INFLUENCE OF THE ENCLOSURE ON THE PERFORMANCE OF RADIAL FANS

Georg KLEPP, Walter KAMPHAUSEN

IFE, Hochschule Ostwestfalen-Lippe,
Liebigstraße 87 32657 Lemgo, Germany

SUMMARY

For certain applications radial impellers are mounted in square enclosures (e.g. plug fans, plenum fans). The performance of this type of fans with one radial outlet is analyzed numerically. The influence of the size and mounting positions on the fan performance is investigated. When the distance of the impeller to the side walls gets too small the flow in the enclosure is choked. The flow patterns are influenced by the volume flow as well as the swirl generated by the impeller.

INTRODUCTION

Due to restrictions in the space that is available for mounting fans, a fan sometimes has to work in a narrowed enclosure. This leads to reductions in the fan performance and to an increase in fan noise. For practical design applications it is necessary to assess this reduction in advance. In order to facilitate this, the flow patterns in the enclosure and their influence on the fan performance are investigated. As we try to get results to support the design process, we restrict ourselves to configurations and operational conditions which seem technically important.

A common arrangement in HVAC and industrial cooling applications is a radial impeller confined in a square housing. The outflow can be either in axial direction or in radial direction. The data available in the literature is scarce, [1], [2], [3], [4]. As there is no appropriate guiding (i.e. by a spiral casing) of the outflow from the impeller an increase in pressure is not expected and if the outflow is significantly restrained additional pressure losses will occur [1]. Some hints are available in the manufacturers documentation, in order to assess the fan performance reduction and the influence of narrowed enclosure on the fan characteristic. These information are only applicable to the measured impeller and enclosure geometry.

For axial outflow from the enclosure a penalty factor, which describes the reduction in volume flow rate, is defined. This parameter can be defined either as a function of the hydraulic diameter or as a function of the area of the outlet section. For a minimum distance (impeller outlet to enclosure wall) of 0.3 impeller-diameter the devaluation is small (3% in [1]). For smaller distances (i.e. 0.1 impeller-diameter) the devaluation gets significant (7% - 8% in [1]). The influence of the location in the enclosure is less important. This corresponds to the data given by the manufacturer [2], [3].

For impellers mounted in an enclosure with one radial outflow the data is even scarcer. In [4] a factor is derived for the required increase in rotational speed and power requirement of the fan to ensure a given operating point. The factor depends on the distance of the nearest wall as a fraction of impeller diameter and the flow number. For an impeller in a symmetric square enclosure an influence has to be taken into account for distances smaller than 0.5 impeller diameter to the wall of the enclosure.

Here we will try to examine the geometry influence of a square enclosure with radial outlet on the performance of a radial fan.

METHODOLOGY

For this investigation the complex three dimensional transient flow is described using a simplified two dimensional stationary model, significantly reducing the complexity of the problem to the main design parameters.

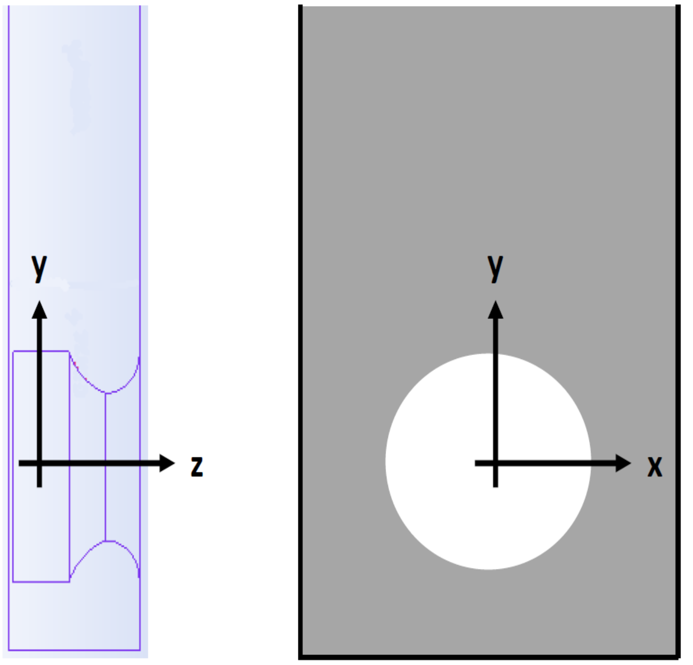

Figure 1: Plane z=0 used as twodimensional (x-y plane) computational domain in the simulation

The flow exits the impeller and enters the enclosure. In the enclosure a complex transient three dimensional flow field develops. An analysis of this flow field would be expensive. In order to facilitate the computations, the flow domain is reduced to a two dimensional configuration, where only the main geometric parameters are considered. Thus the geometry parameters left are the shape, length and height of the enclosure, the impeller diameter and the location of the impeller. The depth of the enclosure and the height of the impeller are neglected in this analysis, figure 1. The flow at the exit of the impeller is reduced to an average velocity and an average (flow) angle in accordance to the one dimensional theory (Euler equation). The flow pattern in the enclosure thus depends only on the velocities, kinematic viscosity, size of the enclosure and location of the impeller.

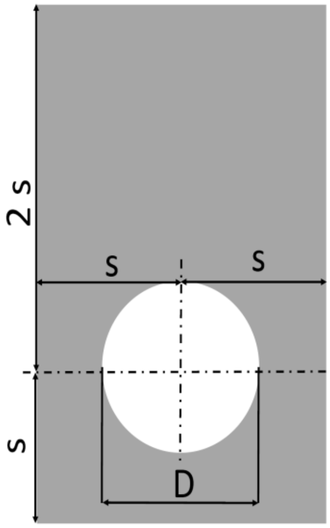

Figure 2: Geometrical parameters

The parameters are scaled accordingly to define suitable dimensionless parameters, which are used in the analysis. Thus the results can easily be generalized. To describe the geometry the center of the impeller is used as a reference point and the impeller-diameter is used as a scaling unit. The geometry of the enclosure is described using the distance of the impeller center to the walls and the outlet. In order to normalize the enclosure geometry, just one length s is used to define the enclosure: Distance s to the walls, distance 2s to the outlet, see figure 2. The length s is scaled by the outer impeller diameter D. The inlet velocities consist of two components, the normal velocity c_{2m} and the tangential velocity c_{2u}. Both are scaled by the tangential velocity u_2 of the impeller at the outer diameter. The resulting pressure values are thus accordingly scaled by the fluid density and the square of the the tangential velocity of the impeller at the outer diameter.

The computations are performed for a Reynolds number of 10^{-6} corresponding to an impeller with an outer diameter of 400 mm rotating with 1440 rpm and air at ambient conditions.

Different types of impellers are tested: a slowrunner with a small radial velocity and a big tangential velocity (i.e. small volume flow and big swirl) and a fastrunner with a big radial velocity and a small tangential velocity (i.e. big volume flow and small swirl).

Table 1:Data for the slowrunner and fastrunner case, in accordance with Bommes 2.3.2, [5]

	fastrunner	**slowrunner**
$\mathbf{D_1/D_2}$	0.7	0.28
β_1	25°	45°
$\beta_{2,\ blade}$	35°	55°
$\mathbf{b_2 / b_1}$	0.85	0.64
$\mathbf{c_{2m} / u_2}$	0.267	0.174
$\mathbf{c_{2u} / u_2}$	0.433	0.702

The analysis is performed using the computational fluid dynamics code openFoam 4.0. to compute the scaled Navier-Stokes equations.

The walls of the enclosure are at the bottom at the left and right side. The outlet is at the top, here a constant pressure is prescribed as a boundary condition. Different sizes of the enclosure were computed. An inlet boundary condition is at the impeller outlet, where the velocity is prescribed. Here the impeller characteristic is implemented yielding the velocity components in normal and tangential direction.

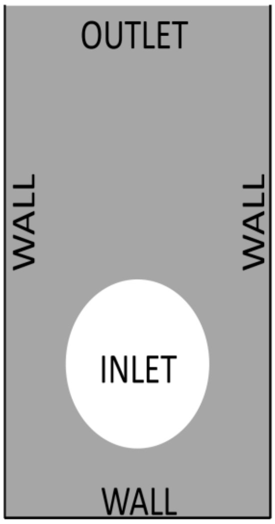

Figure 3: Computational domain with boundary conditions

Block structured hexaedral grids are used for the computations. The discretization of the finite volume scheme is second order accurate, the SIMPLE algorithm is used. The turbulence is modeled using the k-epsilon model with wall functions. The convergence of the solution is established for residuals lower than 10^{-5}.

In order to assess the influence of the enclosure on the fan performance the pressure difference between inlet and outlet is computed. If the pressure difference between inlet and outlet is zero, the enclosure does not influence the performance of the impeller. If the pressure at the impeller outlet is bigger than at the outlet of the enclosure, the performance of the impeller is diminished. If the pressure at the impeller outlet is smaller than at the outlet of the enclosure, then a part of the dynamic pressure is converted to static pressure by the enclosure.

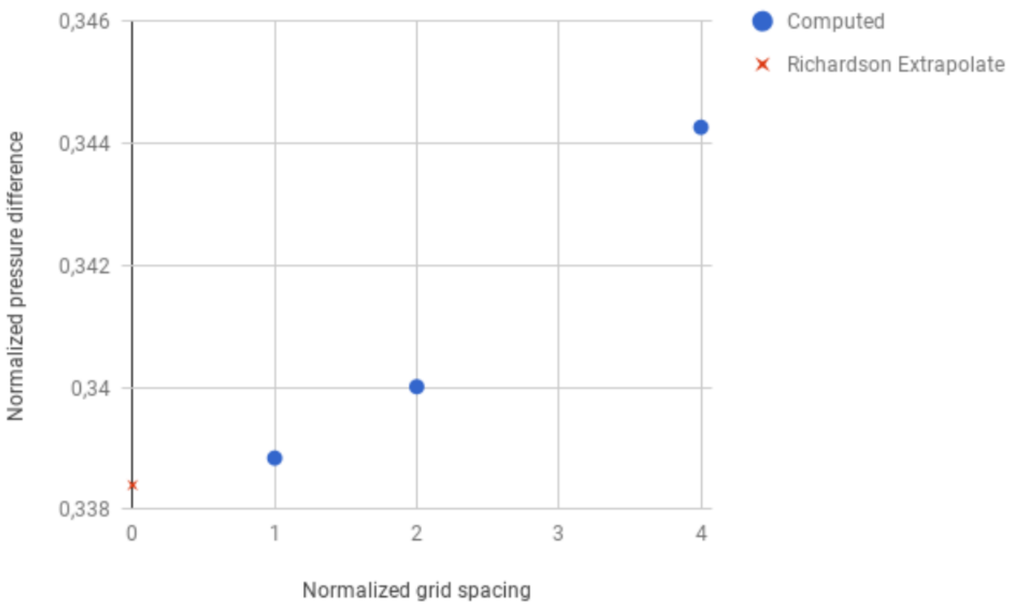

Figure 4: Grid convergence study: scaled pressure difference vs. normalized grid spacing

For the case of an enclosure lengthscale of 0.6 a grid convergence study was performed according to the procedure outlined in [6]. The smallest grid spacing used for normalization was 0.0075, corresponding to a grid size of 32,000. Acceptable results within an error smaller 1% can be achieved for a grid spacing smaller 0.015. The y^+ values in the computations are in the range of 10 to 400, depending on the evolving flow patterns.

SYMMETRIC ENCLOSURE

In a first step the fan is arranged symmetrical in the enclosure. The distance of the fan to all the walls of the enclosure is similar. For most of the computations, the distance to the outlet of the enclosure is twice as big. Thus the geometrical parameters describing the arrangement are reduced to one value.

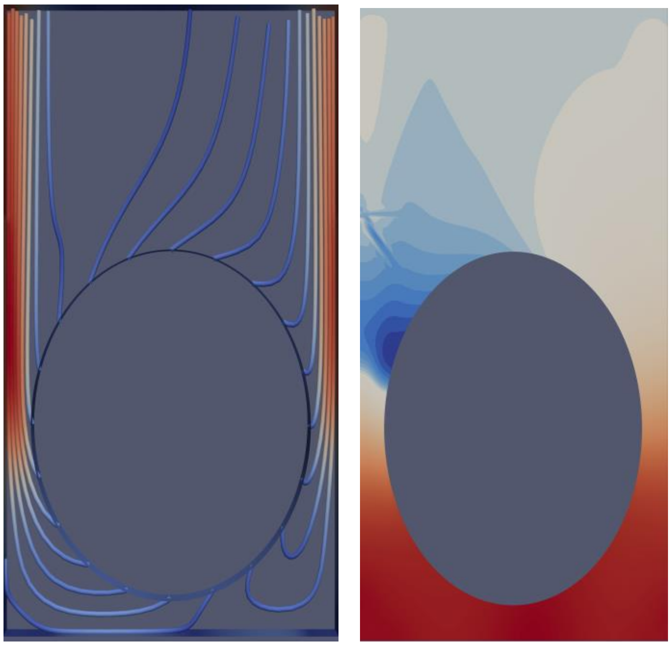

Figure 5: Flow patterns fastrunner s/D=0.6. Streamlines (left) coloured by velocity magnitude, pressure distribution (right). (Colour map: Maximum values red minimum values blue)

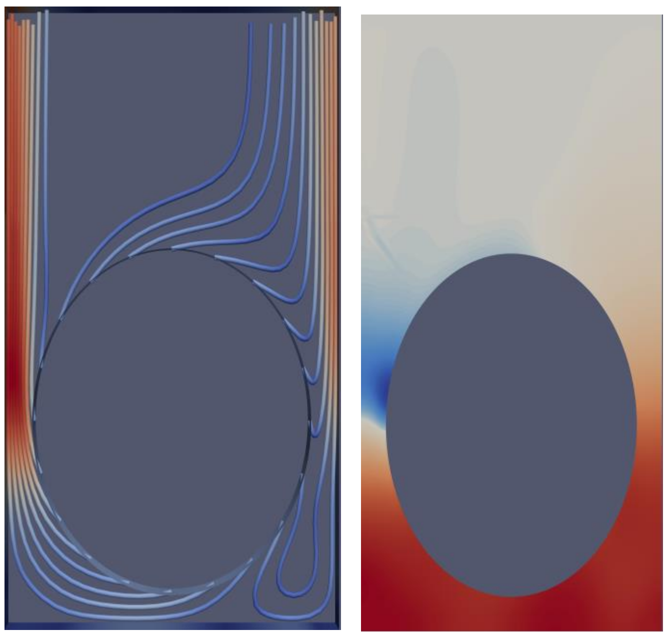

Figure 6: Flow patterns slowrunner s/D=0.6. Streamlines (left) coloured by velocity magnitude, pressure distribution (right). (Colour map: Maximum values red, minimum values blue)

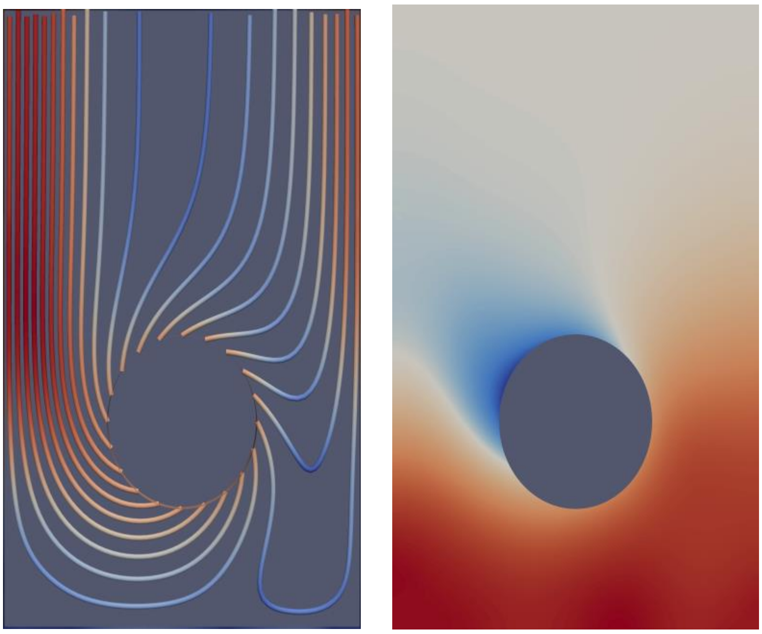

Figure 7: Flow patterns fastrunner s/D = 1.2. Streamlines (left) coloured by velocity magnitude, pressure distribution (right). (Colour map: Maximum values red minimum values blue)

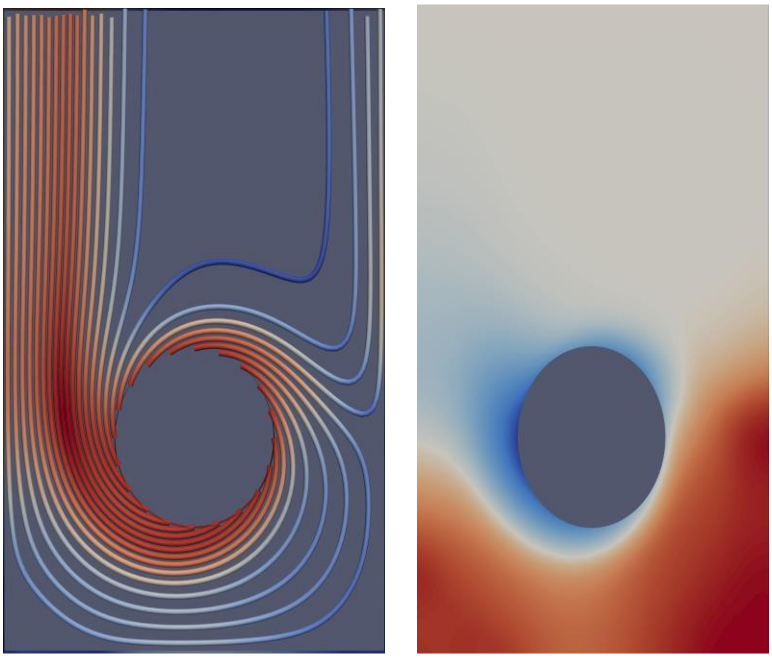

Figure 8: Flow patterns slowrunner s/D = 1.2. Streamlines (left) coloured by velocity magnitude, pressure distribution (right). (Colour map: Maximum values red minimum values blue)

Flow patterns

When there is no enclosure the flow pattern corresponds to the potential flow solution.

When the enclosure is big enough, the velocities at the wall are small compared to the velocity at the impeller and thus the influence of the enclosure on the flow field is negligible.

As the enclosure narrows the flow is redirected by the wall. This influence leads to the formation of vortices and additional dissipation, thus reducing the static pressure and the total pressure at the outlet. In this way the overall fan performance in the enclosure is reduced. If the distance is fitting there might even be a small pressure rise at the outlet, as a part of the dynamic pressure is conversed to static pressure. With a further decrease in the wall distance the dissipation and thus the performance limitations are predominant.

As the enclosure narrows the outflow gets more unevenly distributed with a maximum at the side of the walls and a minimum in the middle. The flow leaving the impeller facing the side wall is choked, the minimum distance between wall and impeller acting as a throat.

The pressure in the enclosure and also at the outlet of the impeller is unevenly distributed: A pressure maximum is at the wall opposite to the outlet. The smaller the distance to the opposite wall of the enclosure, the bigger the pressure gets. In a spiral casing the pressure at the impeller outlet would be more equally distributed. The uneven pressure distribution has a negative influence on the fan performance and fan acoustics.

Increasing the distance of the outlet does not change the flow pattern significantly.

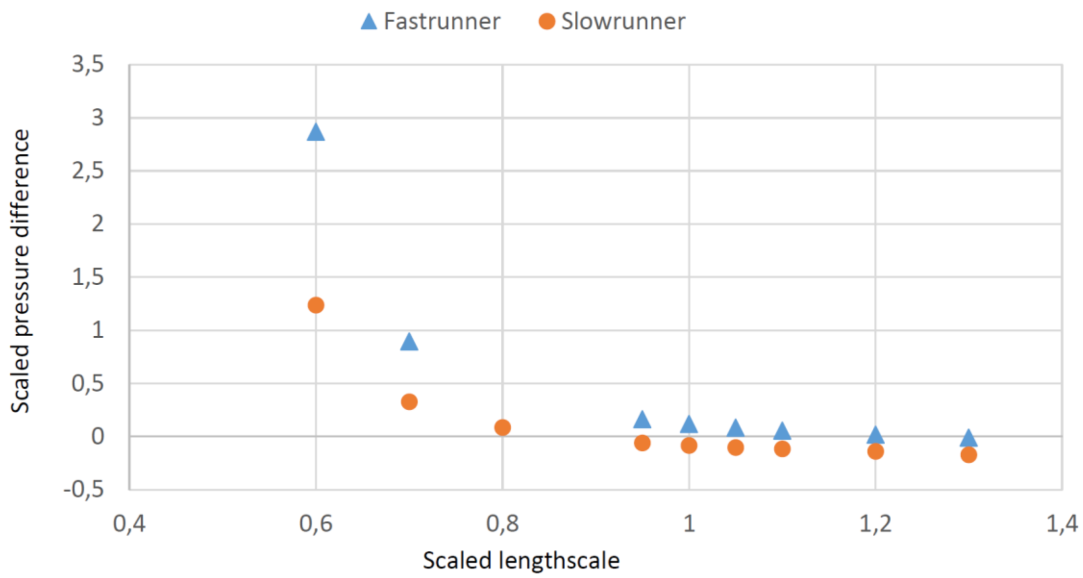

Figure 9: Scaled pressure difference vs. scaled enclosure size for fastrunner and slowrunner

Influence of geometry on performance

In order to describe the influence on the fan performance the dimensionless pressure difference between fan outlet and enclosure outlet is used. Using this pressure difference and relating it to the pressure for the design point, the devaluation of the fan performance is quantified. Using this devaluation a penalty factor can be derived. The geometry of the enclosure can be characterized by one geometric parameter: the distance between wall and impeller outlet scaled by the impeller diameter.

When the distance is smaller then 0.4 impeller-diameter (corresponding to s/D = 0.9) the flow pattern changes significantly and the reduction in fan performance gets so significant, that its operation is significantly restricted. This coincides approximately with the results for the axial outlet, where the fan characteristic changes significantly with a minimum distance smaller 0.3 impeller-diameter, but with a radial outlet the penalty is higher.

The impeller characteristic has a strong impact on the performance restrictions caused by the walls. For fastrunners with a high flow rate the choking effects occur earlier then for the slow runners with less volume flow rate. On the other hand the influence of the wall distance on the efficiency is stronger for the slowrunners. Thus the influence of the side walls also strongly depends on the outflow conditions of the impeller. The volume flow rate, characterized by the normal velocity, has a significantly bigger influence then the fan pressure or swirl, characterized by the tangential velocity.

Influence of impeller outflow on performance

The influence of volume flow rate (normal velocity at impeller outlet) and swirl (tangential velocity) is analyzed next, by changing the ratio of the values of the tangential and radial velocity. The influence of the geometry for the two velocities being equal is computed first. The dependencies are similar to the results for the slowrunner and fastrunner.

For a distance of 0.3 impeller-diameter on all sides the radial velocity is changed whereas the tangential velocity rests constant. Increasing the radial velocity, i.e. volume flow, significantly increases the pressure penalty.

Next for a distance of 0.3 impeller-diameter on all sides the radial velocity stays constant whereas the tangential velocity changes. Here the influence is much less distinct. An increasing swirl component in the flow leads to slight reduction in the pressure penalty.

This confirms the assumption that the main reason for the pressure drop lies in the choking of the flow.

ASYMMETRICAL ENCLOSURE

The symmetrical arrangement is common, but not optimal. An offset in the fans position may reduce the negative effects of the enclosure. In order to enhance the performance of the fans inside the enclosure the choking effect of the walls should be mitigated. So an asymmetric arrangement of the fan in the enclosure is investigated.

Significant geometric parameters are the minimum distance of the impeller outlet to the wall on all three sides of the enclosure. The influence of the variation of the three distances adds significantly to the complexity of the problem: there are two more additional geometric parameters which also influence one another.

The analysis of the flow patterns suggests that the influence of the distance gets more important as the velocity in the cross section between impeller and wall increases Thus the biggest influence comes from the cross section with the biggest velocity, but the influence of the other distances may not be neglected.

A possibility to optimize the arrangement is to arrange the impeller in a similar way as in a spiral casing, i.e. the minimum distances to the walls have similar proportions as the corresponding radii of a spiral casing. For this case this leads to following ratios of the distances wall “inflow” side 0.7 , wall opposite to outlet 0.85 and wall “ouflow” side 1. This leads to a pressure difference that is smaller than for a symmetric enclosure.

CORRELATION

For an axial outflow the penalty on the fan performance can be correlated with geometrical quantities derived from the outlet area. With a radial outflow the outlet area is not the significant parameter, but the minimum distances of the impeller to the side walls. In order to reduce the geometric parameters only the symmetric arrangement, with one geometric parameter is considered. The influence of the volume flow rate has to be taken into account, whereas the swirl and thus the flow angle are of minor importance. In order to have a substantial data base for the correlation computations were performed for lengthscales of the enclosure s ranging from 0.6 to 1.3, scaled normal velocities c_{2m} / u_2 ranging from 0 to 10 and scaled tangential velocities c_{2u} / u_2 ranging from 0 to 10.

Using the results of the computations, leads to the following correlations to describe the influence of the normal velocity at the outlet c_{2m} (flow rate), the tangential velocity c_{2u} (swirl), and the length scale s (wall distance and diameter). Two distinct flow regions are identified:

For a length scale s/D bigger 0.9 the flow is slightly influenced by the enclosure

$$\Delta p / \rho = (0.5\ s/D^{-6})\ c_{2m}{}^{2} + (0.02774\ s/D - 0.0669)\ c_{2u}{}^{2} \tag{1}$$

For a length scale s/D smaller 0.9 the performance is significantly influenced by the enclosure

$$\Delta p / \rho = (0.61\ s/D^{-4.1})\ c_{2m}{}^{2} + (-\ 0.042)\ c_{2u}{}^{2} \tag{2}$$

From these correlations the significant influence of the normal velocities and the length scale is obvious. The 92 results of the CFD simulation are appropriately reproduced with the correlation equations 1 and 2, with an error margin of 12 % (biggest errors for very small values less then 0.1) as shown in figure 10.

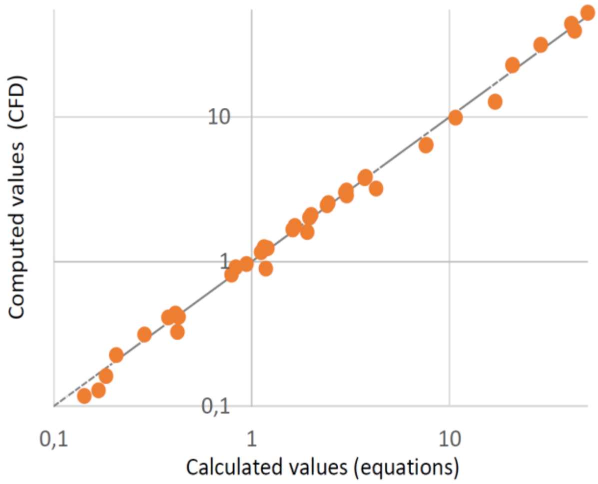

Figure 10: Correlation of computed and calculated values for the pressure difference as a function of enclosure size

A penalty factor may be derived in terms of the pressure values. In practical applications and measurements the performance reduction is realized as a reduction in volume flow, as the pressure difference between enclosure outlet and fan inlet stays constant.

CONCLUSION

Using a simple two-dimensional stationary model the flow of radial fans in an enclosure with one radial outlet was computed. In this model only the main parameters are considered. Due to the restriction of the flow, the fan performance is reduced. The most important parameter is the distance between fan outlet and wall on the “outflow” side. If this distance is reduced too much, the flow is severely choked. Other parameters which strongly influence the flow, are the distance to the other side walls and the volume flow rate. For a wall distance smaller than 0.4 impeller diameter the flow in the enclosure is restrained and a significant pressure loss occurs. The pressure loss due do the influence of the enclosure may be estimated using equations (1) and (2).

Compared with the axial ouflow the size of the enclosure which restrict the flow is of similar magnitude. In contrast the location of the impeller is significant for radial outflow. An optimum arrangement is achieved by using the geometric proportions of the wall distances to impeller outlet similar to the dimensions of the corresponding radii of a spiral casing.

Using a two-dimensional stationary model the flow was significantly simplified. Thus a reduction to the most important geometric and flow parameters was possible. The comparison of flow patterns and the correlation of results was thus greatly facilitated. On the other hand, the real three-dimensional transient flow might differ from this simplified model, thus reducing the applicability of the correlation for practical application. Here only the comparison with three-dimensional transient CFD-simulations and especially as many measured values as possible can help.

REFERENCES

[1] O. Sadi - *Experimental investigation to determine the influence of a air handling unit on the characteristics and acoustics of Ziehl-Abegg centrifugal plug fans*. Ziehl-Abegg, Künzelsau **2006**

[2] Ziehl-Abegg – *Radialventilatoren Hauptkatalog* – Künzelsau **2013**

[3] EBM - *EC-Radialventilatoren RadiPac* Mulfingen **2017**

[4] Twin City Fans & Blowers – *High efficiency plug fans. Catalog 360*. Minneapolis, **2013**

[5] L. Bommes ed. – *Ventilatoren* Vulkan Verlag, Essen **2003**

[6] P. J. Roache – *Quantification of uncertainty in computational fluid dynamics* Annu. Rev. Fluid. Mech. 29:123–60 **1997**