case, producing power and thrust values matching with the experiment. The divergence of results presented in this study is attributed to an influence of test bench, which was not included in the numerical simulation.

Changes in rotating domain thickness produce systematic changes in power and thrust values, while not changing significantly the distribution of those quantities along the blade. In the inboard part, the power and thrust are simultaneously increased, while in the outboard part, the power remains almost constant while the thrust increases significantly. This may lead to a conclusion that the changes in the velocity triangle are not linear and more profound studies are required to fully understand the influence of rotating domain sizing on flow structure in steady state MRF simulations.

## References

[1] Clean Sky, Overview of the Proposed Programme, **2013**. Online: www.cleansky.eu/sites/default/files/documents/20131009_cs2programme overviewfinal.pdf.

[2] [http://newatlas.com/project-zero-electric-tilt-rotor-aircraft/26561/#gallery]

[3] e–volo GmbH, Online: http://www.e-volo.com/.

[4] **Betz, A., with appendix by Prandtl, L.** Screw Propellers with Minimum Energy Loss, *Gottingen Reports*, 193–213, **1919**.

[5] **Glauert, H.: Airplane Propellers, :Aerodynamic Theory, edited by W. Durand, Div. L**, Vol.5, Peter Smith, Gloucester, MA, 169–269, **1976**.

[6] **Gessow A.**: Effect of rotor blade twist and planform tapper on helicopter performance, *NACA Technical note*, no. 1542, **1948**.

[7] **Hartman, E. P. and Birdman, D.**: The aerodynamics of full scale propellers having 2, 3 and 4, blades of Clark Y and R. A. F. 6 airfoil sections, *NACA Report* no. 640, **1938**.

[8] **Harrington, R. D.**: Full–Scale Tunnel Investigation of the Static Thrust Performance of a Coaxial Helicopter Rotor, *NACA Technical Note*, 2318, **1951**.

[9] **Leishman, G. J.**: Principles of helicopter aerodynamics, *Cambridge University Press*, **2006**.

[10] **Juhasz, O., et al.**: Comparison of Three Coaxial Aerodynamic Prediction Methods Including Validation with Model Test Data, *J AM HELICOPTER SOC*, 59, 3, 1–14, **2014**.

[11] **Syal, M. and Leishman, J. G.**: Aerodynamic optimization study of a coaxial rotor in hovering flight, *J AM HELICOPTER SOC*, 57, 4, 1–15, **2012**.

[12] ANSYS, C. F. X. Solver modelling guide, Release 17, **2016**.

[13] **Karczewski, M., and Eglin P.**: Application of the modified blade element method for propeller blade design on high–speed helicopter X3, *CMP-Turbomachinery, LUT*, 145, 63–64, **2014**.