Higher turbulent mixing due to UnoDuct - air supply system

Christian Heschl, Andrea Kelz, Fachhochschulstudiengänge Burgenland GmbH

Abstract

Previous investigations of free jet propagation are usually limited to axialsymmetrical and qasi two-dimensional jets. Multiple jets in the form of nozzle arrays or slot inlets have hardly been subject of systematic investigation. In this article the turbulent mixing processes occuring in this case are discussed with the aid of Computational Fluid Dynamics. It is shown that velocity and temperature reduction as well as the turbulent mixing process are dependent on the distance between the air supplies.

Introduction

The main task of an HVAC system is to provide the required room air quality. To fulfil this requirement suitable air supply systems already have to be selected and dimensioned in the planning stage.

If a mechanical ventilation system is to be used, a displacement or a mixing ventilation system are available. In the majority of cases the mixing concept is chosen, which means that the air is supplied with high momentum. In order to satisfy the user requirements the air supplies have to ensure quick reduction of velocity and temperature between the inlet air and ambient air. Therefore a high turbulent mixing process with the ambient air must be guaranteed.

Experimental and numerical investigations show that the exchange processes are essentially affected by the turbulence level and the inlet flow velocity [1, 2, 3]. Room dependent influence variables such as room geometry, natural convection and temperature stratification are known [4, 5]. In order to create best conditions for the user the producer not only has to optimize the turbulent mixing process but also to provide calculation tools for appropriate utilization of room conditioned influences to designers of ventilation systems.

Experienced ventilation technicians try to use special flow phenomena such as the Coanda effect to improve the flow situation in ventilated rooms. The Coanda effect is based on the fact that air is entrained at the jet boundary due to turbulent mixing processes. If not enough air can be entrained, e.g. on account of ceiling, underpressure arises and the jet is diverted on the ceiling. This effect can be forced by obstructing the lateral entrains of air through long slot inlets. For this reason slot and jet lines are increasingly used. In spite of using the Coanda effect it often comes to a cold snap in cooling cases. If this happens in the inhabited zone, it can lead to draught and thermal uncomfortableness. This undesired effect is produced by the Archimedes principle and is dependent on the temperature difference between the ambient air and the jet core. For this reason ventilation technicians strive to reduce the temperature difference as quickly as possible.

In this article the turbulence parameters of jet line inlets and slot inlets are analysed and their effect on velocity and temperature distribution are discussed. In order to make any periodic interruption of the jet line inlets possible the investigations are based on the UnoDuct-system of Lufttechnik J. Pichler GmbH. The investigations of the slot inlet system are based on a conventional supply system.

Examination method

The classical experiments as well as the numerical experiments, e.g. Computational Fluid Dynamics – CFD, can be used to get the necessary parameters. Although both methods have different expressiveness they are closely related and complement each other. The advantage of the numeric experiment to get all information about substantial physical values, however, is gained at the expense of general validity. The quality of the results depends on the physical model and the used numerical methods. Therefore it is necessary to validate the numerical results with experimental findings.

In order to meet these demands extensive validations were made [6, 7]. For the calculations the commercial CFD code FLUENT 5.5 was used [9].

Analyzed air supply systems

Two air inlet systems which allow air supply over the whole room width are analyzed. The essential flow parameters of the slot inlet and the UnoDuct inlet system are evaluated and interpreted.

With the UNO-DUCT inlet system the supply air is transported by a so-called "nozzle duct". This duct consists of a spiral-fold-duct with form-closed nozzle elements. The layout of the nozzle elements can be arranged any way you want. On grounds of stability a minimum distance between the nozzle elements in axial direction ($s_d \geq 89mm$) and tangential direction ($s_t \geq 30mm$) must be kept. Every nozzle element consists of seven bores with a diameter of approximately 8.6 mm. In this way you get a modular system which allows a high number of combinations which can be easily adapted to local demands.

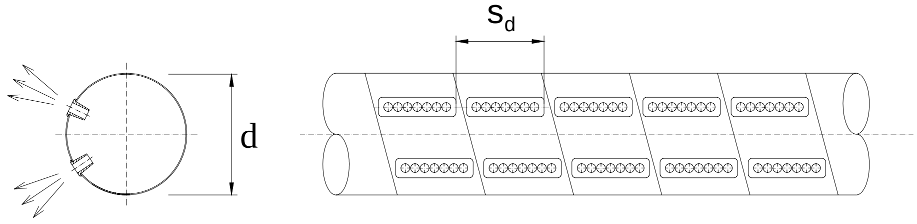

Figure 1: Detail – nozzle element and air supply system UnoDuct

The simultaneous use of the spiral-fold-duct as girder and supply system makes the distribution of supply air in all directions (diffuse) possible. The generated small single jets increase the whole entrained secondary air. For this reason the flow velocity and the temperature difference are reduced considerably faster.

The investigation referred to in this article is limited to in one row assembled nozzle elements. According to the distance s_d a change of entrainment and reduction of temperature and flow velocity are expected. In order to investigate the effect of single jets on jet propagation two simulation models with different inlet boundary conditions are generated.

Simulation models

The simulation results essentially depend on the inlet boundary conditions, i.e. on the flow parameters of the respective air supply. For this reason separate numerical and experimental investigations are made to determine the inlet boundary conditions for the UNO-DUCT nozzle system.

It can be shown that the simulated time averaged outflow velocity, the outflow direction and the turbulence intensity agree well with the experimental data. Besides, as a result of air deflection, a double swirl (similar to air passing a pipe bend) was observed. To demonstrate these findings, figure 2 shows the contour plot in the inlet plane of the velocity components.

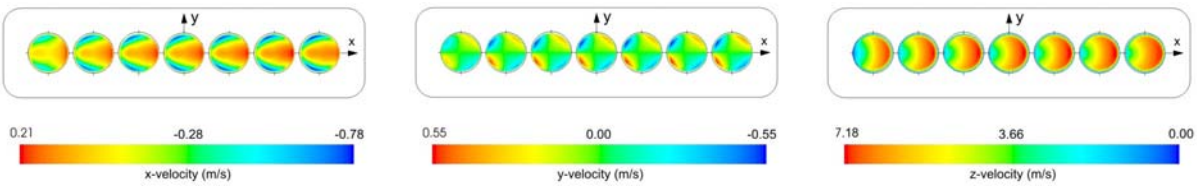

Figure 2: Contour plot of the double swirl - Uno-Duct supply system (p_{stat}=20Pa, u_{duct}=4m/s)

From these autarkic numerical investigations the necessary inlet boundary conditions were generated. On account of the good agreement between measured and simulated data the determination of the boundary condition for the slot inlet system was also generated via numerical investigations. Therefore the fulfillment of the model equations is ensured.

Figure 3 shows the design of the used simulation model. The inlet consisted of eleven nozzle elements with seven bores each and a slot with a width of $b = 520mm$. The diameters of the bores were $d = 8.6mm$ and the slot height was $h = 8.6mm$. The proportions of the simulation domain were defined with $L \times B \times H = 3.6 \times 0.979 \times 2.5m^3$.

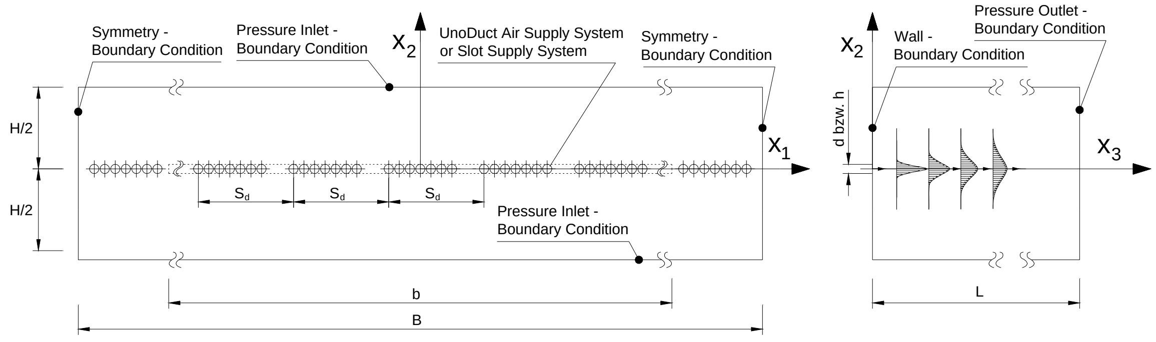

Figure 3: Boundary conditions of the simulation models

The chosen inlet geometry definitions ensure identical fluxes, i. e. both air supply systems have the same mass, momentum and energy fluxes in the inlet plane. Thus the essential requirements for an objective comparison of the system are met.

Discussion of the results

The characteristic values are presented non-dimensional in order to make the results easier to access for general interpretation. On account of the three dimensional flow the $x_1 = 0$ and $x_2 = 0$ plane (cf. figure 3) are evaluated. As calculation basis a nozzle element spacing of $s_d = 89mm$ and a slot spacing of $B = 979mm$ was used.

The outward jet entrains the ambient air from top, bottom and side direction. On account of the chosen model setup air can not entrain across the symmetry planes. For this reason a quasi plane jet is developed downstream. The speed of transition from a multi jet to a single plane jet depends on the spacing between the supply air elements and the turbulent mixing process.

Figure 4 and 5 shows the non-dimensional time averaged velocity and temperature in the $x_2 = 0$ plane.

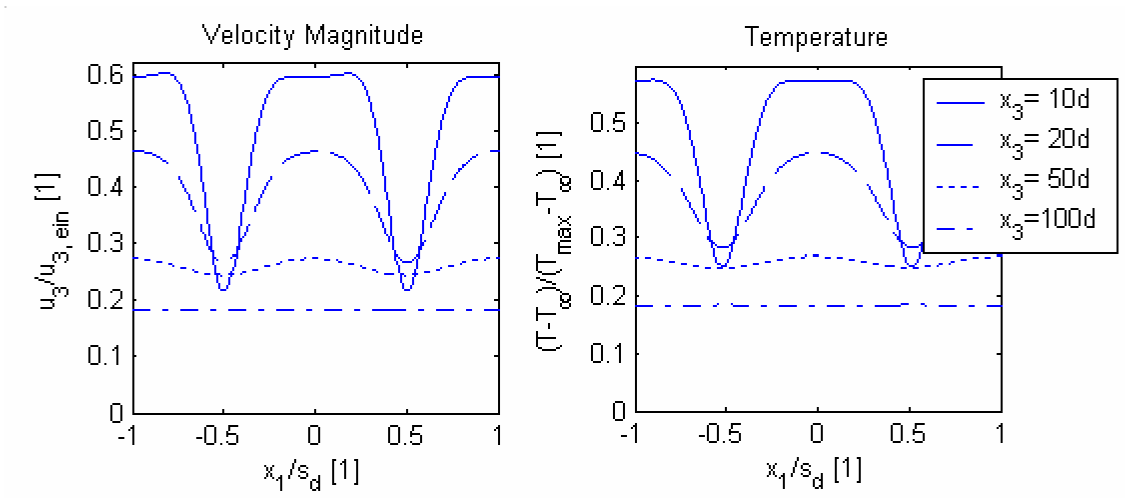

Figure 4: averaged velocity and temperature – air supply system UnoDuct

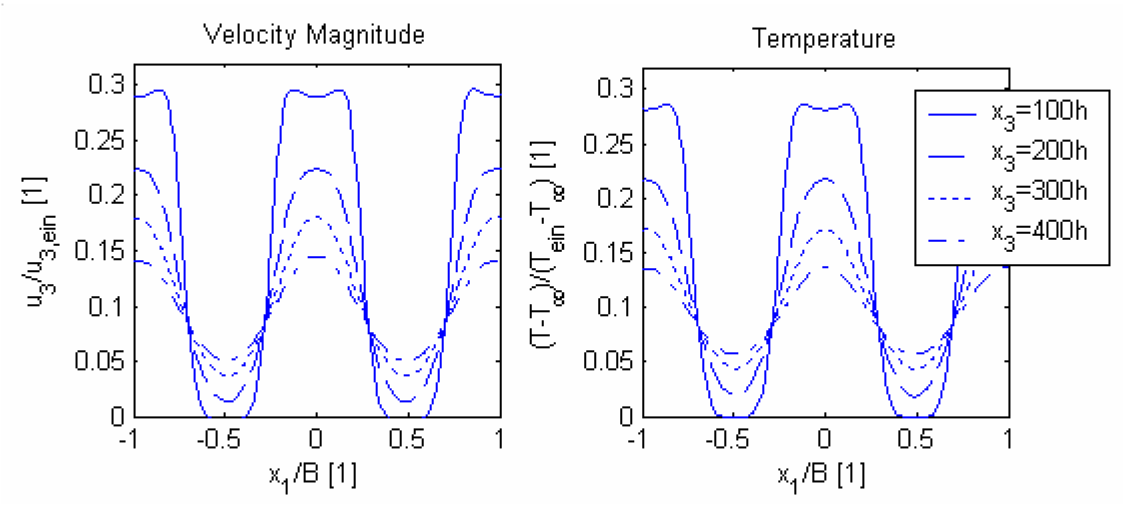

Figure 5: averaged temperature and temperature – air supply system UnoDuct

Within a length of $x_3 < 100d$ the velocity gradients in Figure 4 can be clearly detected. They generate additional, turbulent momentum and heat transfer cross to the main flow direction.

With the slot air supply system turbulent transverse motions inside the $x_2 = 0$ plane are also detected. On account of the big spacing between the inlet elements, the velocity peaks are reduced more slowly. This behavior at the transition between the multi jet and the single jet was expected.

In order to be able to analyze the global effects of the different turbulent transport processes figure 6 shows the ratio between volume flow at the inlet and the volume flow which is entrained within the jet.

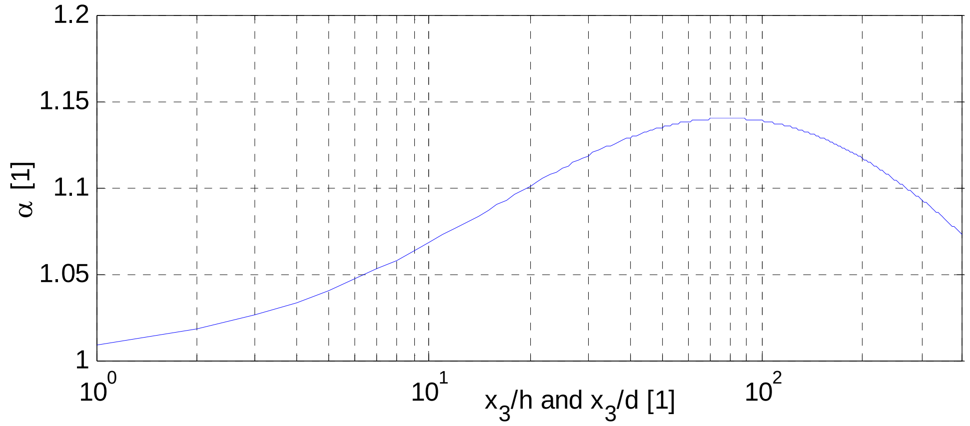

Figure 6: ratio entrained volume flow

The parameter α is defined as follows

$$\alpha = \frac{\dot{V}_{UnoDuct}(x_3)}{\dot{V}_{Schlitzauslass}(x_3)}. \tag{7}$$

It can be clearly proved that in the area of interest (jet length approx. 100 to 400d) up to 15% more ambient air is entrained by the UnoDuct air supply system.

With bigger distances the ratio α gets smaller and approaches asymptotically the value 1. This can be explained by the fact that after a specific jet length a quasi plane jet is developed by the slot air supply system.

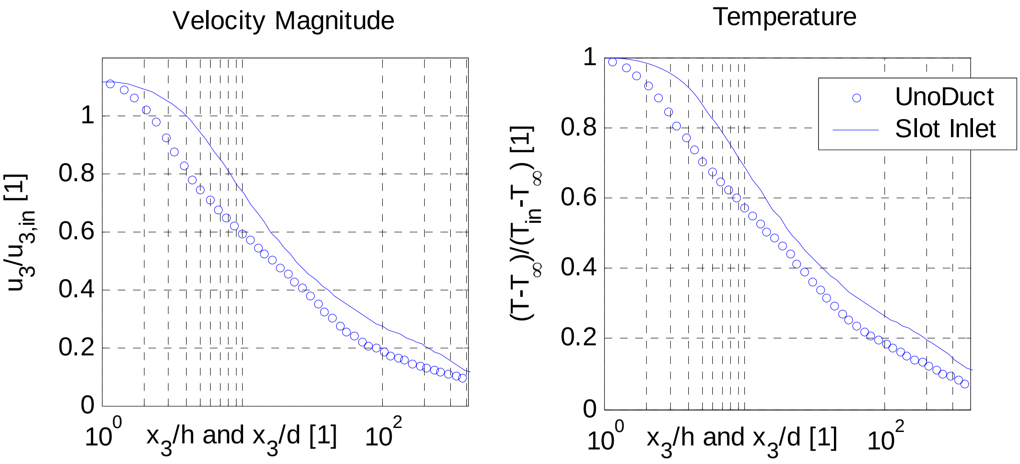

Figure 7: averaged velocity and temperature on the mean jet centerline

Due to the higher entrainment of ambient air and the faster formation of a quasi plane jet, a faster velocity and temperature reduction are allowed. According to figure 7 an about 25 % smaller value for the averaged velocity and temperature on the jet centerline can be expected inside a jet length from 100 to 400d.

Summary

Air supply methods have an essential impact on the thermal comfort of the user. In order to meet the demands of the users various air supply systems are available for the designers. In the cooling case systems are often used which ensure the advantage of the Coanda effect. Apart from slot air supply systems, in a row located nozzle elements are often used. Due to easy construction these nozzle elements can be directly integrated in spiral fold ducts.

Contrary to slot inlet systems the inlet air is supplied by means of a big number of little bores which are integrated in nozzle elements. After entry of supply air small single jets develop. The nozzle elements are arranged next to each other, leaving only tiny gaps. These geometrical conditions immediately lead to high velocity and temperature gradients whose effects were analyzed.

For the investigation numeric methods validated in other investigations were used. Comparative investigations between a conventional slot air supply system and the described UnoDuct system of Lufttechnik J. Pichler GmbH were carried out. It was assumed that the mass, momentum and energy fluxes of both systems agree. In order to eliminate the influence of room geometry, free jet boundary conditions were supposed. In addition, buoyancy effects were neglected.

Due to the smaller spacing between the inlet elements, the UnoDuct nozzle air supply system allows a considerably faster transition to the quasi plane jet than with the flux-adequate (identical mass, momentum and energy flux) multiple jet. Besides, considerably higher turbulent mixing processes inside the interesting area (jet length approx. 100 to 400d) were observed. It could be shown that faster transition to the quasi plane jet and more intensive turbulence quantities lead to better entrainment.

Under the described boundary conditions the entrained volume flow rate is up to 15% larger than with the UnoDuct nozzle air supply system. The reduction of the time averaged velocity and temperature at the jet centerline is about 25% higher. Particularly in the cooling case this facilitates the compliance with the requirements of thermal indoor air quality.

Bibliography

[1] Hanel, B., Beitrag zur Berechnung von Freistrahlen mit erhöhter Anfangsturbulenz, *Luft- und Kältetechnik,* (1977), 63-67.

[2] Hanel, B., Richter, E., Das Verhalten von Freistrahlen in verschiedenen Reynolds-Zahlenbereichen, *Luft- und Kältetechnik,* (1979), 12-17.

[3] Regenscheit, B., Einfluss der Reynoldszahl auf die Geschwindigkeitsabnahme turbulenter Freistrahlen, *HLH 27,* (1976), 122-126.

[4] Peng, S., Modelling of Turbulent Flow and Heat Transfer for Building Ventilation, *PhD-Thesis,* Chalmers University of Technology, Gothenburg, (1998).

[5] Zhang, G., Morsing, S., Bjerg, B., Sviidt, K., A Study on the characteristics of airflow in a full scale room with a slot wall inlet beneath the ceiling, *Proceedings of RoomVent'2000*, (2000), 199-204.

[6] Heschl, H., Fesharaki, M., Raumluftströmungssimulation, *Internationaler Kongress e.nova.2001,* Pinkafeld, (2001).

[7] Hanel, B., Die Berechnung der Mischungszone eines axialsymmetrischen turbulenten Freistrahls, *Luft- und Kältetechnik,* (1976), 193-197.

[8] Heschl, H., Fesharaki, M., Steinkellner M., Validation der CFD-Analyse von Raumluftströmungen mittels Messungen, *Fluent Anwenderkonferenz,* Frankenthal, (2002).

[9] FLUENT, FLUENT 5.0 User's Guide, *Fluent Inc. Centerra Resource Park,* Lebanon, (1998).

Nomenclature

b	slot width [m]
B	width of the calculation domain [m]
d	bore diameter, duct diameter [m]
p_{stat}	static pressure [Pa]
s_d	spacing between two nozzle elements [m]
t	time [s]
T	temperature [K]
T_{ein}	averaged inlet temperature [K]
$T_{\max}$	max. temperature in the observed plane [K]
T_{∞}	ambient temperature [K]
u_i	averaged velocity in the x, y and z-direction (i = 1,2 bzw. 3) [m/s]
$u_{i,ein}$	averaged inlet velocity [m/s]
$u_{i,\max}$	max. velocity in the observed plane [m/s]
x_i	geometry direction x, y and. z-direction (i = 1,2 bzw. 3) [m]
α	parameter – volumetric flow rate [1]