Transient Three-Dimensional CFD Modelling of Ceiling Fans: A Comparison Between Detailed and Simplified Models

Francesco Babich – Eurac Research, Italy – francesco.babich@eurac.edu
Akshit Gupta – Eurac Research, Italy – akshit.gupta@eurac.edu
Wilmer Pasut – Ca' Foscari University of Venice, Italy / Korea University – wilmer.pasut@unive.it

Abstract

Ceiling fans have been widely used for decades for providing thermal comfort in warm environments. They are an effective means of completely avoiding the use of energy-intensive air conditioning systems in milder environmental conditions and of reducing the use of such systems in more severe and hotter thermal environments. Ceiling fans can generate an immediate cooling effect on people, as they act on both sensible and latent heat exchange between the human body and the moved air. However, one of the major potential limitations of ceiling fans is that they generate non-uniform velocity profiles, and their effect is highly dependent on the mutual position between body and fan. Thus, it is essential to carefully evaluate the position in which they are installed to maximize their cooling effect where needed by people. CFD is a powerful technique for investigating the air velocity field generated by different ceiling fan configurations. Due to its high demand for computational power and the need for having stable models, previous studies proposed different approaches to model ceiling fans in CFD with some simplifications. As the available computational power increases, so does the possibility of creating more realistic models, but too little is known about when the benefits produced by more complex models are overtaken by their computational costs. The aim of this study was to compare the results obtained by using two approaches to include the ceiling fan into a CFD model, namely a detailed model of the geometry of the fan and a simplified implicit approach that emulates the effect of the fan only. The results illustrate that (a) both models capture the main regions of the air flow, (b) the implicit model provided considerably more accurate air speed values, (c) the computational time of the model with blades is one order of magnitude higher, and (d) fan geometry and meshing are the most critical issues in the model with blades.

1. Introduction

Ceiling fans have been widely used for decades for providing thermal comfort in warm environments. They are an effective means of completely avoiding the use of energy-intensive air conditioning systems in milder environmental conditions, and of reducing their usage in more severe and hotter thermal environments (Babich et al., 2017a; Pasut et al., 2014; Schiavon & Melikov, 2008). Ceiling fans can generate an immediate cooling effect on people, as they act on both sensible and latent heat exchanging between the human body and the moved air.

However, one of the major potential limitations of ceiling fans is that they generate non-uniform velocity profiles, and their effect is highly dependent on the mutual position between body and fan (Babich et al., 2021). Thus, it is essential to carefully evaluate the position in which they are installed to maximize their cooling effect where needed by people.

Computational fluid dynamics (CFD) is a powerful technique for investigating the air velocity field generated by different ceiling fan configurations. Due to its high demand for computational power and the need for having stable models, previous studies proposed different approaches to model ceiling fans in CFD with some simplifications (Babich et al., 2017b). As the available computational power increases, so does the possibility of creating more realistic models, but too little is known about when the benefits produced by more complex models are overtaken by their computational costs.

The aim of this study was to compare the results obtained by using two approaches to include the ceiling fan into a CFD model, namely a detailed model of the geometry of the fan and a simplified implicit approach that emulates the effect of the fan only.

Part of
Pernigotto, G., Patuzzi, F., Prada, A., Corrado, V., & Gasparella, A. (Eds.). 2023. Building simulation applications BSA 2022. bu,press.
https://doi.org/10.13124/9788860461919

2. Methodology

In this study, two identical set-ups were created in CFD apart from the ceiling fan. In the former, the fan was modeled as a ring to which a body force was applied, while in the latter the actual geometry of the blades was included. As a result, a moving mesh was used in the latter, while this was not needed in the former. Both models were validated with measured air velocity values.

2.1 Features Common to Both Models

In this study, the key features that are common to both models were taken from a CFD model originally developed for thermal comfort studies (Babich et al., 2017b).

The geometry comprises an environmental chamber in which only a 120-cm-diameter ceiling fan was installed (Fig. 1). 36 monitoring points were placed in the model (3 heights; 12 points per height distributed on each level as shown in Fig. 2) in the very same locations of the original measurements.

An unstructured mesh with ten prism layers (added adjacent to the walls to accurately model the boundary layer near surfaces) was adopted.

Transient simulations were performed to better model the real behavior of the ceiling fan. In the original model, 3-minute simulations were performed. In this study, the simulation time was reduced to 1-minute to decrease the computational effort. To ensure that this was not altering the model results, a preliminary test was performed. Using the original air speed values (available for each time step of the 3-minute simulation for all 36 point), the average air speed in each monitoring point was calculated for a set of 60 s intervals (from 0 s – 60 s to 110 s – 170 s). The results showed negligible variations for a study that aimed at comparing two types of ceiling fan CFD modelling. For most points, the ratio between standard deviation and mean (i.e., the mean of the means of each interval) values was equal to 4 % or less (for instance, in point "centre 1300", the std is 0.06 m/s, the mean 2.05 m/s, and their ratio 2.9 %). Only at a very few points was this ratio higher than 10 %. However, all these points were at a higher level (70 cm or 130 cm above the floor) and far from the fan axis, and therefore in regions in which air speed values are considerably lower and less relevant.

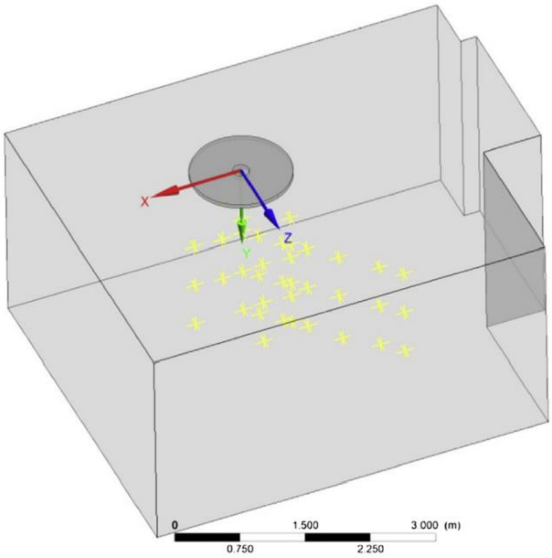

Fig. 1 – CFD model of the environmental chamber. In yellow, 36 monitoring points (Babich et al., 2017b)

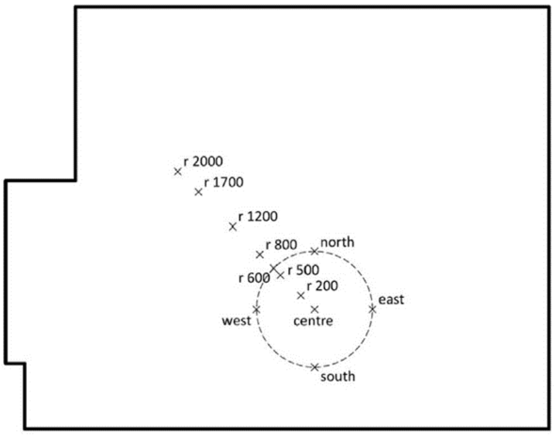

Fig. 2 – Plan view of the environmental chamber with the measurement locations (Babich et al., 2017b)

Convergence criteria were set equal to 1e-05 for the RMS residuals, and an adaptive time step as a function of RMS Courant number was chosen, with the limit for the RMS Courant number set equal to 5. The SST (Shear Stress Transport) k-ω turbulence model was selected as it gave the most accurate results in the original study (best match with measurements).

All CFD simulations were performed with ANSYS CFX version 2021. For the mesh, ICEM CFD was used for its advanced capabilities. All simulations were performed using a workstation equipped with 16 GB RAM and a 6-core Intel Xeon Gold 6154 CPU.

2.2 Fan Implicit Model (Without Blades)

In this former model (Fig. 1), the ceiling fan was modeled as in the original study (Babich et al., 2017b). A ring with the same diameter and distance from the ceiling (30 cm) as the actual fan was created. At the centre of the ring, a cylindrical solid element was added to emulate the fact that, in a real ceiling fan, no air emanates from the centre. A momentum source defined by means of cylindrical components was applied to this ring: axial component 55 kg m^{-2} s^{-2} (push air downwards), theta component 8 kg m^{-2} s^{-2} (rotational movement), and radial component 0 kg m^{-2} s^{-2}. This model led to a 1,933,004-element mesh.

2.3 Fan Explicit Model (With Blades)

In this second model (Fig. 3), the ceiling fan was modeled in more detail by implementing its three blades and central part. To enable the rotation of the fan, two domains were defined in the CFD model, namely a rotating domain (i.e., the cylindrical element shown in Fig. 3), which contained the fan, and a static domain (i.e., the remaining part of the room). One interface was used to link the two domains (lower and upper circles, and vertical side as shown in Fig. 3). In CFX, a "general connection" interface model and "transient rotor stator" were chosen. For mass and momentum, a "conservative interface flux" was used. The selected mesh connection method was "general grid interface" (GGI).
In this case, the model required a rotational velocity as an input, and this was set equal to 290 rpm, which was the corresponding rotational velocity for which the original model (Babich et al., 2017b) was developed. In this model, the total number of mesh elements is 2,088,669 (rotating domain = 217,150; remaining part of the room = 1,871,519).

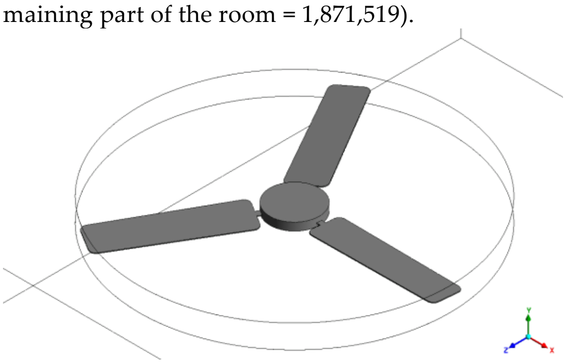

Fig. 3 – Ceiling fan with blades

3. Results

For both models, this section presents their capability of capturing the main regions of the air flow generated by the ceiling fan, the predicted air speed values, and the computational power required.

3.1 Regions of the Air Flow

Both modeling approaches capture the key qualitative features of the air flow generated by a ceiling fan (Fig. 4 and 5). In both cases, the main typical regions of the air flow can be identified (Jain et al., 2004; Wang et al., 2019), namely the region below the fan in which the highest speeds occur (excluding the small area immediately below the motor – also captured by both models), the regions near the floor and then the walls in which air (after having hit the floor) moves horizontally and then vertically to finally return to the blade areas, and lastly the remaining part of the room in which the lowest air speed values are usually recorded. In the region below the fan, like in previous studies (Babich et al., 2017b), the downward flow diverges with a variable half-cone angle depending on which time-step is analyzed.
However, by using the same scale (0.0 m/s to 2.0 m/s) to show the results of both models, it clearly appears that the absolute air speed values generated by the two modeling approaches are considerably different, the values obtained with the detailed model being significantly lower in almost all regions of the flow. This difference is particularly evident in the region below the fan in which the highest values are expected. On the other hand, there are no regions or other qualitative aspects of the flow that seem to be better captured by the detailed model.

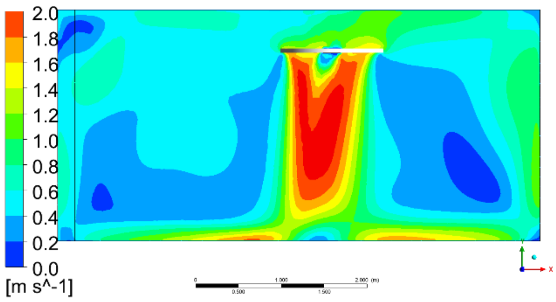

Fig. 4 – Air speed distribution - model without blades (scale from 0.0 m/s to 2.0 m/s)

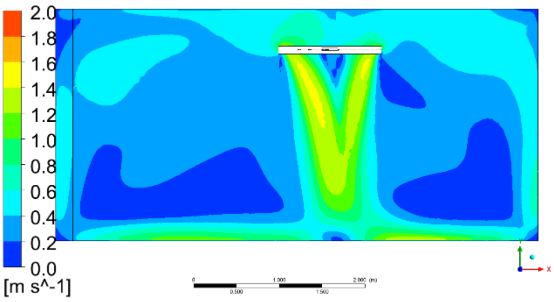

Fig. 5 – Air speed distribution - model with blades (scale from 0.0 m/s to 2.0 m/s)

3.2 Air Speed Values

Comparing the air speed values calculated by the two models and the measured values taken from previous studies (Babich et al., 2017b), in all 36 points the figures of the detailed model are considerably below the predictions of the implicit model (Fig. 6). Moreover, in most points, the output of the model without blades is much closer to the measured values that the predictions of the model with blades.

At the two higher levels (130 cm and 70 cm above the floor), the trend is very similar. Below the center of the fan and in the other points located in the region below the blades (r200 and r500), the air velocity obtained with the detailed model is approximately half the values generated by the implicit model, and the latter are usually in good agreement with measured values. Only in one (point r200 at 130 cm height) does the model without blades also underestimate the air speed, but it is still considerably closer to the measured value.

Focusing on the points that are more distant from the fan (r800 to r2000, i.e., 80 cm to 200 cm away from the fan rotational axis), air speed values are typically below 0.5 m/s, but even in this region the model with blades outperformed the one with blades by providing results that are closer the measurements.

Only in the points placed under the perimeter of the ceiling fan (which are r600, north, west, south and east) the trend is less clear. Both models show limited capability of capturing this region, which is characterized by rapid air speed variations. For the model without blades, this was already noted when it was originally developed (Babich et al., 2017b). On the other hand, modeling the blades did not lead to any noticeable improvement in this region of the flow.

At the lowest level (10 cm above the floor), the model without blades consistently underestimated the air speed values at all 12 points, and in none of them it proved to be superior to the model without blades.

Thus, overall, the model with blades did not lead to better results in any of the regions of the air flow, being considerably less accurate (i.e., less close to measured values) than the implicit model.

3.3 Computational Effort

The computational effort required to complete the simulations is considerably different (Table 1). Using exactly the same workstation and hardware-related settings, the total clock time of the model with blades is one order of magnitude higher than the total clock time of the model without blades. While the former simulation was completed in less than 4 hours, the latter required nearly 9 days.

Table 1 – Computational time

Model	Total clock time	Total clock time
Without blades	1.354e+04 s	3h 45m 40s
With blades	7.643e+05 s	8d 20h 18m 20s

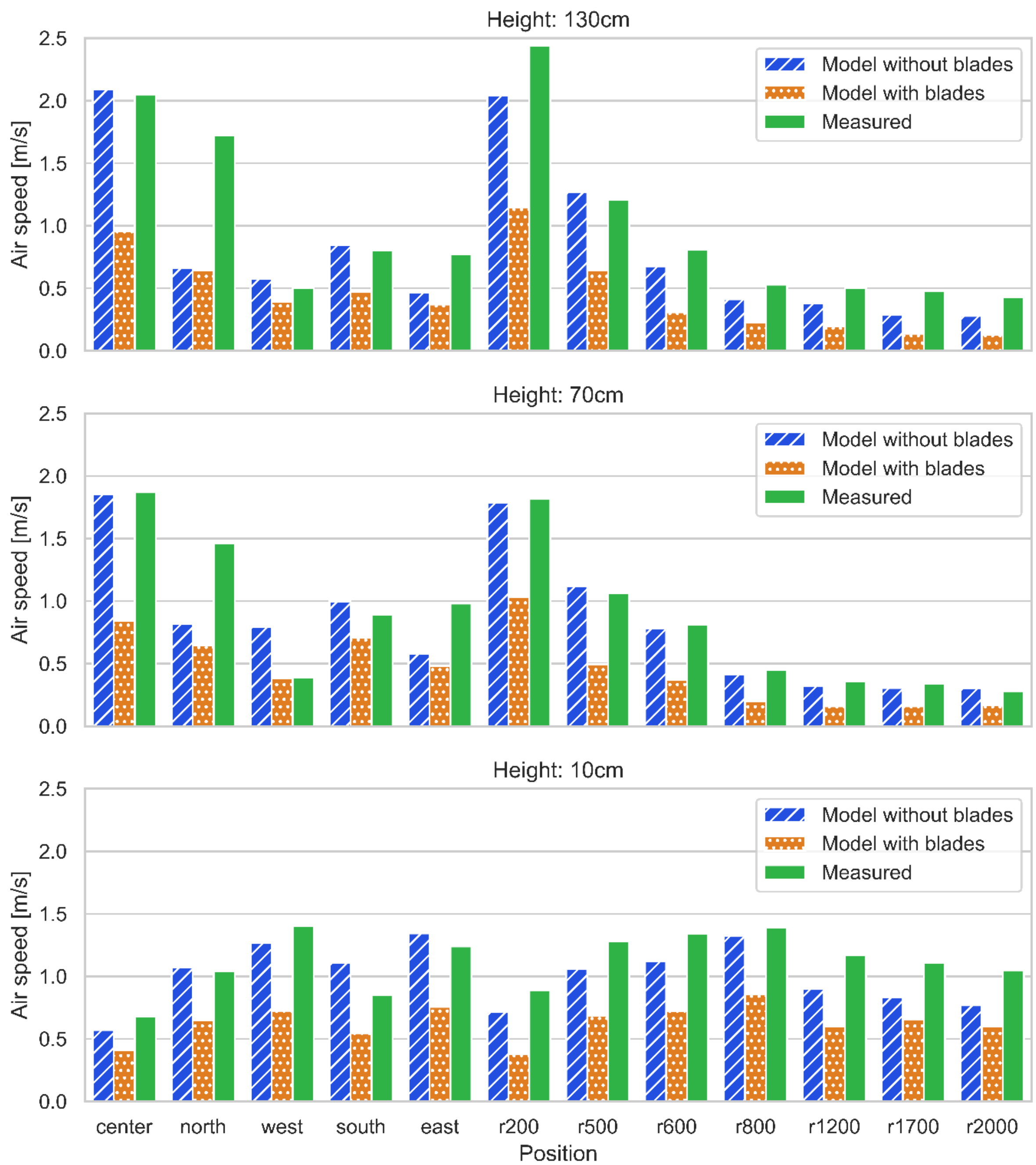

Fig. 6 – Simulated (60 s average) and measured (Babich et al., 2017b) air speed values

4. Discussion

The results of this study showed how the model without blades in which the ceiling fan is implicitly modeled by applying a body force to a cylinder provided more accurate air speed values in less than a tenth of the time, while qualitatively capturing all the main regions of the air flow. Thus, in this study, this modeling approach proved to be superior to a more detailed approach by all aspects. Since the implicit model had previously been fully validated based on experimental data (Babich et al., 2017b), it was expected to be accurate. However, the considerably lower accuracy of the model with blades was less predictable. Therefore, questions arise as to why this happened, and if and when using a model with blades might be the best solution.

In this study, several key parameters were identical in both models. Both used the SST k-ω turbulence mode. For the implicit model, previous studies (Babich et al., 2017b) showed that this turbulence model led to the best agreement with measured values by testing several Reynolds Averaged Navier-Stokes (RANS) turbulence models. For the model with blades, although some tests could be performed, there are no evident reasons to think that the use of a different RANS model, such as the widely used Re-Normalisation Group (RNG) k-ε, might significantly change the results. It might be also possible to explore the usability of other approaches to simulate the turbulence, such as large eddy simulation (LES). While the benefit (intended as better results) should be assessed, an increase in the computational time and in the set-up time would be certain.

Moreover, in both modeling approaches, an adaptive time step was set by using the same target, which is RMS Courant number equal to 5. For the turbulence model, also in this case there is no reason to assume that using a stricter target would considerably affect the results of the model with blades. However, the likelihood of a considerable increase in the computational time would be very high.

Focusing on the model with blades, the elements that are therefore more likely to explain the poor performances are the geometry of the fan, its meshing, and the interface between the two domains (rotating and stationary).

While in the implicit model the geometry is largely simplified (only dimensions of the total and central motor diameters are required), a considerably larger number of geometrical parameters are needed when the blades are modeled. If a computer-aided drafting (CAD) file is made available by the fan manufacturer, then the main effort is usually the cleaning of the geometry to remove all those small details that are required for production, but not CFD simulations, since they would be likely to increase the number of mesh elements and therefore also the computational time. However, an oversimplification might lead to inaccurate results.

On the other hand, if a CAD file is not made available, the geometry must be created by the modeler starting from the measurement taken on the actual fan. In this case, the accurate representation of the blades is usually the most difficult and time-consuming part (especially when there are multiple curvatures and variations of the profile). This is the approach used in this study. Further details could be added to try to improve the model predictions, but this is likely to increase the computational time, too.

Likewise, the use of a finer mesh in the rotating part might enhance the results. In this study, the minimum size of the surface mesh (on the fan surface) was 8 mm. This can be further reduced, and the overall meshing approach investigated in more detail. However, also in this case the computational time is likely to grow.

Focusing on the interface between the two domains, different settings might be evaluated, such as using three separate interfaces for the cylinder that encapsulates the fan to enable its rotation (upper circle, lower circle, and vertical side – Fig. 3). However, there are no evident reasons to expect considerably better results due to a change in the modeling of the domain interface only (ANSYS 2015).

Thus, fan geometry and meshing are likely to be the two most critical issues to be addressed to improve the capabilities of the model with blades. Although this study showed the higher accuracy and lower computational cost of an implicit ceiling fan model, its main limitation is the fact that each rotational speed requires different values for the momentum source, and the definition of the most appropriate values needs experimental data. The outcome might be a discrete set of momentum values (particularly useful for fans with a finite number of rotational speed levels) or a function that calculates the momentum values for any given rotational speed (more appropriate when direct current – DC – motors are used). On the other hand, an explicit CFD model of a ceiling fan uses the rotational speed as an input and therefore would not need experimental validation for multiple rotational speeds.

Thus, the main open question is whether it was better to develop an implicit model (the main effort is the creation of the experimental data) or to opt for an explicit model with blades (the main effort is on geometry and mesh implementation, and then on computational power). Assuming that measurements for different rotational speeds could be taken in one or two days, then the implicit model appears to be the best choice in most cases. However, especially if energy consumption for computing is not considered, this might not be always the case (e.g., if a considerably high computational power such as a high-performance computing -HPC- cluster with hundreds of cores is available).

5. Conclusion

Ceiling fans are an effective means of completely avoiding the use of energy-intensive air conditioning systems in milder environmental conditions, and for reducing their usage in more severe and hotter thermal environments. CFD can be used to predict the air flow generated by ceiling fans, and therefore to better evaluate their capability of delivering comfort cooling.

The aim of this study was to compare the results obtained by using two approaches to include the ceiling fan into a CFD model, namely a detailed model of the geometry of the fan and a simplified implicit approach that emulates the effect of the fan only. The main findings of this study are as follows.

- both modeling approaches capture the key qualitative features of the air flow generated by a ceiling fan, and no relevant difference was noted.
- comparing the air speed values calculated by the two models and the measured values, the implicit model proved to be considerably more accurate. Especially in the regions of the flow in which the most elevated air speeds occur (such as below the fan), the measured figures and the values predicted by the implicit model were close, while the values obtained by the detailed model were much lower (half on the others in several points).
- using exactly the same workstation and hardware-related settings, the total clock time of the model with blades is one order of magnitude higher than the total clock time of the model without blades (less than 4 hours as opposed to nearly 9 days).
- fan geometry and meshing are likely to be the two most critical issues to be addressed to improve the capabilities of the model with blades. However, the use of a more detailed geometry and finer mesh would increase the computational time (and related energy consumption).
- the implicit model appears to be the best choice in most cases.

Considering that the main limitation of the implicit model is the fact that each rotational speed requires different values for the momentum source, and the definition of the most appropriate values needs experimental data, further work will focus on the geometry and meshing of the fan in the explicit model (in which the rotational speed is directly set).

Acknowledgement

The research presented in this paper was performed within H2020 Cultural-E project, which received funding from the European Union’s Horizon 2020 research and innovation programme under grant agreement N. 870072.

References

ANSYS. 2015. “CFX Solver Theory Guide.” Canonsburg, PA, USA.

Babich, F., M. Cook, D. Loveday, R. Rawal, and Y. Shukla. 2017a. “A New Methodological Approach for Estimating Energy Savings Due to Air Movement in Mixed Mode Buildings.” In IBPSA.

Babich, F., M. Cook, D. Loveday, R. Rawal, and Y. Shukla. 2017b. “Transient Three-Dimensional CFD Modelling of Ceiling Fans.” *Building and Environment* 123: 37–49. doi: https://doi.org/10.1016/j.buildenv.2017.06.039

Babich, F., W. Pasut, and A. Belleri. 2021. “The Use of Ceiling Fans in Reverse- Flow Mode for Comfort Cooling : Physiological Aspects and Relationship with International Standards.” *ASHRAE Transactions* 127(2).

Jain, A., R. R. Upadhyay, S. Chandra, M. Saini, and S. Kale. 2004. “Experimental Investigation of the Flow Field of a Ceiling Fan.” In *Proceedings of the ASME Heat Transfer/Fluids Engineering Summer Conference* 2004, HT/FED 2004, 3:93–99. American Society of Mechanical Engineers. doi: https://doi.org/10.1115/ht-fed2004-56226

Pasut, W., E. Arens, H. Zhang, and Y. Zhai. 2014. “Enabling Energy-Efficient Approaches to

Thermal Comfort Using Room Air Motion." *Building and Environment* 79: 13–19. doi: https://doi.org/10.1016/j.buildenv.2014.04.024

Schiavon, S., and A. K. Melikov. 2008. "Energy Saving and Improved Comfort by Increased Air Movement." *Energy and Buildings* 40(10): 1954–60. doi: https://doi.org/10.1016/j.enbuild.2008.05.001

Wang, H., H. Zhang, X. Hu, M. Luo, G. Wang, X. Li, and Y. Zhu. 2019. "Measurement of Airflow Pattern Induced by Ceiling Fan with Quad-View Colour Sequence Particle Streak Velocimetry." *Building and Environment* 152: 122–34. doi: https://doi.org/10.1016/j.buildenv.2019.02.015