# Numerical simulation of the unsteady aerodynamics in an axial Counter-Rotating fans stage

Long Abstract

Florent Ravelet, DynFluid laboratory, Arts et Metiers - ParisTech, Paris, France
Mohand Younsi, ANSYS France SAS, Montigny-le-Bretonneux, France
Farid Bakir, DynFluid laboratory, Arts et Metiers - ParisTech, Paris, France
Sofiane Khelladi, DynFluid laboratory, Arts et Metiers - ParisTech, Paris, France

## Introduction

Adoption of a Counter-Rotating machine has opened a way to design high performance and compact turbomachines in various industrial domains. It has been already applied in the areas of subsonic fans, pumps and turbines [1, 2, 3, 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 better homogeneous flow downstream of the stage.

On the other hand, the mixing area in between the two rotors induces complicated interacting flow structures. 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 load, the axial distance, the ratio of the rotation rates and so on...

In the Dynfluid Laboratory, series of experiments focused on ducted axial Counter-Rotating fans have been performed [5]. Based on this research, three differente stages (JW1, JW2 and JW3) have been designed to attain the same design point, while varying the distribution of load, and their global performances have been widely studied [6].

The present work consists of a numerical study of the unsteady flow in the Counter-Rotating system JW1, performed with ANSYS 16.2.

## 1. Test case and numerical procedure

The Counter-Rotating stage that is under consideration is the “JW1”case that is described in Ref. [6]. The main features are in Tab. 1 and a sketch of the stage is displayed in Fig. 1.

| $D$ (mm) | $R_{tip}$ (mm) | $R_{hub}$ (mm) | $\Delta P_{tC}$ (Pa) | $Q_{vC}$ ($m^3.s^{-1}$) | $N_{FR}/N_{RR}$ (rpm) | $Z_{FR}/Z_{RR}$ |
|---|---|---|---|---|---|---|
| 380 | 187.5 | 65 | 420 | 1 | 2300/2200 | 10/7 |

**Table 1.** Design point for air at $\rho_a = 1.21$ kg.m$^{-3}$. $FR$: Front Rotor, $RR$: Rear Rotor. $D$: pipe diameter, $R$: radius of the fan, $\Delta P_{tC}$: design total pressure rise, $Q_{vC}$: design volumetric flow-rate, $N$: rotational velocity, $Z$: number of blades.

The overall performances have been measured in an ISO-5801 test rig, together with radial profiles of axial and tangential velocities, and casing wall pressure fluctuations. More informations about the accuracy of the experimental measurements can be found in Ref. [5].

The bladed domains have been meshed with ANSYS TurboGrid. The inlet and outlet domains that include the casings of the motors (see Fig. 1) have been meshed with ANSYS Meshing. The resulting number of cells is $1.5 \times 10^6$ for one blade pitch for the FR and RR. The total number of cells for the

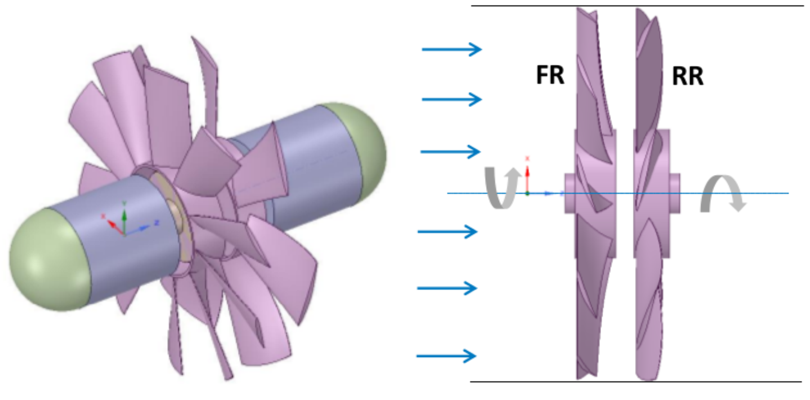


**Figure 1.** Sketch of the Counter-Rotating stage.

whole domain is $3.1 \times 10^6$. Please also note that the blade tip clearance is also taken into account with 10 cells in the radial gap. The wall $y+$ is in the range $3 \leqslant y+ \leqslant 8$.

A first steady simulation is performed with ANSYS CFX 16.2, with a mixing plane approach, a total pressure condition at the inlet and a mass flow-rate condition on the outlet. Only one blade pitch for each rotor is simulated. The Shear-Stress Transport $k-\omega$ RANS model [7] is used for the turbulence modelisation. The mixing plane options are "Constant Total Pressure" and "Implicit Stage Averaging".

Then, a full 3D unsteady RANS simulation is performed. The total number of cells in that case is $26.7 \times 10^6$. The boundary conditions are a total inlet pressure, and a static outlet pressure, with radial equilibrium. The time step is chosen as $\delta t = 3.26 \times 10^{-5}$ s, that corresponds to 80 time steps per Front Rotor blade passing period and 120 for the Rear Rotor blade passing period.

## 2. Results

| | Experimental | ANSYS CFX 16.2 mixing plane |
|---|---|---|
| $\Delta p_s$ | $363 \pm 4$ | 365 |
| Static Efficiency (%) | $66.1 \pm 1.4$ | 64 |
| FR power (W) | $335 \pm 2.5$ | 327 |
| RR power (W) | $211 \pm 2$ | 240 |

**Table 2.** Comparison between the experimental and the CFD results at the design point.

The results of the mixing-plane CFD approach are listed in Tab. 2. The global performances of the Counter-Rotating stage are well predicted, though the power consumption of the Front Rotor (resp. the Rear Rotor) is slightly underestimated (resp. 10% overestimated).

The unsteady simulation give similar results. The time-averaged velocity profiles are plotted in Fig. 2. Spectra of the wall-pressure fluctuations and of one component of the velocity at one point are compared in Fig. 3. The experiment and the CFD show excellent agreement, especially the full 3D unsteady simulation, 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 [5].

Further analysis is to be made, 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 LDA have indeed 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 experimentaly. The comparison of a highly resolved CFD of the Front Rotor working alone with one of the full stage, particularly in the

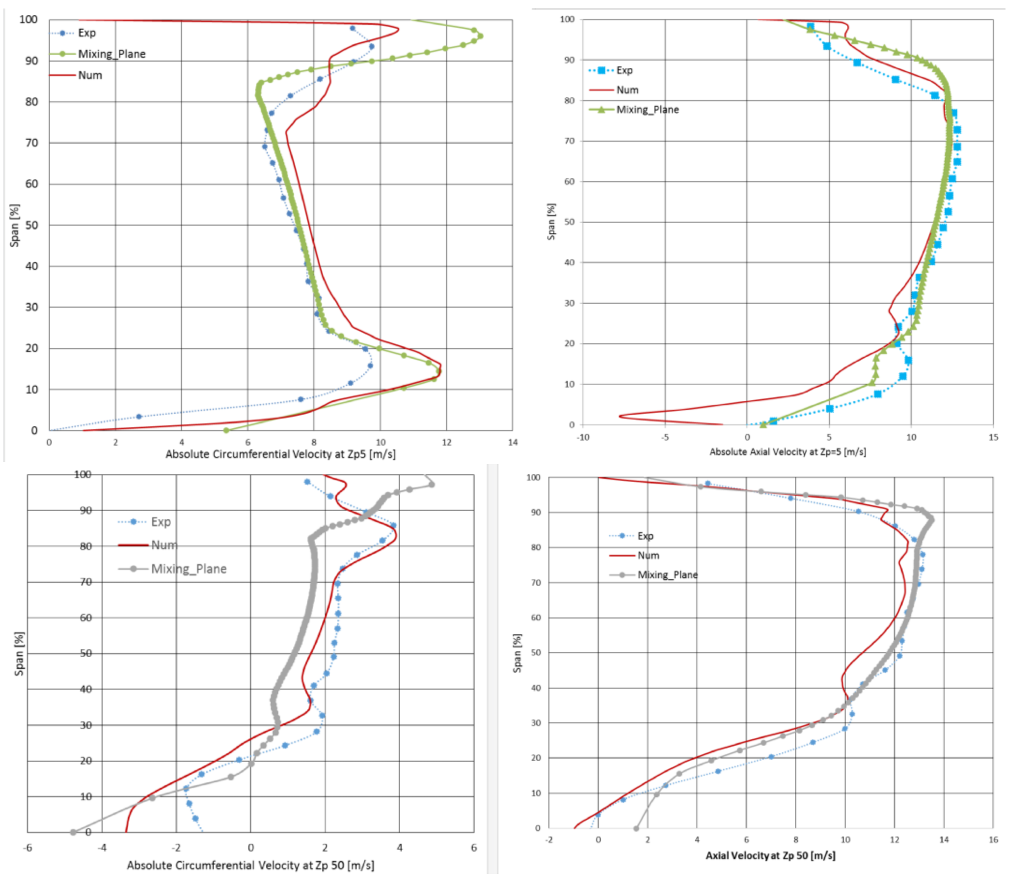


**Figure 2.** Axial and circumferential time-averaged velocity profiles between the two rotors ($Zp$ = 5 mm), and downstream of the Rear Rotor ($Zp$ = 50 mm).

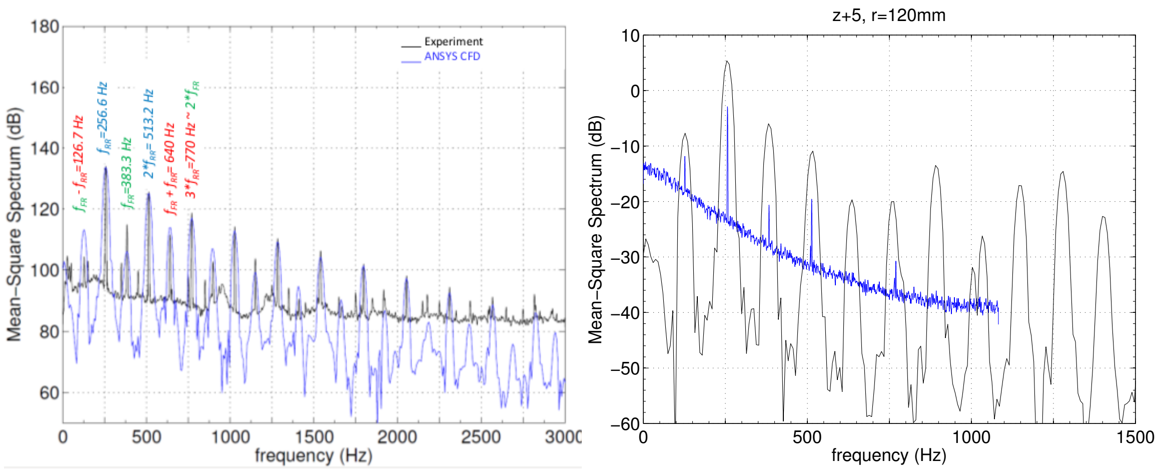


**Figure 3.** Spectra of the wall casing pressure fluctuations (left) and of the axial velocity between the two rotors, at $r = 120$ mm (right).

radial gap (see Fig. 4) will be presented at the conference.

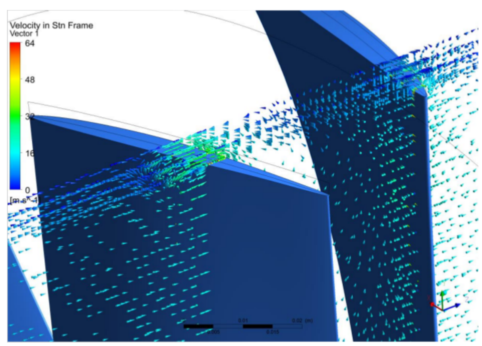


**Figure 4.** Velocity vectors close to the tip of the rotors.

## References

[1] T. Shigemitsu, J. Fukutomi, and Y. Okabe. Performance and flow condition of small-sized axial fan and adoption of contra-rotating rotors. *Journal of Thermal Science*, 19:1–6, 2010.

[2] M. Joly, T. Verstraete, and G. Paniagua. Full design of a highly loaded and compact contra-rotating fan using multidisciplinary evolutionary optimization. In *ASME Turbo Expo 2013: Turbine Technical Conference and Exposition, San Antonio, Texas, USA*, pages GT2013–94433, 2013.

[3] L. Cao, S. Watanabe, S. Momosaki, T. Imanishi, and A. Furukawa. Low speed design of rear rotor in contra-rotating axial flow pump. *International Journal of Fluid Machinery and Systems*, 6:105–112, 2013.

[4] L. Moroz, P. Pagur, Y. Govorushchenko, and K. Grebennik. Comparison of counter-rotating and traditional axial aircraft low-pressure turbines integral and detailed performances. In *Int. Symp. on Heat Transfer in Gas Turbine Systems, Antalya, Turkey*, 2009.

[5] H. Nouri, A. Danlos, F. Ravelet, F. Bakir, and C. Sarraf. Experimental study of the instationary flow between two ducted counter-rotating rotors. *J. Eng. Gas Turbines Power*, 135:022601, 2013.

[6] J. Wang, F. Ravelet, and F. Bakir. Influence of design parameters on the global performances of low-speed counter-rotating axial-flow fans. In *ASME 2014 4th Joint US-European Fluids Engineering Division Summer Meeting, Chicago, USA*, pages FEDSM2014–22172, 2014.

[7] F. R. Menter. Zonal two equations k-$\omega$ turbulence models for aerodynamic flows. In *Proceedings of the 24th AIAA Fluid Dynamics Conference, Orlando, USA*, pages AIAA paper 93–2906, 1993.