NUMERICAL INVESTIGATION OF AN AXIAL FAN FOR AUTOMOTIVE APPLICATIONS

Nicola ALDI[1], Nicola CASARI[1], Stefano OLIANI[1], Michele PINELLI[1], Enrico MOLLICA[2], Filippo MENICHINI[2]

[1] *University of Ferrara, Engineering Department, via Saragat, 44122 Ferrara, Ferrara*

[2] *SPAL Automotive Srl, Via Carpi, 26b, 42015 Correggio RE*

SUMMARY

The requirements of high efficiency and low noise of axial flow fans are pushing manufacturers and researchers to look into the operation of these machines in the effort of minimizing losses and noise sources. In this work the authors present the numerical investigation of an axial flow fan with *OpenFOAM*. The results are compared with experimental data obtained in two test benches: one *ad hoc* developed for acoustic and flow field analyses and one built in compliance with ANSI/AMCA international standards, for airflow performance evaluation. The capability of the software of correctly replicating the fan performance and flow field even in this complex set-up (due to the loose constraints of the fluid flow) is shown both with steady and transient simulations.

INTRODUCTION

Axial flow fans are of great importance for automotive as well as for motorcycle and off-road markets, since they are key components for engine cooling and vehicle air conditioning. Axial fan makers are engaged in continually product refining according to customer requirements, working closely with OEMs and seeing new ideas in order to meet emission standards and operate efficiently and reliably. The requirements of high efficiency and low noise of axial flow fans are pushing manufacturers and researchers to look into the operation of these machines in the effort of minimizing losses and noise sources. For this reason, *ad hoc* test benches have been developed for the experimental analysis. The fluid-dynamic performance of the axial flow fans is evaluated by means of airflow tests according to AMCA 210 and ISO 5801 international standards. The airflow test setup can be performed both in the “free blowing condition” (“fan only”) or with a radiator. In the latter case the interactions between fan and radiator can be also studied and evaluated. As far as PIV measurements are concerned, that put insights into the flow behaviour, the “free blowing” test setup at 0 Pa is used, allowing for optical access. Moreover, concerning the acoustic performance

evaluation, the noise tests on the axial fan are performed in a semi-anechoic chamber, according to the test setup agreed with the customer.

Over the last decades, numerical simulations with CFD software have become an industrial standard for complementing and getting insight into the experimental evidence and proposing solutions. Specifically, the increasing reliability and accuracy of the results, coupled with the growing computational power, has led to a more extensive usage of the numerical investigation as an alternative to the conventional experimental techniques. Among the possible available CFD suites, *OpenFOAM* is gaining an increasing share of the market due to its opensource nature and great accuracy and reliability of the results. The software has been largely validated as applied to incompressible flows [1], compressible flows [2] and turbomachinery applications [3-5] and therefore it represents a good candidate for the analysis of axial fans for the automotive field.

The present work shows the capabilities of the *OpenFOAM* suite to correctly replicate the behaviour of the fan. The machine under investigation, reported in Figure 1, is an axial flow fan capable of delivering more than 500 m^3/h at 0 Pa ("free blowing") and full speed (5900 rpm).

Figure 1: Axial fan investigated in this work

EXPERIMENTAL SETUP

The fluid-dynamic performance of the investigated axial fan (Figure 1) has been evaluated at the SPAL Automotive airflow test bench, built according to AMCA 210 and ISO 5810 (Figure 2, left). The "Integrated air/water thermal airflow bench" has a closed loop chamber suitable for different setups: "fan only", radiator only and complete cooling system, including radiator coolant pressure drop. For the purpose of comparison with the numerical results of this study, the "free blowing" airflow test (axial fan alone) has been considered.

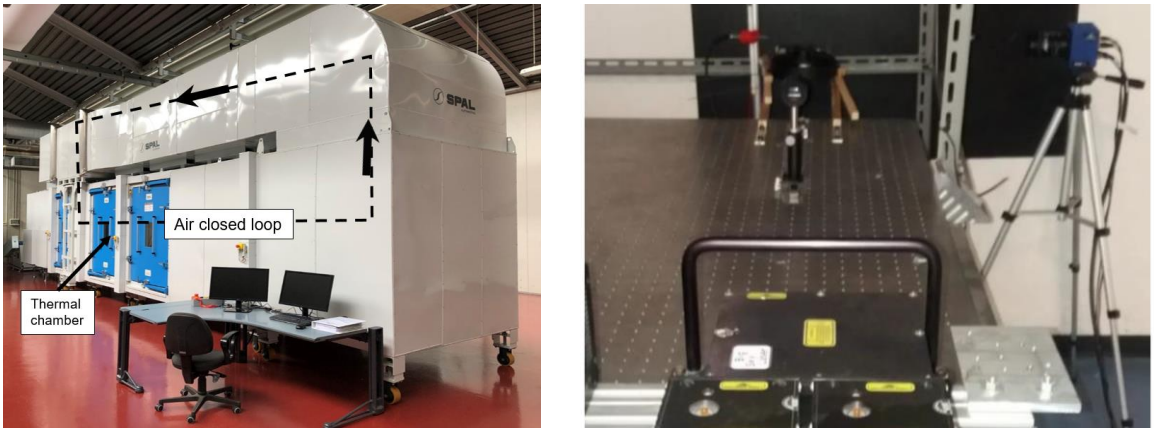

Figure 2: Left: SPAL Integrated air/water thermal airflow bench
Right: PIV measurement setup (credit to University of Rome "Roma Tre", Fluid Dynamic Laboratory "G. Guy")

The axial fan has been driven at the maximum speed 5900rpm. Moreover, for a qualitative comparison with the *OpenFOAM* velocity results, a PIV measurement with a cross-correlation camera (2,360 x 1,776 px) and a double cavity Nd:YAG laser 200 mj/pulse at 10 Hz has been performed (Figure 2, right). The instantaneous velocity fields have been acquired at a camera distance of 0.5 m from the fan axis, imaging a window with 1mm grid resolution.

NUMERICAL SETUP

OpenFOAM in its version v2006 [6] has been employed for this analysis. This software suite covers the three steps needed by a numerical simulation: preprocessing, solver and postprocessing. The main advantage of using this specific suite is related to its open source nature: (i) many researchers are actively working to improve its performance – including continuous bug fixing; (ii) no licensing costs are related to its usage; (iii) very good scalability makes massive simulations cheap, using as many cores as needed; (iv) customized, tailored and case-specific tuned models can be easily implemented.

For the study of the axial fan, the computational domain reported in Figure 3 has been obtained. Such domain is considered as composed of three parts that will be divided by non-conformal interfaces (Arbitrary Mesh Interface – AMI): inlet, fan and outlet zone. At the inlet side, the domain extends one and a half (1.5) fan external diameters upstream of the fan intake section. The stator domain is axisymmetric and the radius of the cylinder which bounds the computational domain is two and a half (2.5) external fan diameters both in the inlet and outlet side, while the outlet extends three (3) external fan diameters downstream of the fan outflow section. The meshing procedure has been carried out by means of the software *cfMesh* [7]. The software is a mesh generator provided with *OpenFOAM* and has been employed for creating cartesian trimmed meshes. A preliminary mesh sensitivity analysis has been carried out by simulating a single operating point in steady state.

The results are reported in Figure 4. It can be seen that while major differences have been found between the coarse and medium grid, slight changes in both the so-called “Fan Static Pressure” (FSP – difference between inlet total pressure and outlet static pressure) rise and in the velocity profile on the suction side of a blade can be detected. Given the high number of elements of the “fine” grid, the “medium grid” has been chosen for the current analysis.

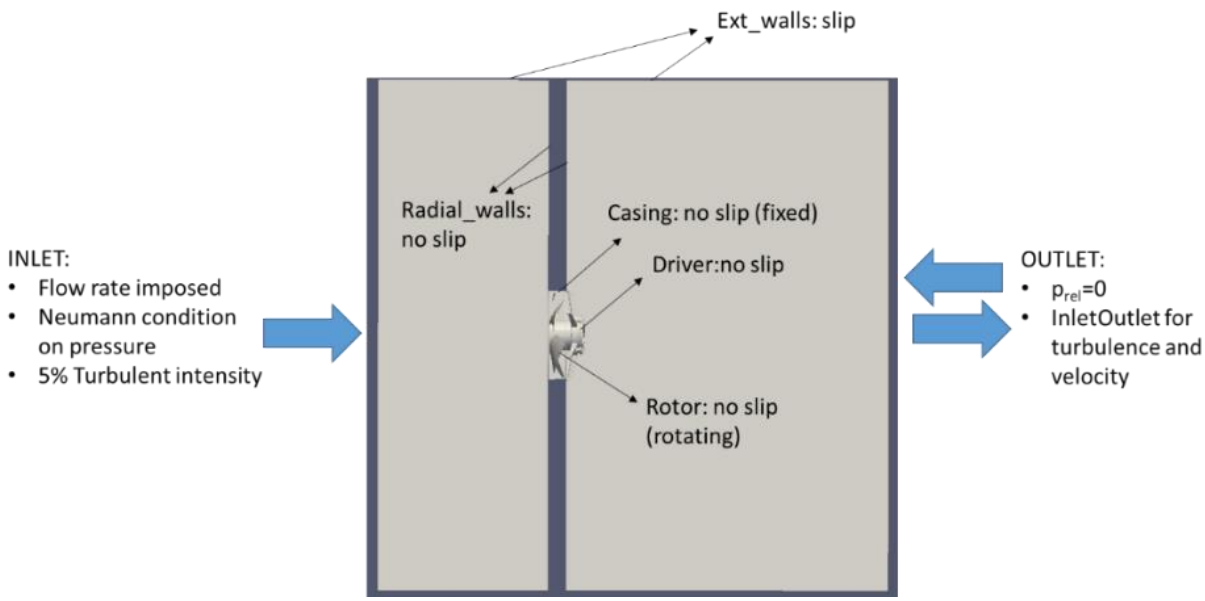

Figure 3: Computational domain considered in this work

Grid	# Elements	FSP/FSP_{FINE} [%]	NON ORTHOGONALITY (0-90)
COARSE	7,845,443	84.7	Max: 63.05, Avg: 5.47
MEDIUM	13,535,922	103.4	Max: 63.05, Avg: 4.56
FINE	17,084,013	100	Max: 63.06, Avg: 4.58

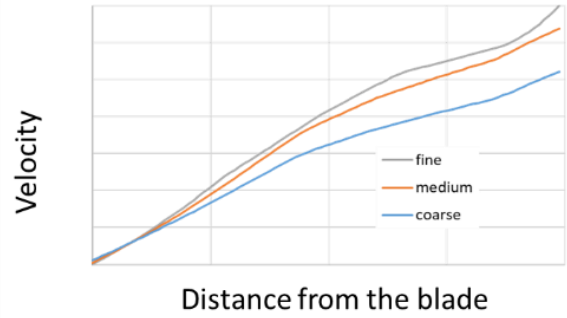

Figure 4: Grid sensitivity analysis – FSP and velocity profile on the suction side of a blade.

Figure 3 also reports the set of BCs imposed for obtaining the performance map of the fan. Specifically, the different operating points of the curve have been obtained by changing the inlet flow rate, in a steady state approach. The employed solver is *simpleFOAM*, an incompressible, turbulent and steady state solver based on the SIMPLE algorithm [8] for the pressure velocity coupling. The employed turbulence model is a k-ω SST model [9]. The simulation was carried out with a low-Reynolds approach, with an average y^+ on the rotor around 2. A frozen rotor model has been considered. This approach, which is the one consolidated in *OpenFOAM*, has been preferred due to the absence of a stator: the casing supports are very thin with respect to the disk area, and therefore the disturbance of the flow field is negligible from the performance standpoint. A second order discretization scheme has been chosen for velocity divergence, while maintaining a first order on the turbulent quantities.

RESULTS

The performance curve in terms of FSP and power absorbed by the machine are reported in Figure 5. In terms of FSP, a very good agreement has been found in the trendline. However, for all the investigated points, an underestimation of the inlet total pressure has been found. In a similar way, the torque and therefore the mechanical power has been found to be lower than the experimental data. These discrepancies, consistent with what found with other CFD software employed for assessing such a phenomenon, might be related to unsteady phenomena not well captured by steady simulations.

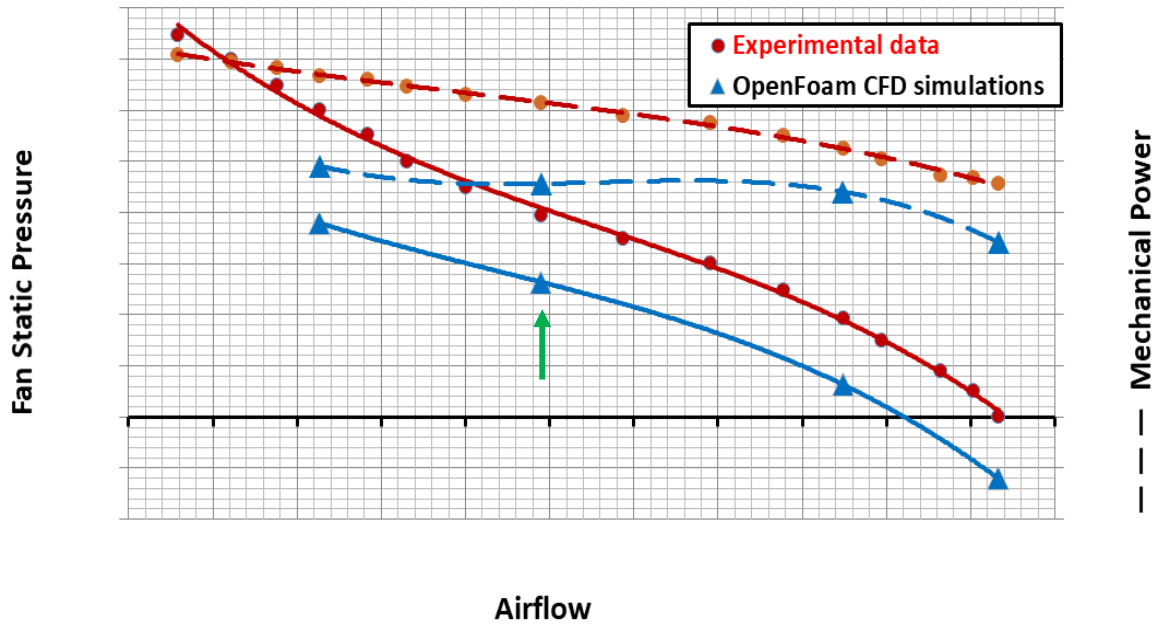

Figure 5: FSP (plain) and mechanical power (dashed) experimental (red dots) vs numerical (blue triangles). Arrow indicates the operating point investigated in Figure 6

Notwithstanding the acceptable match of the performance (at least in the overall trend), the visualization of the flow field shows an unexpected pattern. Specifically, Figure 6-left reports the streamlines as obtained with the steady calculations in *OpenFOAM* for the operating point indicated with a green arrow in Fig. 5 which is the best efficiency point. It can be clearly seen that the outlet side is a huge recirculation system in which it is not straightforward to identify the jet downstream the fan. More precisely, the flow seems to remain attached to the wall in which the fan is inserted, detaching far from where the core jet is expected. A similar analysis has been carried out with ANSYS-CFX. In such software, the boundary conditions and the setup are very similar to the *OpenFOAM* ones (as described in Fig. 3). However, for the same operating point, ANSYS-CFX predicts a flow field (Figure 6-right) which is similar to the one predicted with *OpenFOAM*, even if in both cases the convergence from the standpoint of the residuals (i.e. lower than 10^{-4}) and of the monitored FSP (i.e. variation below the 1 %) are satisfying. This remark, together with the not perfectly matched performance curve, confirmed the need to switch to a transient simulation.

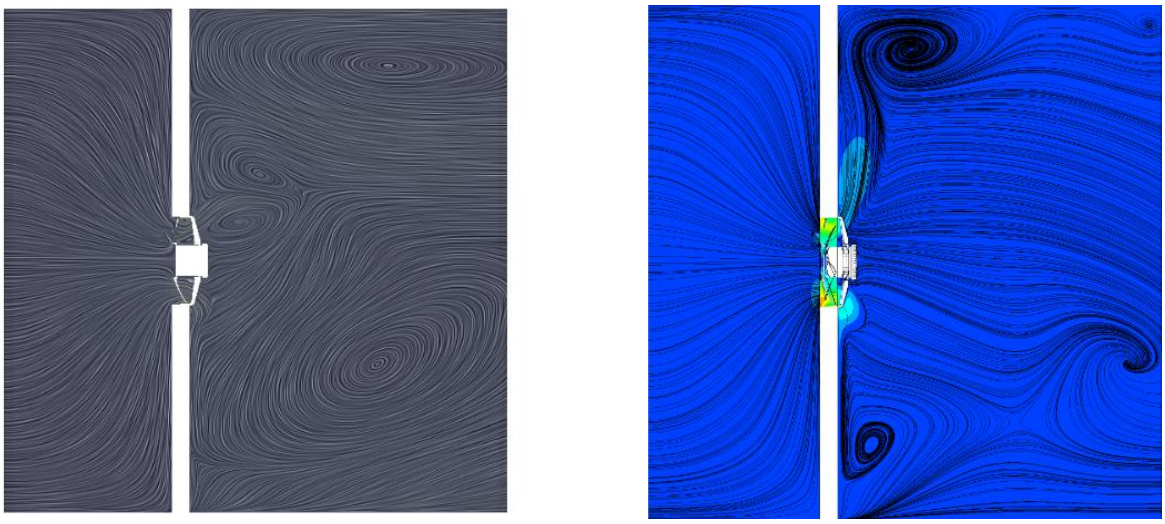

Figure 6: Steady state streamlines: OpenFOAM (left) vs CFX (right)

In this case the *pimpleFOAM* solver has been employed, requiring the capability of a transient approach. For the time discretization, a second order, fully implicit backward scheme has been imposed, maintaining the above-mentioned discretization schemes for the other components of the governing equations. Only two points of the performance curve have been simulated in this case, since for accuracy reason a time step of 0.5° of fan rotation has been imposed, which increases the required computational time. By comparing the transient results with the steady ones and with the experimental performance curve (Figure 7), it can be clearly seen that the working point at the 50 % of free blow flow rate improves noticeably, approaching the experimental performance curve. This applies both to the FSP and power. It should be noticed that the transient quantities are averaged over one revolution, and fluctuations around the 5 % of the monitored quantities are recorded. However, for the free blowing conditions, the transient simulation does not improve the predicted performances. Based on experience, it is believed that those discrepancies might be partially related to the fact that CFD does not include the deformation due to centrifugal forces of the polymeric fan blades. In fact, there is a variation in the rotational speed of roughly 700 rpm moving from the leftmost point to the free blowing condition along the performance curve reported in Fig. 7.

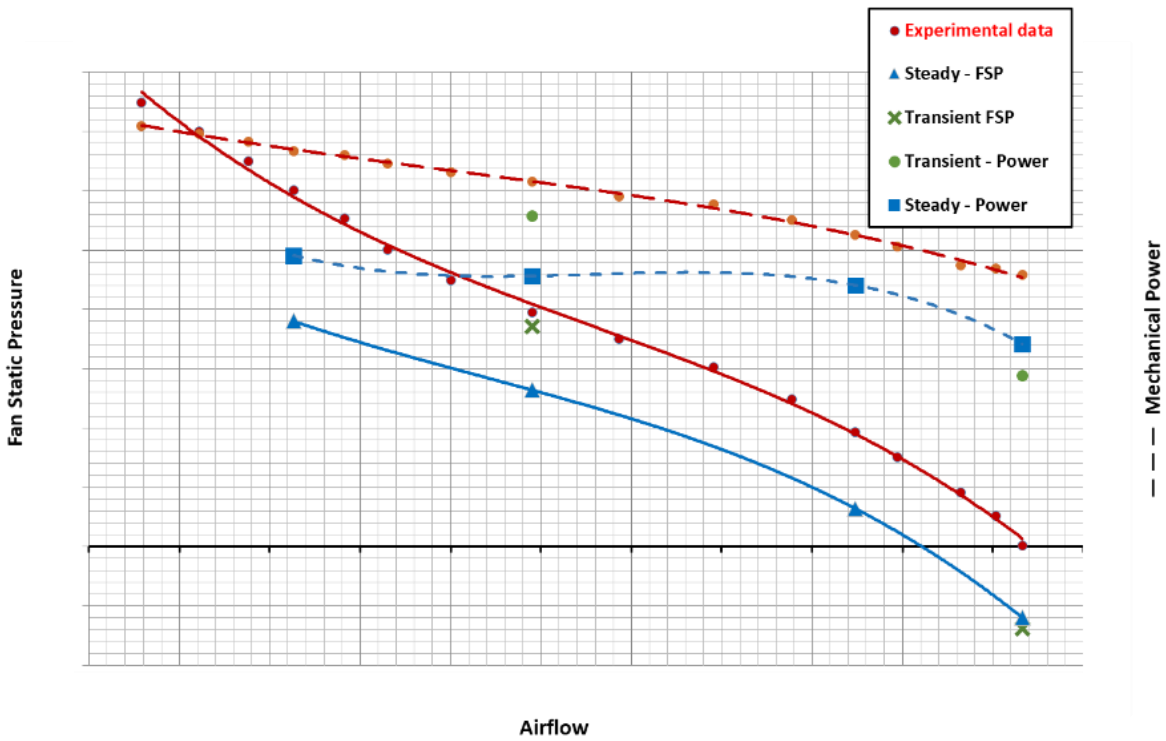

Figure 7: Performance curves: experimental and CFD (steady and transient) results

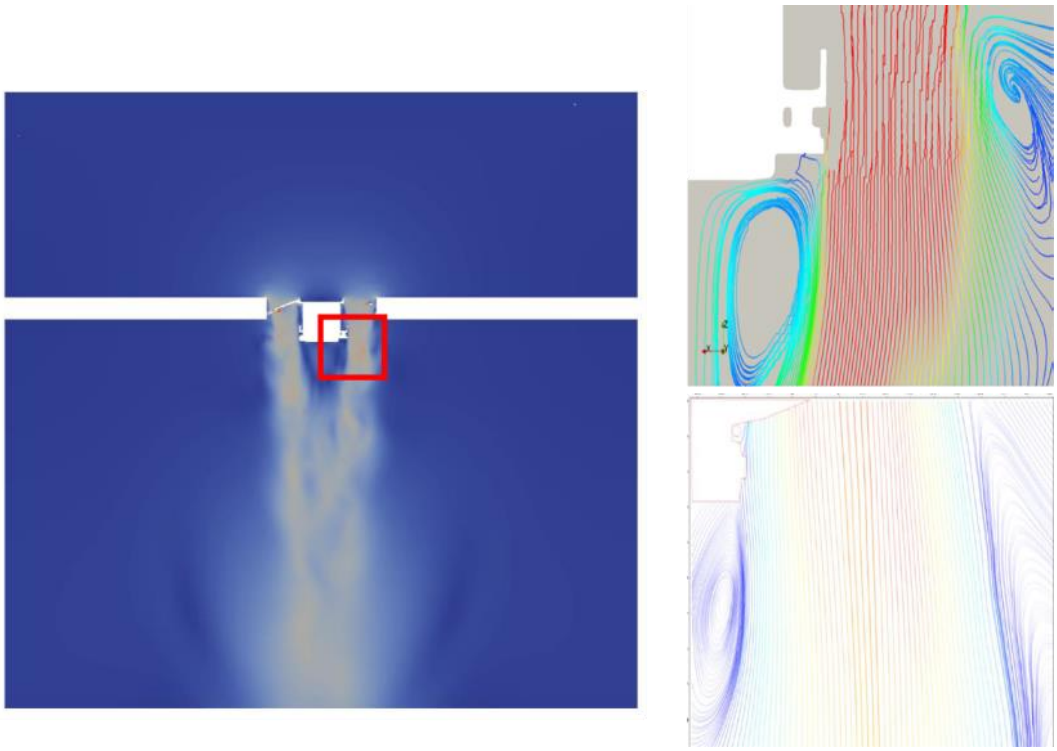

Figure 8: Transient simulation of free blowing condition. Left: velocity contour and inspected area; Right Top: CFD streamlines; Right Bottom: PIV results – The representation of the driver is for reference only.

In the desire of assessing the accuracy of the CFD simulations, the free blowing conditions (i.e. 0 Pa pressure rise) have been investigated, since PIV data were available for this setup. Figure 8 reports a snapshot of the transient simulations both in terms of contour (left) and streamlines (top right). It is immediately clear from this figure that transient simulations are much more suitable for the understanding of the fluid dynamics related to the operation of the fan, as the contour shows a flow

field closer to the expectation. This remark is confirmed by the careful comparison of the CFD streamlines in the boxed area with the PIV data. All the most relevant features are correctly replicated by CFD: the recirculation in the innermost area due to the wake of the motor drive; the recirculation at the outermost area due to the presence of the wall; the jet due to the fan operation in between these two regions. Also the shape of the jet is correctly replicated, showing an increase in its size, moving away from the fan. Additionally, the color scale bounds are consistent for the right cases of Figure 8 and since very similar values are found, a very satisfactory result from the CFD simulation is shown.

CONCLUSION

In this work, the investigation of the aerodynamic performance of an axial fan has been carried out. The *OpenFOAM* suite has been employed and the numerical results have been compared to experimental evidences obtained from two test setups. Two sets of simulations have been carried out:

- Steady state simulations, to replicate the performance curve of the fan, as well as the absorbed power. It has been shown that the overall trend is well captured, even though an offset in the absolute value of around the 15-20 % is recorded for FSP and 5-10 % for mechanical power.
- Transient results that show a relevant improvement of the expected performance, reducing the mismatch in the predicted value, around the 5 %. These values are in line with common proprietary CFD software suite. The flow field in free blowing condition has been analyzed and very good match with PIV results has been found: the goodness of the flow field behind the fan represents a severe test for any CFD software suite, requiring the reliability both of the solution algorithm and of the implemented turbulence model.

BIBLIOGRAPHY

[1] Robertson, E., Choudhury, V., Bhushan, S., & Walters, D. K. (**2015**). *Validation of OpenFOAM numerical methods and turbulence models for incompressible bluff body flows*. Computers & Fluids, 123, 122-145.

[2] Fadiga, E., Casari, N., Suman, A., & Pinelli, M. (**2020**). *CoolFOAM: The CoolProp wrapper for OpenFOAM*. Computer Physics Communications, 250, 107047.

[3] Anderson, M. R., & Bonhaus, D. L. (**2014**, June). *Validation Results for a Diverse Set of Turbomachinery Cases Using a Density Based OpenFOAM® Solver*. In Turbo Expo: Power for Land, Sea, and Air (Vol. 45615, p. V02BT39A020). American Society of Mechanical Engineers.

[4] Aldi, N., Buratto, C., Pinelli, M., Spina, P. R., Suman, A., & Casari, N. (**2016**). *CFD Analysis of a non-Newtonian fluids processing pump*. Energy Procedia, 101, 742-749.

[5] Sedano, C. A., Berger, F., Rahimi, H., Lopez Mejia, O. D., Kühn, M., & Stoevesandt, B. (**2019**). *CFD validation of a model wind turbine by means of improved and delayed detached eddy simulation in OpenFOAM*. Energies, 12(7), 1306.

[6] OpenCFD Ltd 2021, *OpenFOAM® v2006*, https://www.openfoam.com/news/main-news/openfoam-v20-06, accessed 30/09/**2021**.

[7] Juretic, F. (**2015**). *cfMesh user guide*. Creative Fields, Ltd.

[8] Patankar, S. V. (**2018**). *Numerical heat transfer and fluid flow*. CRC press.

[9] Menter, Florian R., Martin Kuntz, and Robin Langtry. *Ten years of industrial experience with the SST turbulence model.* Turbulence, heat and mass transfer 4.1 (**2003**): 625-632.