Use of perforated acoustic panels as supply air diffusers in diffuse ceiling ventilation systems

Iqbal, Ahsan; Kazemi, Seyed Hossein; Ardkapan, Siamak Rahimi; Afshari, Alireza

Publication date:
2013

Document Version
Publisher's PDF, also known as Version of record

SBi 2013:12

Use of perforated acoustic panels as supply air diffusers in diffuse ceiling ventilation systems

Danish Building Research Institute
AALBORG UNIVERSITY

Use of perforated acoustic panels as supply air diffusers in diffuse ceiling ventilation systems

Ahsan Iqbal
Hossein Kazimi
Siamak Rahimi
Alireza Afshari

SBi 2013:12
Danish Building Research Institute, Aalborg University · 2013

Title	Use of perforated acoustic panels as supply air diffusers in diffuse ceiling ventilation systems
Serial title	SBi 2013:12
Edition	1 edition
Year	2013
Authors	Ahsan Iqbal, Hossein Kazimi, Siamak Rahimi, Alireza Afshari
Language	English
Pages	18
References	Page 18
Key words	Ventilation, Troldtekt acoustic panels, CFD, computational fluid dynamics.
ISBN	978-87-92739-33-9
Drawings	Ahsan Iqbal, medmindre andet er angivet
Photos	Ahsan Iqbal, medmindre andet er angivet
Cover	Troldtekt
Publisher	SBi, Statens Byggeforskningsinstitut, Aalborg Universitet, Danish Building Research Institute, Aalborg University A.C. Meyers Vænge 15 DK-2450 København E-post sbi@sbi.aau.dk www.sbi.aau.dk

Content

Preface

Troldtekt has been manufacturing perforated acoustic panels for the last 13 years. The panels can be used not only in applications related to acoustics but also as low pressure drop supply air diffusers, particularly in diffuse ceiling ventilation systems. Troldtekt has performed experiments in order to investigate the performance of the perforated acoustic panels as supply air diffusers. The aim of the present study is to theoretically verify the performance. Empirical findings of the experiments are examined through the use of CFD, computational fluid dynamics.

The authors would like to express their gratitude to all those who helped completing this report. The authors are grateful to Prof. Dr. Peter V. Nielsen for his time to review the CFD results and to Senior researcher Niels C. Bergsøe for helping us accomplish the task.

Danish Building Research Institute, Aalborg University
Department of Energy and Environment
April 2013

Søren Aggerholm
Head of the department

Executive Summary

The Troldtekt acoustic panels are extensively tested Danish products, which has undergone continuous development and been used in the construction industry for more than 75 years. For last 13 years, Troldtekt has been developing acoustic panels that are perforated and can be used as supply air diffusers. The pressure drop through these perforated acoustic panels/diffusers is proportional to the superficial velocity. However within reasonable working range of superficial velocity (0-0,04 m/s), the pressure drop does not exceed 2Pa. The company has tested the pressure drop through several acoustic panels in a real building. In this report, the experimental results regarding the performance of perforated acoustic panels from Troldtekt are verified through computational fluid dynamics (CFD) techniques. The CFD simulations were performed to verify whether the perforated acoustic panels can be used as low pressure drop airflow diffusers or not. Troldtekt performed experimental and the Danish Building Research Institute (SBi) – Aalborg University Copenhagen did the CFD simulations.

This report concerns the findings of the CFD simulations. The dimensions of the model space used in the CFD simulations was exactly the same as the dimensions of the real space where these perforated acoustic panels are physically installed. However, the locations of perforated acoustic ceiling panels were arbitrary. The pressure drops through the perforated acoustic panels were examined according to the airflow rate through the panels. The CFD results were compared with the experimental results. The maximum deviation of the CFD results from experimental results is below 15%. However, at superficial velocities lower than 0,03 m/s, the error is less than 1%. The CFD techniques show very promising results within a reasonable time. Therefore, CFD techniques can be used to design ventilation systems with Troldtekt perforated acoustic panels.

Introduction

Fresh and clean outdoor air is supplied to buildings to dilute the contaminants, odour and excess heat from the building interior. It is important that the inhabitants perceive the ventilated spaces as comfortable. Therefore, the supply air should reach all parts of the occupied zones. In fluid dynamics, the term “diffuser” is used for a device that decelerates a fluid and enhances the mixing with the surrounding fluid [1, 2]. Therefore, in HVAC systems the diffusers (along with grills, nozzles etc.) are often used to supply air into the required spaces at a relatively lower velocity than supply air ducts. Conventionally, ducts are used to supply air from a mechanical fan to the required space through diffusers (or grills, nozzles etc.). The pressure drop in the system depends on the ducting design and the pressure drop on the diffusers. One of the aims of the design of an air distribution system is to find the limits of the diffuser regarding possible flow rates into the room and temperature differences between the supply and return air temperatures. This means to find the limits that maintain an acceptable comfort level with little draught and low temperature gradients in the room in the case of cooling [3]. Draught occurs if the kinetic energy of the airflow is not absorbed correctly in the room air volume. This is often the case in normally used ventilation systems, with highly concentrated air inlets [4]. Therefore, low pressure air diffusers are needed as well as alternative air distribution systems.

Troldtekt acoustic panels and air diffusers

The Troldtekt acoustic panels are extensively tested Danish products, which has undergone continuous development and been used in the construction industry for more than 75 years. Recently Troldtekt has developed acoustic panels that are perforated and can be used as supply air diffusers. The pressure drop through these perforated acoustic panels/diffusers is proportional to the superficial velocity. However, within reasonable working range of superficial velocity (0-0,04 m/s) the pressure drop does not exceed 2 Pa. The company has tested the pressure drop through several acoustic panels and the results for one type of panel are given in Figure 1. Three panels (of the same type) were tested. However, the data from Panel 3 was refused due to the high standard deviation in its weight.

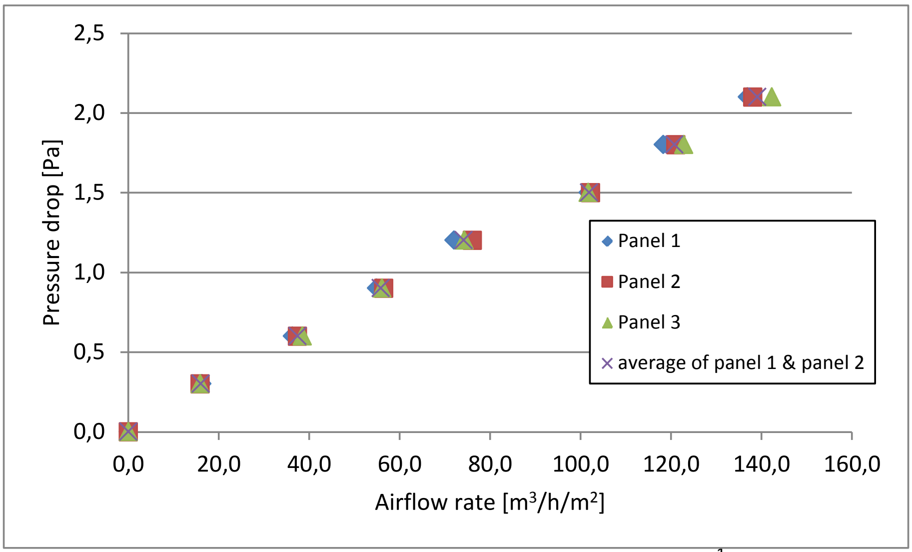

Figure 1 The Troldtekt low pressure drop air diffusers (perforated acoustic panels) [1]

Diffuse ceiling ventilation

Diffuse ceiling ventilation (DCV) is an alternative to a conventional air distribution system. This system is designed to minimise the risk of draught and of course another benefit is less ducting and consequently a lower pressure drop. In diffuse ceiling ventilation, the entire space between false ceiling and real ceiling is used as a plenum for the air distribution system [4]. The pressure in the entire plenum must be uniform in order to get the benefits from the diffuse ceiling ventilation system. The use of Troldtekt low pressure drop diffusers (acoustic panels) in a diffuse ceiling ventilation system will results in a low-pressure-drop air distribution system and a minimum risk of draught.

Troldtekt has installed the perforated acoustic panels in one of their offices. The company has performed some experiments to find out the relation between pressure drop and superficial velocity through the perforated acoustic panels. The superficial velocity is the net airflow rate per unit cross-sectional area of the perforated medium. The findings of Troldtekt are presented in Figure 1. In order to verify the empirical findings of Troldtekt, the CFD techniques were performed by the researchers at the Danish Building Research Institute – SBi – Aalborg University. The procedure and findings of the CFD simulations are explained in the remaining text of this report.

Aim of the study and methodology

The aim of this study was to verify the empirical results of pressure drop through perforated acoustic panels that were performed by Troldtekt under real conditions. Computational fluid dynamics (CFD) was used to verify the experimental results. A commercial CFD package was used to perform the simulations. This CFD analysis was not performed to find out the proper location of the diffusers nor to find out the air distribution in the room.

[1] Note: Comma is used as a decimal separator in this report (except in the CFD drawings)

CFD Model

The Troldtekt acoustic panels were physically installed in a 40 m long and 10 m wide room. The height of the room from floor to the false ceiling was 3,8 m. The thickness of the false ceiling (acoustic panels) was 25 mm and the distance between the false ceiling and the real ceiling was 20 cm. The entire false ceiling, except four panels, was made up of normal acoustic panels. The remaining four were the perforated ones. These four perforated acoustic panels could be used as ventilation diffusers. The air entered the false ceiling space through a circular duct opening called “Inlet”. The Inlet of the airflow was 18 cm in diameter and was placed over the false ceiling and was in the middle of a 10 m wide side wall. The entire false ceiling behaved like plenum i.e. diffuse ceiling ventilation. Four perforated acoustic panels were used as airflow diffusers. These four diffusers were termed the “Outlets”. The air dropped into the inhabited space through these Outlets and exhausted through an exhaust opening 20 cm in diameter. The exhaust opening was called “Exhaust”. The centre of the Exhaust was 80 cm below the false ceiling and was in the middle of the wall. Both Inlet and Exhaust were on the same wall. There were two structural beams (25 cm x 10 cm) in the real building. The first beam was at a distance of 5,0 m and the second was at a distance of 10,0 m from the Inlet.

Due to symmetry in geometry of the real space, only a half of the room was simulated in CFD. Figure 2 is a 3 dimensional model diagram and the corresponding boundary conditions of the CFD domain. The blue wall is the symmetric wall – it means that in reality the domain is the sum of domain shown in Figure 2 plus the mirror image of the same across the symmetric wall. In reality, the Inlet was located in the 10 m wide wall; however only 5 m was simulated with symmetric boundary conditions in the middle. Therefore, the inlet in the enlarged diagram is shown as a half circle – likewise the Exhaust. The Inlet was defined as a velocity inlet whereas Exhaust was defined as a pressure outlet. Outlets were the interfaces between ceiling plenum and the room space. All other boundaries were with wall boundary condition, no slip and adiabatic condition. The length of the real house was 40 m; however in CFD only 20 m was simulated. The idea was to verify the performance of the diffusers in the similar buildings, which is why only 20 m was simulated to reduce the computational time. The two Outlets are also shown in the enlarged views of Figure 2. Of course remaining two diffusers are omitted due to symmetry.

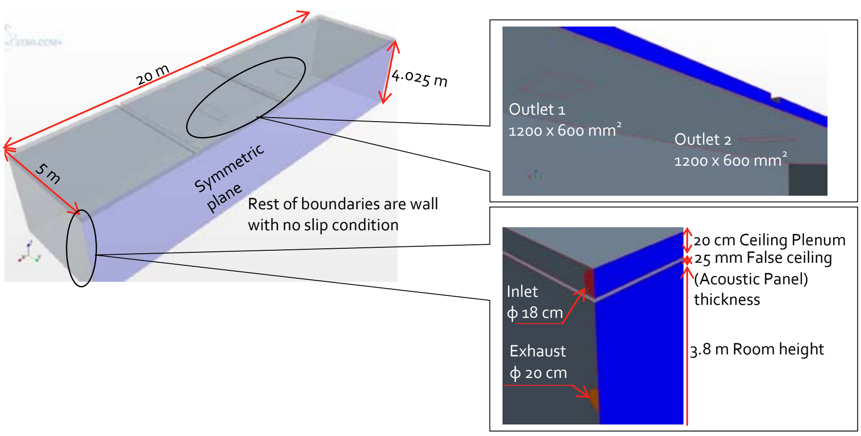

Figure 2 Three dimensional CFD Domain

Meshing

To obtain the required results the CFD domain had to be meshed into very small volumes so that the computational error could be minimised. The small volumes are called cells. Therefore, the CFD domain shown in Figure 2 was meshed. The volume is meshed in according to the velocity and pressure gradients. Areas with higher gradients are meshed with very small cell volumes whereas areas with low gradients are meshed with relatively larger cell volumes. Inlet, False ceiling, Beams, Outlets and Exhaust were more critical for higher velocity gradients compare to the rest of the domain. Therefore, the meshing size was different for different parts of the domain. Likewise, in some areas a boundary layer was critical and in some areas it was not. Therefore, the effect of the boundary layer was not considered where it was not required. To mesh the entire domain into a number of cells a base mesh side was defined. For this simulation, base meshing size was 1 m. Surface mesh (for boundaries) and volume mesh (for inner volume) sizes were defined according to the base mesh side. A commercial CFD package was used for meshing. The CFD package worked in a way that first the surface should be meshed and then a growth rate had to be defined in order to generate the volume mesh. The rate at which the size of volume mesh should increase after the surface was called the growth rate. The growth rate is necessary for the stability of the solution. The entire volume was meshed according to the surface mesh. For this simulation the growth rate was 1,2. Furthermore, where the boundary layer was important, a specific volume mesh had to be defined near by the boundary – in the used software it was called prism layer. All mesh sizes were finalised after several trials of the Outlets velocities and pressure drops and convergence for different mesh sizes at different positions and with and without prism layer meshes.

There were several meshing techniques available in the CFD software. The domain was checked for almost all type of available meshes. Each mesh was checked for the Outlets velocities and pressure drops. Tetrahedral meshes gave the same results as polyhedral meshes. However, the time required for simulation with tetrahedral was almost two thirds of the simulation time with polyhedral. That is why tetrahedral meshes were used for several Inlet velocities. Figure 3 illustrates the tetrahedral meshes in the 3D CFD domain. Likewise, the small meshes near Inlet, Exhaust and Outlets are also shown in the enlarged views of Figure 3.

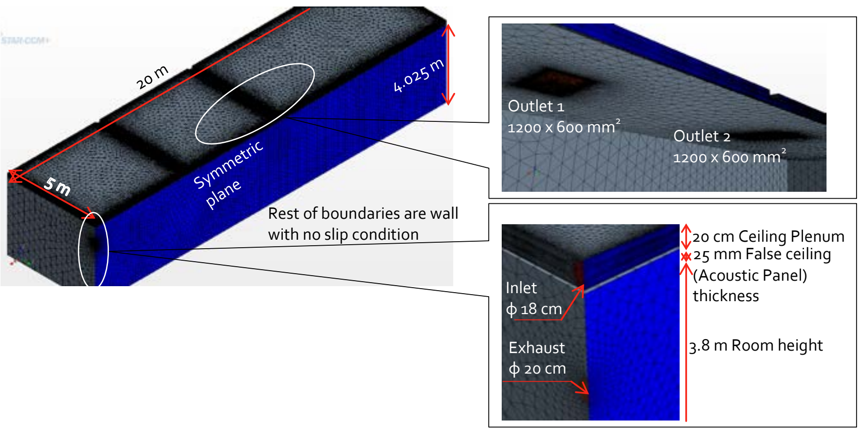

Figure 3 Tetrahedral meshes on three dimensional CFD Domain

Physics

Once the stage of meshing was finalised, the proper physics model had to be defined. To find out, whether the flow was turbulent or laminar flow, the Reynolds number at the inlet was calculated.

$$\mathrm{Re} = \frac{D \cdot V}{\upsilon} = \frac{0{,}18 \cdot 0{,}25}{1{,}521 \cdot 10^{-5}} = 2{,}96 \cdot 10^{3}$$

Where, *Re* is the Reynolds number, *D* is the characteristic length (in this case – Diameter of inlet) in m, *V* is the velocity (inlet velocity) in m/s, and ν is the kinematic viscosity (of air) in m^2/s.

This *Re* shows that the inlet flow (flow in the pipe) is turbulent flow even for the minimum velocity (used in this CFD simulation), i.e. 0,25 m/s. Therefore, a turbulent flow CFD model was required for the simulation. Several turbulent flow models were available in the software. Simulations for the Large Eddy or the DNS models were not possible due to the limited availability of the computational power. Therefore, Reynolds-Averaged Navier- Stokes (RANS) was selected. Among several RANS models the Realizable k-ε turbulent flow model predicts the flow inside the building in a reasonably accurate manner with comparatively less time [5]. However, it should be noted that the Realizable k-ε turbulent flow model predicts only fully developed turbulent flow. Therefore the solution from this model is only valid where the flow is fully developed turbulent flow. It is presumed that the flow through the Outlets is fully developed turbulent flow. Hence, the Realizable k-ε turbulent flow model was chosen to perform the CFD simulation. Incompressible ideal gas was selected as the working fluid. Simulations were three dimensional steady state. Energy and momentum were calculated in segregation. Two layer y+ treatment in the Realizable k-ε turbulent flow model were performed during simulation. Initial conditions of k and ε were calculated according to Nielsen's [6] recommendations.

The ceiling plenum and room were considered as two different fluid regions. The perforated acoustic panels (Outlets) were the interface between these two regions. Therefore a model was adapted that defines the flow through the Outlets. This model is called porous baffle interface. Hence the Outlets were the interface between two regions with Porous Baffle Interface type. Physically, a porous baffle represents a membrane through which fluid passes and undergoes a pressure drop. The heat and mass transfer through the porous baffle interface depends on the porosity and pressure drop [Star ccm guide]. The value of porosity was given by the manufacturer and it was 20%. The pressure drop through the interface was calculated according to the modelled Darcy's formula (it is assumed that the direction of flow is unchanged as it passes through the baffle):

$$\Delta P = -\rho(\alpha |V_n| + \beta) V_n$$

Where, ΔP is the pressure drop across the Outlet in Pa, ρ is the density in kg/m^3, α is the inertial resistance factor and it is unit less, V_n is the superficial velocity in m/s i.e. (total volume flow/ cross-sectional area), β is the viscous resistance factor in m/s. Both α and β were calculated from the amended diagram of Figure 1 and the least square regression of the data as shown in Figure 4.

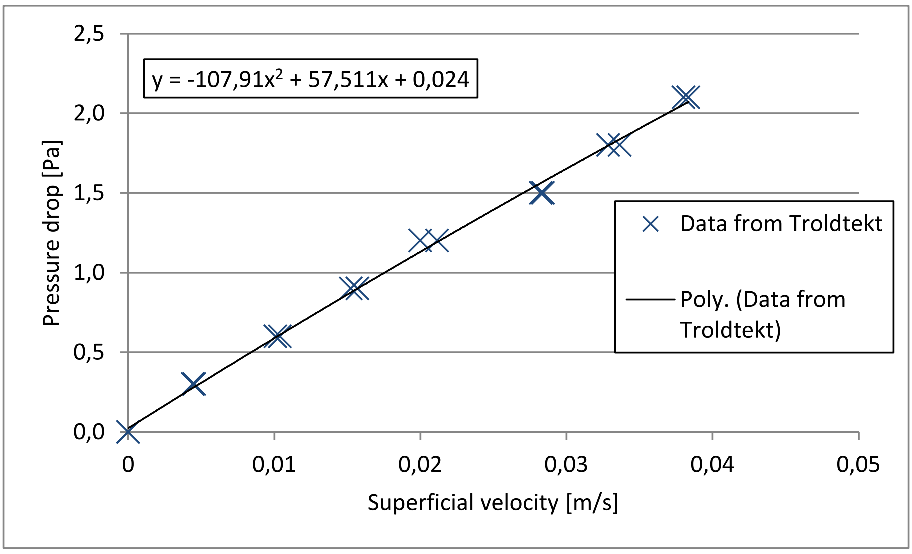

Figure 4 Pressure versus superficial velocity

$$\Delta P \approx -107{,}91V_n^2 + 57{,}51V_n$$

$$\Delta P = -1{,}2 \cdot (89{,}925|V_n| + 47{,}925)(-V_n)$$

Therefore; α = 89,925 {--} and β = 47,925 {m/s}.

From known porosity, inertial resistance and viscous resistance, the flow through the Outlets (interfaces) was calculated.

Results

CFD simulations were performed in order to find out the pressure drop across the Outlets. The simulations were performed for several inlet velocities. However, snapshots are taken only for 0,25 m/s inlet velocity.

Pressure distributions at different planes are shown in Figure 5 and Figure 6.

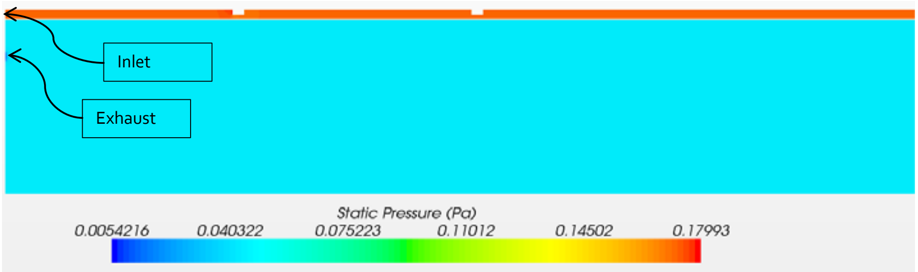

Figure 5 Static pressure distribution at the symmetric plane

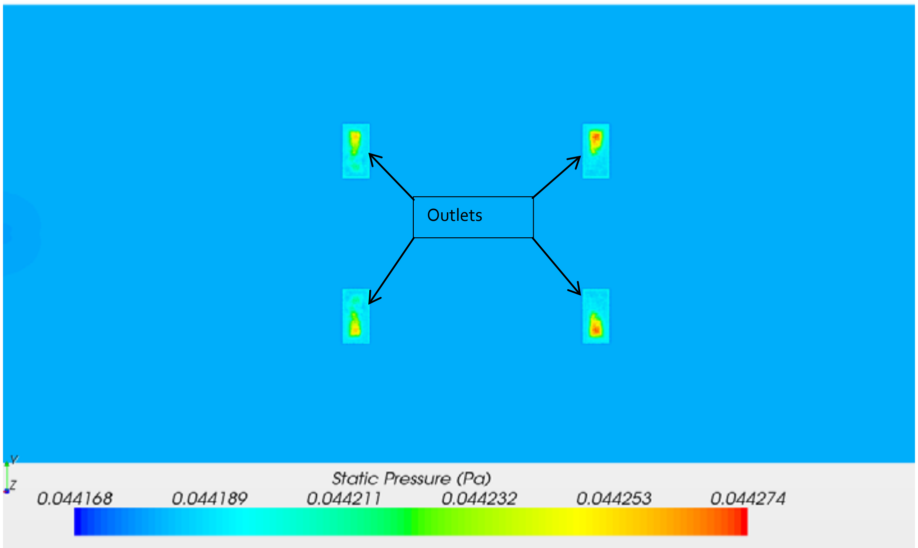

Figure 6 Static pressure distribution of at the false ceiling of the room

In Figure 5 the static pressure is almost uniform in the room and in the false ceiling space. As mentioned earlier, two Outlets were simulated, however; four Outlets are shown in Figure 6 due to symmetric boundary conditions. In Figure 6 the pressure is distributed according to the flow in the Outlets. Velocity distributions at different planes and sections are shown in Figure 7 to Figure 10.

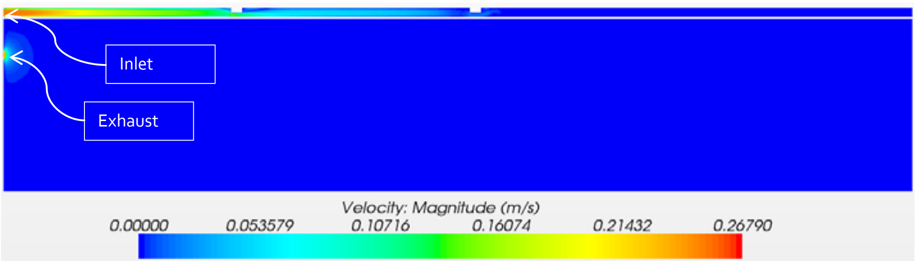

Figure 7 Velocity distribution at the symmetric plane

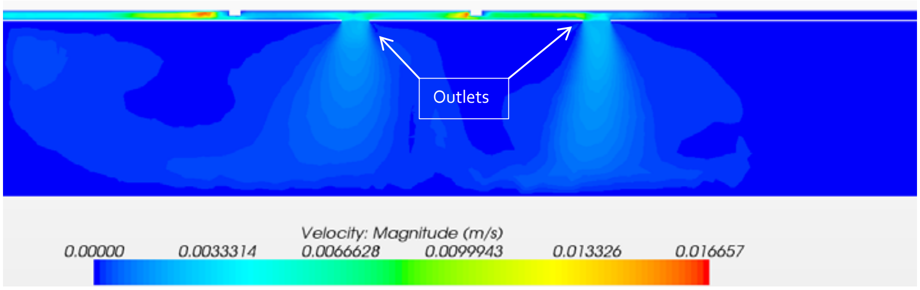

Figure 8 Velocity distribution at vertical section through centre of the Outlets

In Figure 7 velocities at the Inlet and Exhaust are according to the given boundary conditions. Figure 8 is a vertical section cut through the mid of the Outlets. Figure 8 shows the air coming out of the diffusers and then being diffused into the space. In Figure 9, four Outlets are shown with velocity distribution on at the level of false ceiling height of the room. Since the boundary condition of a false ceiling is a wall with no slip condition, the velocity at the false ceiling is therefore zero (Figure 9). In Figure 9 the velocity distribution in the Outlets are according to the given porosity, inertial resistance and viscous resistance. Figure 10 shows a section between the false ceiling and the real ceiling. It shows that the jet from the Inlet is disturbed by the structural beams. It may minimise the velocity head of the airflow.

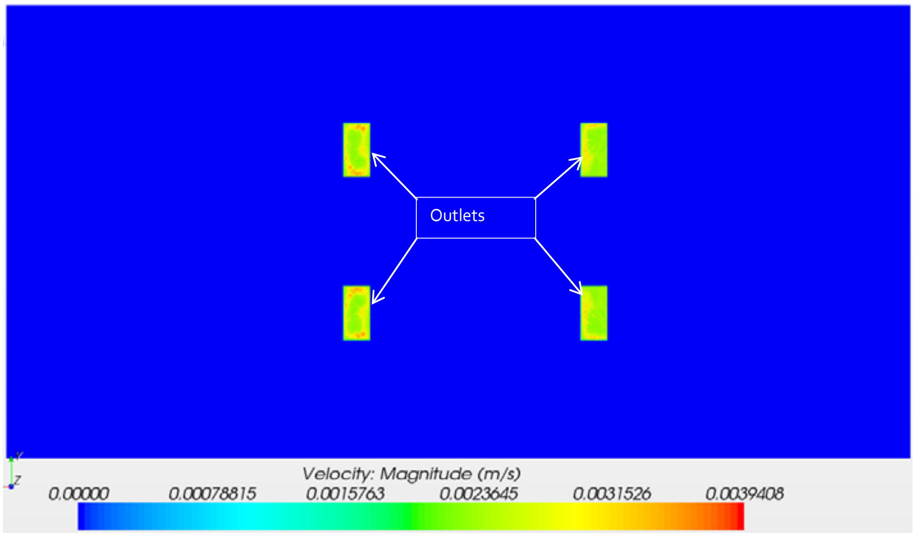

Figure 9 Velocity distribution at the false ceiling level of the room

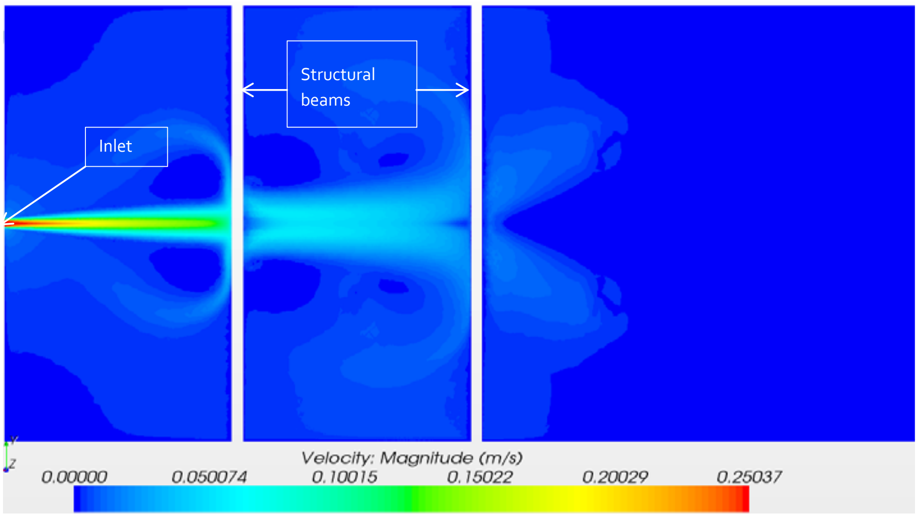

Figure 10 Velocity distribution at the section between false ceiling and real ceiling

From CFD simulations the following results are obtained:

Tabel 1 Pressure drop and airflow rate through the Outlets

Inlet-Vel {m/s}	Inflow {m^3/s}	Pressure {Pa}		Pressure {Pa}		ΔP {Pa}	Outflow {m^3/s}	Error
0,25	0,00318	Plenum: Outlet1	0,1717	Room: Outlet1	0,044	0,1275	-0,001566	1,55 %
		Plenum: Outlet2	0,1719	Room: Outlet2	0,044	0,1277	-0,001565	
0,75	0,00954	Plenum: Outlet1	0,7855	Room: Outlet1	0,400	0,3846	-0,004781	- 0,86 %
		Plenum: Outlet2	0,7887	Room: Outlet2	0,400	0,3878	-0,004842	
1,5	0,01909	Plenum: Outlet1	2,3810	Room: Outlet1	1,607	0,7739	-0,009394	- 0,09 %
		Plenum: Outlet2	2,3971	Room: Outlet2	1,607	0,7900	-0,009707	
2	0,02545	Plenum: Outlet1	3,9039	Room: Outlet1	2,856	1,0477	-0,012624	0,53 %
		Plenum: Outlet2	3,9104	Room: Outlet2	2,856	1,0540	-0,012687	
3	0,03817	Plenum: Outlet1	8,0264	Room: Outlet1	6,432	1,5936	-0,018988	0,25 %
		Plenum: Outlet2	8,0421	Room: Outlet2	6,432	1,6092	-0,019087	

Where, InletVel is the inlet velocity in m/s and Inflow is the inlet airflow rate in m3/s. Plenum: Outlet1 and Plenum: Outlet2 are the surfaces of perforated acoustic panels (Outlets) at the ceiling plenum side. Room:Outlet1 and Room:Outlet2 are the surfaces of perforated acoustic panels (Outlets) from the room ceiling side. Pressure is in Pascal. ΔP is the pressure difference between Plenum Outlet surface and Room Outlet surface. Outflow is the air-flow rate through the Outlets in m^3/s, negative values show that the direction is downward. Error is the error in mass conservation between Inflow and Flow.

Results of Outlet1 and Outlet2 were compared with the experimental results of Troldtekt. The comparisons are shown in Figure 11 and Figure 12. The crosses Troldtekt 1 and Troldtekt 2 are experimental results of two Troldtekt acoustic panels whereas, the blue line is the outcome of the CFD simulation.

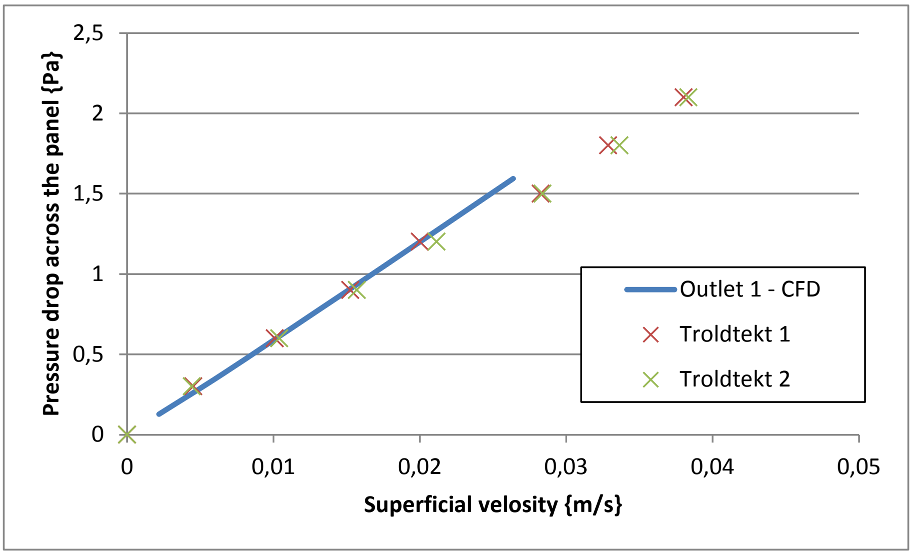

Figure 11 Pressure versus superficial velocity of Outlet 1 and comparison with the experimental results

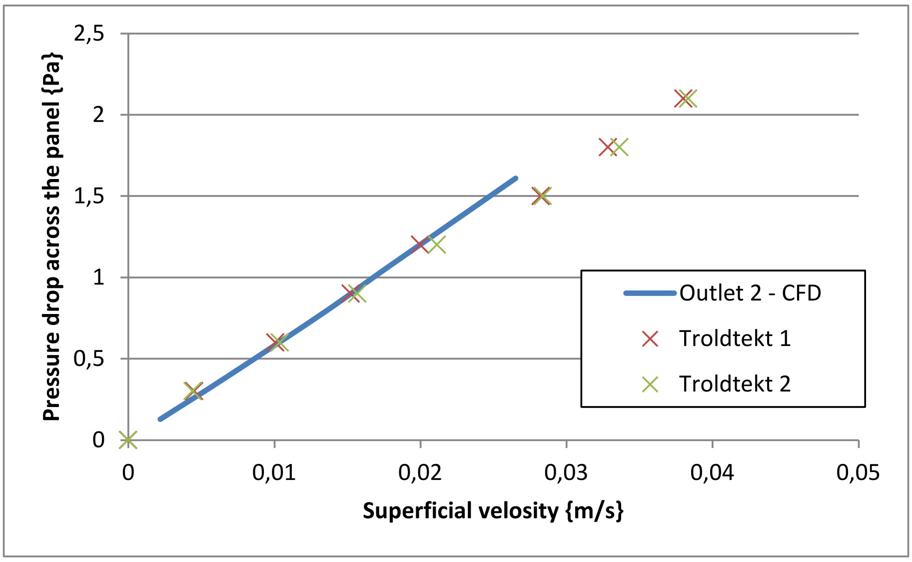

Figure 12 Pressure versus superficial velocity of Outlet 2 and comparison with the experimental results

Discussion

The main idea of this CFD analysis was to verify the experimental results. Therefore, only part of the examined room was simulated in CFD by using symmetric boundary conditions. However, sometimes air jet deflects to from its centre position therefore, in designing of the DSV with Troldtekt diffusers use of symmetry boundary condition is not advisable. Furthermore, there was no temperature difference between the room and the ceiling. Hence, the effect of temperature stratification on the performance was not analysed. The pressure difference was only due to the Inlet airflow rate.

Meshing was verified by conservation of momentum, energy and Outlet pressures on both sides for different sizes and types of meshes. Mesh size was decided where the effect of mesh size on pressure distribution on each side of the Outlets was as minimum as 10^{-4}. Therefore, it is believed that at least the solution for pressure drop and superficial velocity across the Outlets are grid independent.

The airflow at the inlet was fully developed turbulent flow even for minimum inlet velocity used in this simulation. Therefore, the Realizable k-ε turbulent flow model was used in accordance with the recommendations in previous scientific literature. For estimation of the airflow rate through the Outlets (i.e. perforated acoustic panels), the modelled Darcy's formula was used. The coefficient of the Darcy's formula was extracted from the experimental results. Hence mainly this simulation is about the use of the Realizable k-ε turbulent flow model along with the modelled Darcy's formula was used to define the diffuse ceiling ventilation with perforated acoustic panels for ventilation system.

The results of pressure distribution showed the expected results that the pressure in the ceiling is higher than in the room. If there is a heating medium inside the room, then the situation will not necessarily be the same. Therefore, there is a need of analysing the DCV with Troldtekt diffusers with heating source inside and without consideration of symmetry. Likewise, the situation has to be analysed when the airflow coming into the system is of higher temperature. Different numbers of diffusers at different places have to be examined for estimation of the minimum required number of diffusers for DCV.

Different inlet air velocities were simulated in CFD. Then the results were compared with the experimental results from Troldtekt. The results were in very good agreement with the experimental results. The deviation of CFD results from the experimental results is less than 1% from 0 to 0,03m/s (0-0,03) of the superficial velocity of air through the perforated acoustic panels. However, the deviation is up to 15% after 0,03 m/s of superficial velocity of air.

Conclusion

In this study, experimental results are compared with the numerical results. The CFD simulation shows promising results. The maximum deviation of pressure drop calculation through CFD is below 15% of the experimental results. Therefore, CFD techniques (using the Realizable k-ε turbulent flow model) can be used to design a ventilation system with perforated acoustic panels as diffusers, especially with the diffuse ceiling ventilation system.

Use of the amended Darcy's formula to define the acoustic panel shows that the simple model can be used to define the flow through perforated acoustic panels in a very accurate manner. By using simplified formulae there is a tremendous saving of time. Likewise, the Realizable k-ε turbulent flow model (RANS equations) predicts the airflow rate and corresponding pressure drop through the perforated acoustic panels with a 15% accuracy. However, the accuracy at lower airflow rates is as low as 1%.

For future work, there is a need to define a simplified formula for the minimum requirement for perforated acoustic panels to ventilate a specified volume of an inhabited space. Likewise, the behaviour of acoustic panels in a thermally stratified space should also be investigated.

References

[1] B. R. Munson, D. Young and T. Okiishi, *Fundamentals of Fluid Mechanics.* John Wiley & Sons, 1999.

[2] Anonymous *ASHRAE Handbook HVAC Systems and Equipment.* ASHRAE, 2012.

[3] P. V. Nielsen, R. L. Jensen and L. Rong, "Diffuse ceiling inlet systems and the room air distribution," 2010.

[4] P. Jacobs and B. Knoll, "Diffuse ceiling ventilation for fresh classrooms," in *30th AIVC Conference "Trends in High Performance Buildings, Berlin, Germany,* 2009, .

[5] Anonymous "- Computational Fluid Dynamics in Ventilation Design REHVA Guidebook No 10," *- International Journal of Ventilation,* pp. - 291.

[6] M. Skovgaard and P. Nielsen, "Simulation of simple test case, case 2D1,(two-dimensional isothermal forced convection);annex 20: Airflow patterns within buildings[http://Cfd-benchmarks.com/]," International energy agency; Energy conservation in buildings and community systems, Tech. Rep. ISSN: 0902-7513 R9131, 1990.

[7] J. Fan, C. A. Hviid and H. Yang, "Performance analysis of a new design of office diffuse ceiling ventilation system," *Energy Build.,* vol. 59, pp. 73-81, 2013.

[8] C. A. Hviid and S. Svendsen, "Experimental study of perforated suspended ceilings as diffuse ventilation air inlets," *Energy Build,* 2012.

[9] Y. Li and P. V. Nielsen, "CFD and ventilation research," *Indoor Air,* vol. 21, pp. 442-453, 2011.

[10] P. V. Nielsen and E. Jakubowska, "The performance of diffuse ceiling inlet and other room air distribution systems," 2009.

Ventilation is needed for diluting and removing the contaminants, odour and excess heat from the building interior. It is important that the inhabitants perceive the ventilated spaces as comfortable. Therefore, the supply air should reach all parts of the occupied zones. Troldtekt has been manufacturing perforated acoustic panels for the last 13 years. The panels can be used not only in applications related to acoustics but also as low pressure drop supply air diffusers, particularly in diffuse ceiling ventilation systems. The present study verifies on a theoretically level the performance of the panels and empirical findings are examined through the use of CFD, computational fluid dynamics.

1st edition, 2013
ISBN 978-87-92739-33-9