

NUMERICAL SIMULATION OF THE UNSTEADY AERODYNAMICS IN AN AXIAL COUNTER-ROTATING FAN STAGE

Younsi M.* and Hutchinson B.

*Author for correspondence
ANSYS, Inc.

15 place Georges Pompidou
78180 Montigny le Bretonneux
France

E-mail : mohand.younsi@ansys.com

Ravelet F., Khelladi S. and Bakir F.

DynFluid Laboratory
151 boulevard de l'Hôpital
75013 Paris
France

ABSTRACT

The purpose of this work is to investigate and to understand the unsteady flow behavior in axial counter-rotating fan stage.

3D numerical simulations combined with experimental investigations were carried out, taking into account the entire environment of the stage. The numerical results were compared to the measurement data in terms of overall performance and local unsteady variables.

In the numerical procedure, the complete geometric details of the fan stage were taken into account, such as tip clearances, motor casings, etc. enabling an accurate presentation of the complex unsteady flow.

INTRODUCTION

The counter-rotating rotor configuration provides a way to design high performance and compact turbomachines for various industrial applications. It has been already applied in the areas of subsonic fans, pumps and turbines [1 - 4]. Compared with a traditional rotor-stator stage, the rear rotor in a counter-rotating stage is used not only to recover the static head but also to supply energy to the fluid. Therefore, to achieve the same performance, the use of a counter-rotating stage may lead to a reduction of the rotational speed and may generate more homogeneous flow downstream of the stage.

On the other hand, the mixing area between the two rotors induces complicated interacting flow structures [4]. The understanding of this highly unsteady flow in the mixing area is an open problem.

Moreover, the design method of such machines is still not sophisticated, due to a lack of systematic studies on the influence of free parameters, such as the distribution of loading, the axial distance, and the ratio of the rotation rates. The DynFluid laboratory has developed a design methodology for counter-rotating axial fans [4] and currently works on the influence of different design parameters [5].

Computational Fluid Dynamics (CFD) for 3D viscous flow fields provides an efficient tool for analysis, design and optimization. There is a continuously increasing demand in all areas of CFD for steady or unsteady flow simulations. In the

turbomachinery area, the unsteady behavior of the flow field can have a significant influence on the aerodynamic performances and the acoustic behavior.

As a complementary or alternative approach to experimental methods, CFD analysis provides a viable way to understand the interactions and the flow unsteadiness in rotating machinery.

Previous works with CFD tools were realized in order to study counter-rotating configurations for fans, propellers, pumps and turbines, etc.

Numerical approaches were employed [6, 7] to study the flow field in a counter-rotating open rotors used in propulsion systems. The complexity of the unsteady flow induced by the interactions between the rotating blades was highlighted.

As the far field acoustic behavior depends on the flow unsteadiness, the CFD results showed that the most critical region for acoustics purposes is at the two rotors and the wake regions. Flow between the front and rear rotors is of particular interest as there is a subsequent phenomenon of fluid being continually "twisted" and "untwisted" by the counter-rotating front and aft rotors respectively, causing a potent area for waveform fluctuations causing disturbances in the audible range. Other investigations were dedicated to ducted configurations. The impact of the blade pitch angles and the axial spacing between the counter-rotating propellers was investigated numerically [8].

NOMENCLATURE

<i>BEP</i>	[-]	Best Efficiency Point
<i>BPF</i>	[Hz]	Blade Passing Frequency
<i>FR</i>	[-]	Front Rotor
<i>nf_{FR}</i>	[Hz]	Harmonics of Front Rotor
<i>mf_{RR}</i>	[Hz]	Harmonics of Rear Rotor
<i>RR</i>	[-]	Rear Rotor
<i>R_{hub}</i>	[mm]	Hub blade radius
<i>R_{tip}</i>	[mm]	Tip blade radius
<i>S</i>	[mm]	Axial distance between FR and RR
<i>Z_p</i>	[mm]	Axial position

NUMERICAL SIMULATION PROCEDURE

In this paper, 3D numerical simulations were carried out using the commercial CFD code ANSYS CFX release 16.2. The flow solver is a coupled pressure-based solver. It employs a fully implicit solution strategy. The focus of this work is transient CFD simulations. However, steady calculations were also performed to predict the overall performance and to initialize the unsteady calculations.

The steady method uses a mixing plane type of approach called a “Stage Interface” where momentum and energy are conserved across the interface, but entropy increases as a result of the circumferential averaging process. This steady method requires simulation of only one blade per row.

The unsteady calculations use the Transient Rotor-Stator (TRS) approach where the true change in relative position of the front rotor and rear rotor are accounted for within fully implicit interface discretization. In this approach, either full wheel is assumed or the smallest possible circumferential sector with an integer number of blades, according to the pitch ratio, is modelled with standard periodicity applied to the blade ensemble (unity ensemble pitch ratio between blade rows).

In this work, the blade counts were such that it was necessary to model the full wheel to achieve unity pitch ratio.

Geometry and Mesh

Figure 1. shows the axial counter-rotating stage geometry and its main characteristics are summarized in Table 1.

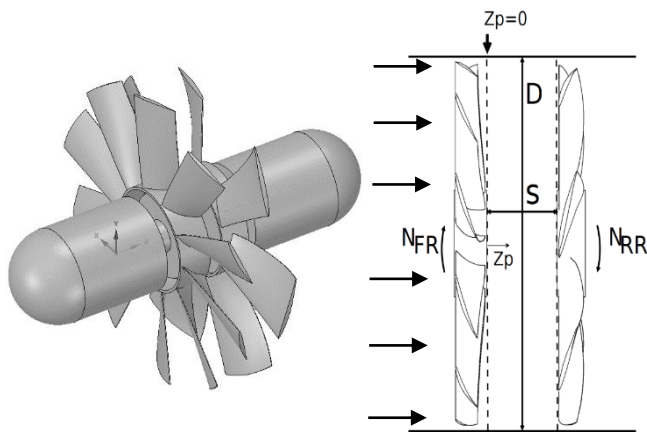


Figure 1 Geometry of the counter-rotating fan

	FR	RR
R_{tip} [mm]	187.5	
R_{hub} [mm]	65	
Tip Clearance [mm]	2.5	
Blades Count	10	7
Rotational Velocity [rpm]	-2300	2200
Blade Passing Period [s]	0.0026	0.0039
BPF [Hz]	383.33	256.66

Table 1 Main characteristics of the axial counter-rotating fan

The entire geometry features, including tip clearance, were taken into account in the meshing procedure. The bladed fluid domains were meshed using the ANSYS TurboGrid software. Elsewhere, ANSYS Meshing software was used to mesh the upstream domain, the downstream domain and the motor casings. One and one half million cells were used for each of the FR and RR domain sectors, thus the full computational domain consists of three million cells.

Particular attention was devoted to the mesh distribution in the boundary layers and the tip clearance. Ten cells were imposed in the radial gap between the blade tip and the shroud. The near-wall mesh was adapted to the boundary layer flow. The dimensionless wall distance y^+ stands in the range [3 - 8]. Figure 2 illustrates the resulting hexahedral mesh and Figure 3 shows the mesh details for the FR blade domain.

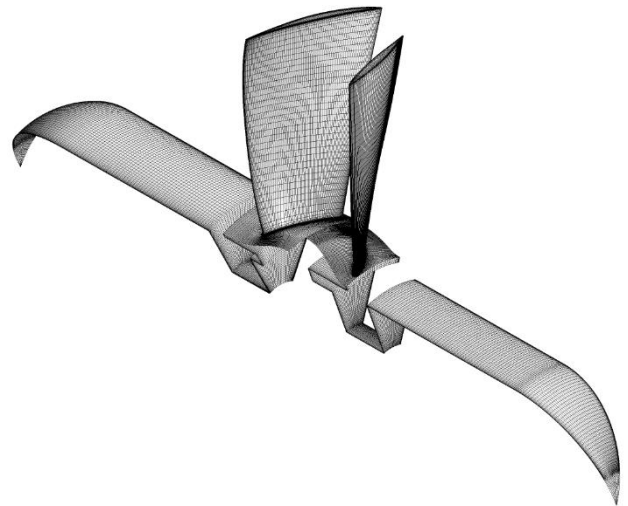


Figure 2 Overview of the periodic domain mesh

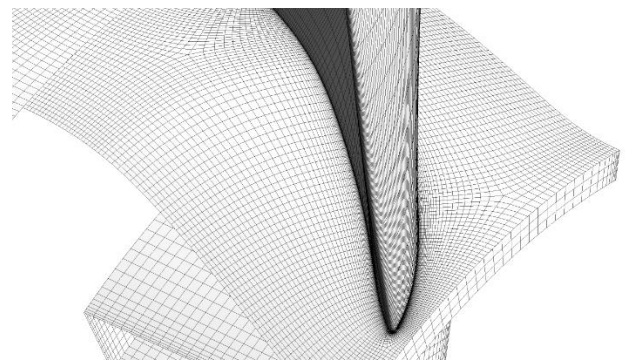


Figure 3 Mesh details for the FR blade domain

Simulation Parameters and Setup

The numerical simulations (steady and unsteady) were performed at the design point of the fan which corresponds to a pressure rise of 363 Pa and a mass flow rate of 1.2 kg/s.

A first steady simulation was performed with total pressure and mass flow boundary conditions at the inlet domain and the outlet domain respectively. Only one blade pitch for each rotor was simulated with rotational periodic boundary conditions. The

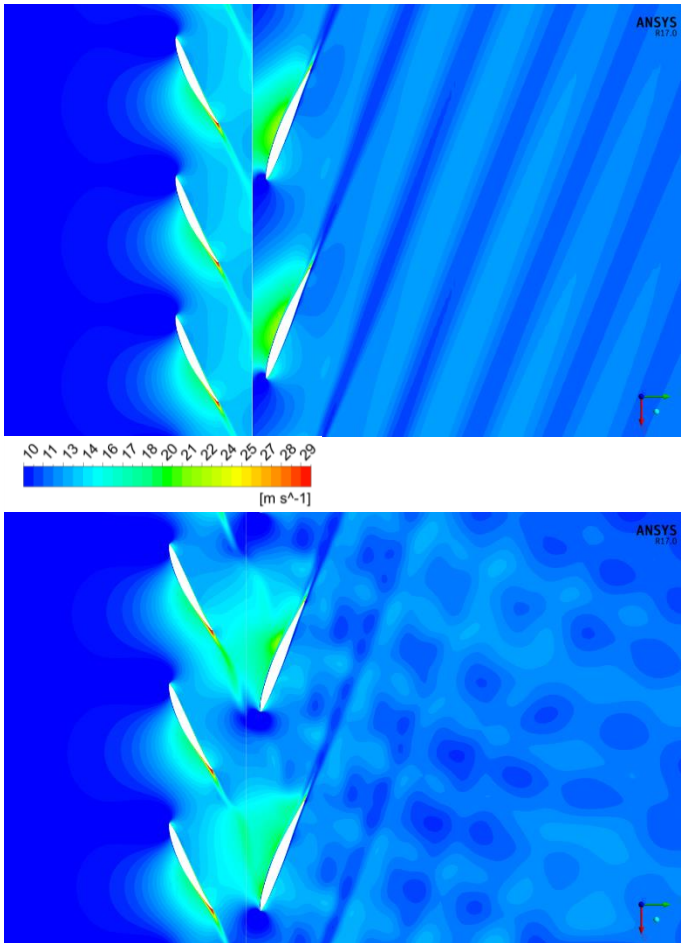


Figure 7 Blade-to-blade absolute velocity at mid-span (top: steady, bottom: unsteady)

Overall Performance

Table 2 gives a comparison between the overall performance obtained numerically (steady state) and experimentally at the BEP of the counter-rotating fan. It can be seen that the global performance is well predicted by the steady numerical approach. The unsteady calculations give similar results.

	Experiment	ANSYS CFX
Pressure Rise [Pa]	363 ± 4	365
Static efficiency[%]	66.1 ± 1.4	64
FR power [W]	335 ± 2.5	327

Table 2 Overall performance at the BEP

Radial Distribution of Velocity Components

The time-averaged velocity profiles given by the unsteady calculations were compared to LDV measurements. Figure 8 shows the radial distribution of the absolute circumferential velocity at $Z_p = -46$, $Z_p = 5$ and $Z_p = 50$ planes. The curves at the upstream of the fan stage ($Z_p = -46$) demonstrate clearly that the flow comes without pre-rotation. The mean value is around 0 m/s. However, a particular behavior can be observed at the tip blade zone which is due to the tip leakage phenomena.

Good agreement between the numerical and the experimental data can be observed.

According to Figure 9 and due to the radial equilibrium condition, the axial velocity profiles are globally conserved from the upstream to the downstream domain.

The analysis of the absolute circumferential velocity profiles at the three locations shows that the rear rotor plays an important role to reorient the flow axially at the exit domain. Thus, at $Z_p = 50$ plane, the mean circumferential velocity magnitude is very small. This result confirms the preliminary design objective of achieving low swirl at the exit of the counter-rotating fan stage. Some LDV velocity measurements reveal that some strong modifications of the flow close to the hub and to the blade tip [5] seem to take place, in regions where it is difficult to have access experimentally. That could explain the discrepancies between the numerical model and the measurements at these zones.

Wall Pressure Fluctuations

The wall pressure fluctuations and its corresponding spectrum are illustrated in Figure 10. The signal corresponds to 5 FR revolutions time-signature. Experiment and CFD results show excellent agreement, especially the full 3D unsteady simulations that succeed in reproducing most of the spectral peaks, including the blade passing frequency of the FR, of the RR, and their harmonics and interactions. The pressure spectrum clearly distinguishes the linear combination of different modes which are the result of the unsteady flow interactions between the two rotors (peaks at f_{FR} , f_{RR} , $n.f_{FR}$, $m.f_{RR}$, $n.f_{FR}+m.f_{RR}$,...).

Velocity Fluctuations

Figure 11 presents the velocity fluctuations at $Z_p = 5$ plane. The point of interest is located at 120 mm radially from the rotational axis.

The velocity fluctuation amplitude stands in the range of [-3.5, 3.5] m/s. Some discrepancies can be noticed between the numerical and the experimental results. The measurement procedure used here shows some limitations to accurately capture the velocity fluctuations. More efforts have to be considered in the future to analyse and improve the measurement accuracy.

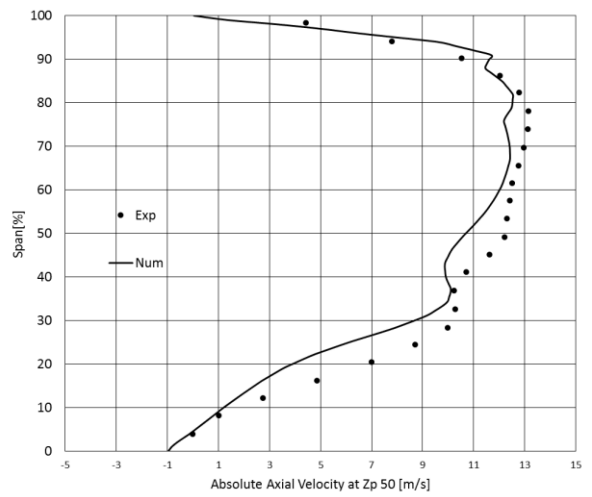
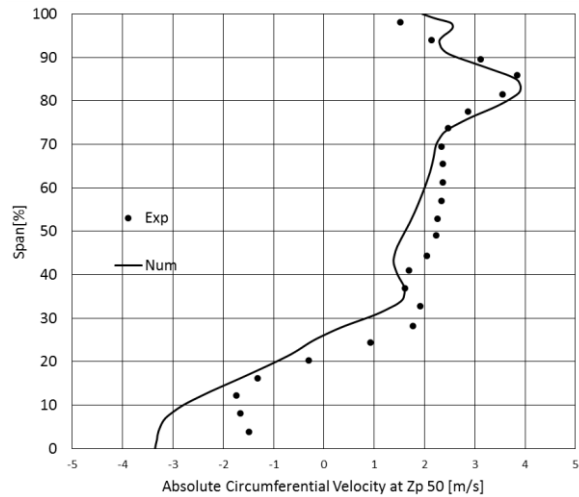
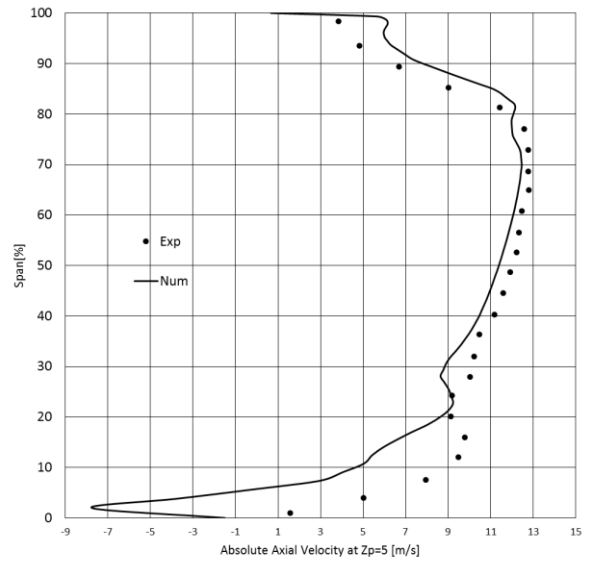
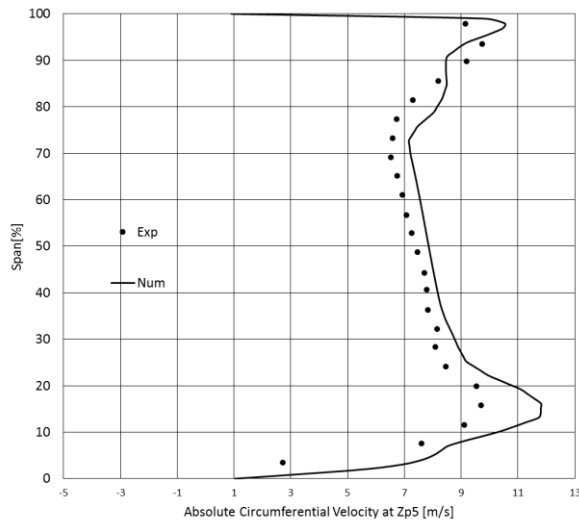
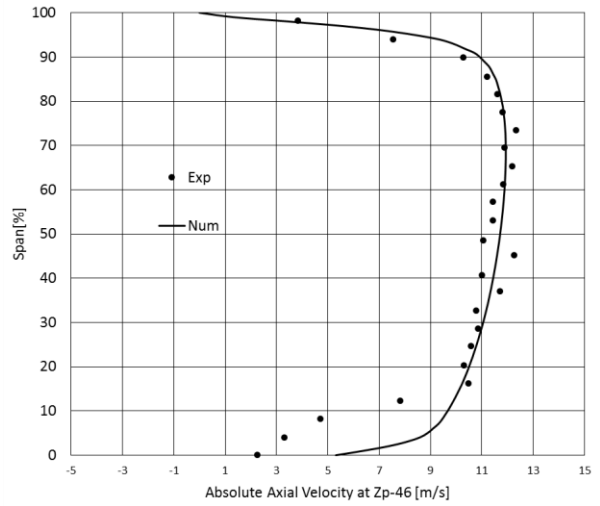
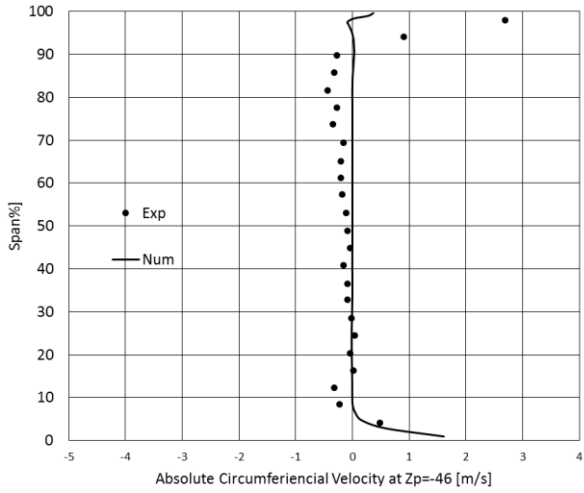


Figure 8 Radial distribution of absolute circumferential velocity

Figure 9 Radial distribution of axial velocity

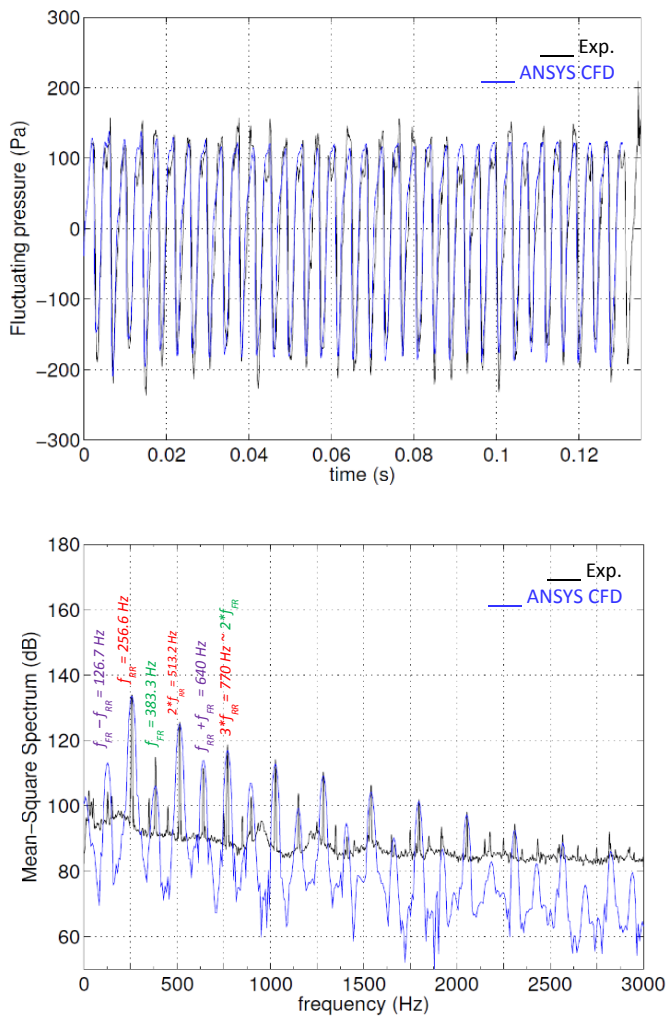


Figure 10 Wall pressure fluctuations and its spectrum

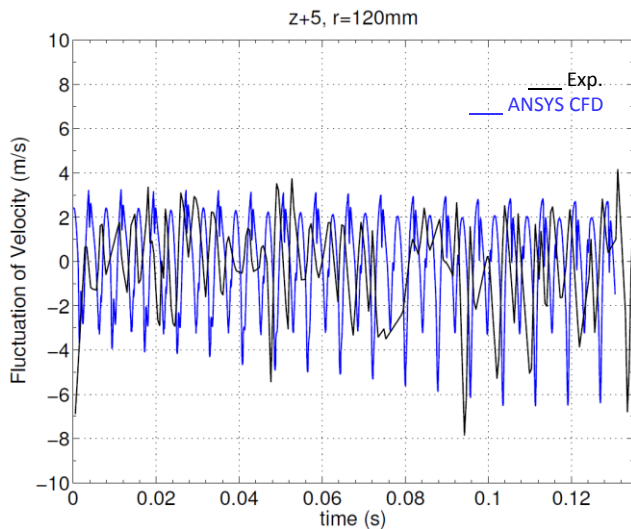


Figure 11 Velocity fluctuations at $Z_p = 5$

CONCLUSION

Unsteady flow through an axial counter-rotating fan stage was studied and investigated using ANSYS CFX in this work. The numerical procedure was compared locally and globally to experimental data. The complete geometry details of the fan stage were taken into account, such as tip clearances, motor casings, etc. contributing to an accurate representation of the complex unsteady flow. Good agreement between numerical and experimental results was observed.

It should be interesting to extend this study in the future by:

- Studying and optimizing the impact of the axial distance between the two rotors on the internal flow and the acoustic behaviour
- Acoustic measurements at the far field in an anechoic room to avoid sound reflections.
- Further analysis in order to better understand the coupling of the two rotors, i.e. the effects of the presence of the Rear Rotor onto the Front Rotor. Some measurements of velocity with LDV have indeed revealed that some strong modifications of the flow close to the hub and to the blade tip seem to take place, in regions where it is difficult to access experimentally. The use of Scale-Resolving Simulations (SRS) models available in ANSYS CFX should give more information.

REFERENCES

- [1] Shigemitsu, T., Furukawa, A., Watanabe, S., Okuma, K., and Fukutomi, J., Internal flow measurement with LDV at design point of contra-rotating axial flow pump, *Journal of Fluid Science and Technology*, Vol. 4, 2009, p. 723-734
- [2] Joly, M., Verstraete, T., and Paniagua, G., Full design of a highly loaded and compact contra-rotating fan using multidisciplinary evolutionary optimization, Proceedings of the ASME Turbo Expo 2013: Turbine Technical Conference and Exposition, San Antonio, Texas, USA, 2013
- [3] Cho, L.S., Cha, B.J., and Cho, J.S., Experimental study on the three-dimensional unsteady flow characteristics of the counter-rotating axial flow fan, *Journal of Fluid Science and Technology*, Vol. 4, 2009, pp. 200-209
- [4] Nouri, H., Danlos, A., Ravelet, F., Bakir, F., and Sarraf, C., Experimental study of the instationary flow between two ducted counter-rotating rotors, *Journal of Engineering for Gas Turbines and Power*, Vol. 135, 2013, pp. 022601
- [5] Wang, J., Ravelet, F., and Bakir, F., Influence of design parameters on the global performances of low-speed counter-rotating axial-flow fans. *American Society of Mechanical Engineers, Fluids Engineering Division*, 2014, paper FEDSM2014-22717.
- [6] Roosenboom E.W.M., Schröder, A., Geisler, R., Pallek, D. and Agoes, J. Experimental and numerical investigation of a counter rotating open rotor flow field, *29th AIAA Applied Aerodynamics Conference*, 2011, Honolulu, Hawaii
- [7] Farooque, M. S., CFD grid generation and flow analysis of a counter-rotating open rotor propulsion system, Thesis, Ohio State University, 2010.
- [8] Xu, C., Bil, C., and Cheung, S.C.P, Fluid dynamics analysis of a counter-rotating ducted propeller, *29th Congress of the International Council of the Aeronautical Sciences*, 2014, Petersburg, Russia.
- [9] Menter, F.R., Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA-Journal.*, Vol. 32, 1994, pp. 1598-1605