On the Use of Downdraft Spoilers in Glazed Atria

Bård Venås and Bent A. Børresen
Norconsult AS, Norway

ABSTRACT

The effect of downdraft spoilers is questioned. CFD results are presented for a large atrium equipped with downdraft spoilers (“obstacles”) to counteract downdraft from glazed façades. Simulations are in addition presented for one- and two-story test rooms, where aspects of the efficiency of downdraft spoilers are investigated. Among conclusions are that the spatial thermal balance of rooms, e.g. stratification or other buoyancy driven flows, are essential to the downdraft, more so, than downdraft spoilers.

1. INTRODUCTION

In countries with variable climate a number of factors should be addressed when designing thermal solutions for large glazed atria. Two of the most important during wintertime being, discomfort due to downdrafts and possible high energy use from transmission losses through the glazing. These two factors are also, evidently, closely interlinked.

To counteract downdrafts distributed heating along the interior framework of the glazed walls has been the traditionally solution. This may, however, yield high energy use - as the zones heated are those with the lowest resistance to transmission losses. This type of heating is also usually of less benefit to the thermal climate of the room, as the heat is added in no-occupancy zones.

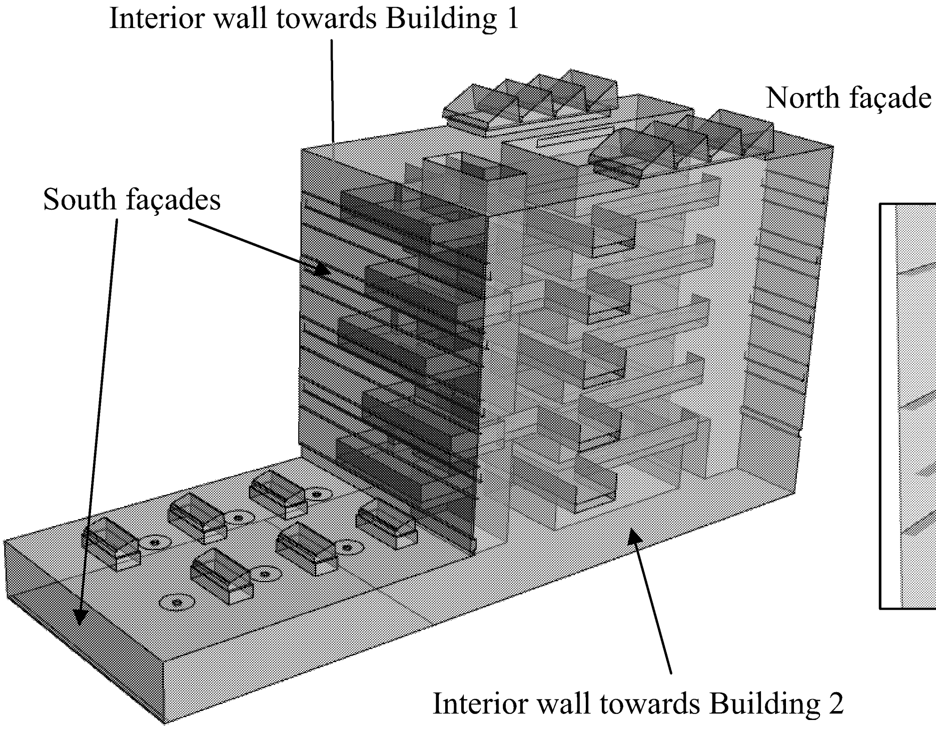
Another means for addressing downdraft problems is mounting of “*downdraft spoilers*” along the inside of the glazing, in order to break up the downwards cold convective flow. The method has been investigated in a number of papers (e.g. Heiselberg, 1994). The downdraft spoilers are often simply termed “obstacles” and come in the form of horizontal plates protruding from the glazed surfaces and into the room. The spoilers generally span the width of the glazing and have a depth, often said, that “should be larger than the depth of the convective boundary layer”. This may typically be 200-300 mm when spoilers are place one at each floor level.

The first part of this paper is based on simulations of a six story atrium, initially designed with downdraft spoilers and no façade heating. The study revealed a complex thermal climate in the large space and that the downdraft spoilers seemed to be of limited or no use. The results initiated ongoing research on flow in atria. Initial results of this study are reported in the second part of the paper, highlighting some effects that seem likely to affect the usefulness of downdraft spoilers

2. SIMULATION OF GLAZED ATRIUM

2.1 Building - original design

The six story atrium is located between two office buildings for which it serves as vestibule and restaurant. The multi-story part of the atrium has a horizontal cross section of ca. 500 m^2. In addition, on the ground floor the atrium has an open connection to a one storied ca. 500 m^2 restaurant area. The multi-storied



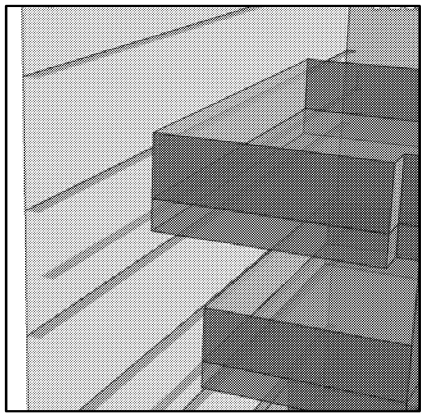


Figure 1. Geometrical model of atrium. Details of inside of façade with downdraft spoilers (right).

part further contains a central shaft for elevator, stairway, technical installations, and galleries on all floors. The atrium is shown in Figure 1. The two vertical surfaces not facing the office buildings are glazed façades, with a U-value of 1.1 W/m^2K.

In the original design the space was heated solely by floor heating on the ground floor. To counter downdrafts the façades were equipped with, as much as, three 250 mm deep downdraft spoilers for each floor. Details of these can be seen in the right part of Figure 1.

2.2 Simulations

CFD (Computational Fluid Dynamics) was used to investigate the thermal climate inside the atrium. The CFD code was ANSYS CFX 11.0, which is a state-of-the-art general CFD package. The simulations were performed with the standard k-ε turbulence model, including buoyancy terms. It was evident that the flow was time varying and the simulation was therefore run as a transient case, using 1 s time steps.

The model covered the interior air volume of the atrium. Heat conduction through glazing and walls was defined as boundary conditions, based on the outside temperature, and the conduction heat resistance of the individual walls. Thermally massive elements, i.e. ground floor surface and concrete slab galleries, were defined as adiabatic surfaces. Thermal radiation was included using Monte Carlo ray tracing, tracking 100 000 random ray histories per time step. The emissivity of the surfaces was set to 0.9. The simulations that are reported here were performed for the dimensioning ambient winter condition of -20°C.

2.3 Results with the initial design

The results for the initial design clearly showed a circulating air motion in the atrium. The overall picture consisted of updrafts near the south façade and downdrafts in sections towards the northern façade. This can be seen in Figure 2. The flow was unsteady and the figure shows an instantaneous, but representative, situation for a typical vertical cross section (i.e.

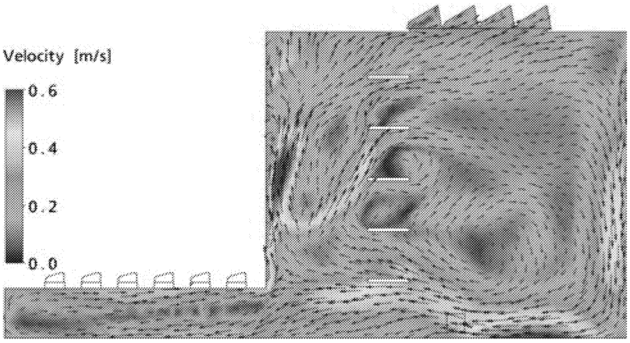


Figure 2. Initial design. Air flow in a cross section.

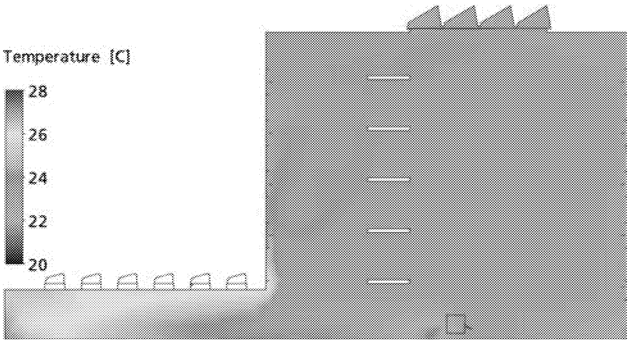


Figure 3. Initial design. Temperatures in a cross section.

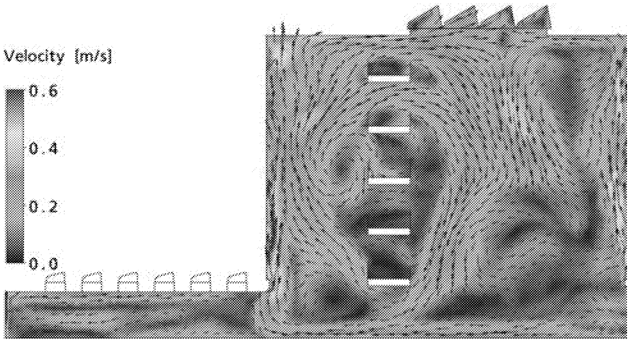


Figure 4. Altered design. Air flow in a cross section.

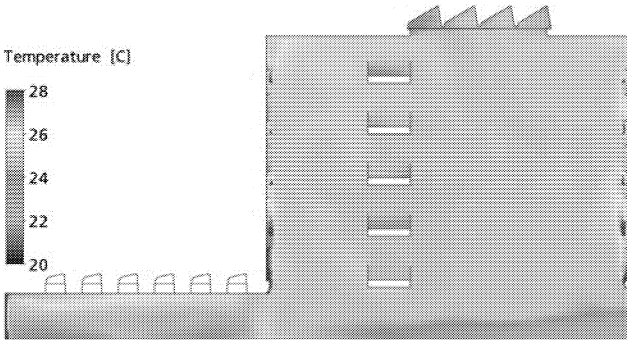


Figure 5. Altered design. Temperatures in a cross section.

“a snapshot”). The results show local velocities as high as 0.6 m/s in the vestibule area, and 0.4 m/s in the restaurant area.

Studying the temperature distribution in the same cross section; see Figure 3, it was evident what caused the unfavorable draft situation. The evenly distributed floor heating resulted in higher temperatures in the restaurant than in the atrium. Buoyancy caused warmer air from the restaurant to rise up along the south façade. The volume of air set in motion in this manner was larger than the ventilation of the room, and forced air downwards along the north façade. The air from here is deflected along the floor and again “feeds” the buoyant updrafts along the south façade - and in the restaurant. As a whole the flow can, in fact, be seen as a type of buoyancy-driven cavity flow.

The downdraft spoilers seem to have no significant effect on the draft situation, which is controlled by the overall 3D heat balance in the room, and not by local convection along the glazing.

2.4 Results with altered design

For the project to keep on schedule the atrium was redesigned, and heat was added along the façades, in the traditional “safe” manner. The atrium was finally built without floor heating in the restaurant area. Instead the heating here was provided by convectors, in the floor near the façade, balancing the local transmission losses.

Figures 4-5 show air flow and temperatures in the same vertical section as the previous figures. The air flow is still somewhat unstable, but the flow along both facades are generally in the upwards direction. Some fluctuating downward motions are seen in the centre of the room. However, velocities near the ground floor were reduced to a maximum of ca. 0.3 m/s in the vestibule, and less than 0.2 m/s in the restaurant.

It should be noted that several other changes were implemented between these to designs. One of these was that the total heating capacity was increased, and the temperature in the atrium is therefore higher in the case of the altered design. In general the temperatures can be said to be quite homogeneous in the multi-storied part of atrium, for both designs. The lowest temperatures (i.e. colored blue), for both situations, are due to colder air from the main air inlets in the vestibule area. These inlets were relocated from the initial to the altered design, and also affect the air flow in the vestibule.

3. TEST CASES

To investigate the use of downdraft spoilers, detailed simulations have been performed for a number of test cases. These studies are part of an ongoing research program and not directly linked to this specific atrium.

3.1 Simulations

All test simulations presented here were run with the SST (Shear Stress Transport) two-equation turbulence model. This is the recommend model in CFX for most cases, and is a blend between a k-Ω model near the wall, and a k-ε model further from the wall. In early tests the SST model showed much better results, compared to experiments, than the standard k-ε model (used for the atrium). This has been reported in literature to be due to the superior wall treatment of the k-Ω model, combined with the free stream robustness of the k-ε model (Esch et al., 2003).

The tests were designed to be comparable to the one-story experiment reported in Heiselberg (1994). They used a room 3 m high, with a floor 6 m in width and 7 m in depth. The same group has also performed experiments for a two-story room with downdraft spoilers. At the time of the writing of this paper we only had access to these results from a report that did not contain full documentation of these experiments (Heiselberg et al., 1998). Our two-story simulation "test room" was defined as having the same floor area, as the one story case, but twice the height, that is, 6 m.

During the experiments one surface of the room was kept at a constant low temperature. We specified the same conditions in the simulations, using the maximum wall-to-room temperature difference from the experiment, ca.

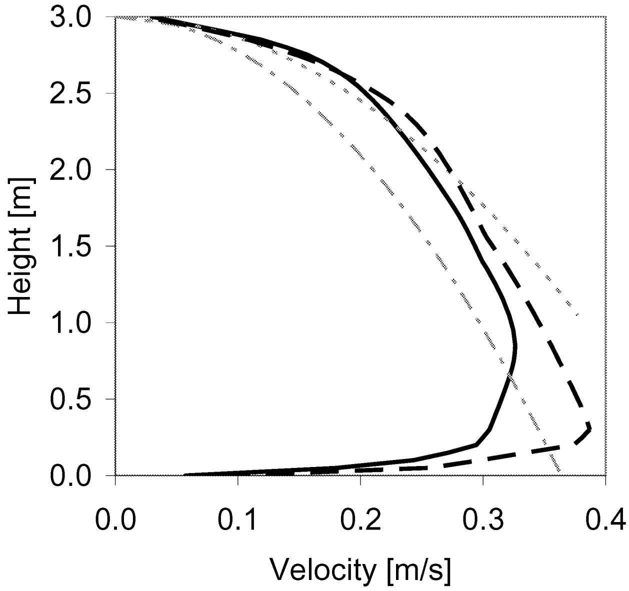


Figure 6. Max velocity along cold wall.
Simulations: closed room ━━ , open room ━ ━ ━
Empirical : turbulent ═ · ═ , laminar ═ ═

9°C (room at 20°C and wall at 11°C, i.e. resembling a window).

3.2 One story room with cold wall

In the experiments the heating of the room was obtained by heat flow through walls and ceiling, from the lab surrounding the test room, which was kept at ca. 25°C. When using similar boundary conditions in the simulation we found an increasing stratification in the room as time elapsed. To study the effect of the stratification we performed an additional simulation where the rear wall was replaced by an opening, for in and outflow, at 20°C. This can be perceived as having an infinitely large room, where stratification does not establish.

Figure 6 shows the maximum velocities close to the cold wall for the two simulations, compared to empirical correlations for turbulent and laminar flow, as found in (Heiselberg et al. 1998).

The simulations were performed with the CFX automatic wall treatment and a dense boundary mesh with $y_+ \approx 1$. This modeling has capability to reproduce the laminar flow and transition effects, although such details are arguably somewhat uncertain. We see that both simulations are close to the laminar correlation along the upper part of the wall. Further down the velocity follows the gradient of the turbulent correlation, but are shifted somewhat to higher velocities. About 1 meter above the floor the results show reduced velocities for the closed room. This is probably due to stratification in the room. The temperature is 20-21°C in the central part of the room. However, it starts to fall below about 1 m above the floor, to as low as 17°C near the floor.

The overall conclusion is that the simulations reproduce experiments quite well. This is also the case for the width of the downdraft, which is not shown here. However, there are some uncertainties, as to whether simulations and experiments have exactly same test conditions, with regards to stratification in the test room.

3.3 Two story room with cold wall

The same simulations were performed for the two-storied room. In addition, simulations were run for both closed and open room, with a 30 cm downdraft spoiler 3 meters above the floor. Figure 7 shows all these results, as well as the empirical correlations. The temperature 1 m from the cold wall, well outside the convective layer, is shown in Figure 8.

The velocity results for the open room, without spoiler, show the same resemblance to the correlations as was the case for the one-story simulation. When closing the room a thermal stratification is established, with a near linear gradient. The stratification leads to reduced velocities in the cold convective layer, showing maximum values 2-3 m above the floor.

When mounting the downdraft spoiler, the flow is disrupted 3 m above the floor. Max velocities for the open room, with and without downdraft spoiler, are however quite similar, and close to the value of the correlation near the floor. The main difference, seen between the max velocities in the four simulations, is in fact between the open and closed rooms, i.e. with and without stratification.

We did not see any results indicating that cold air is ejected into the room "resembling a jet", as is sometimes claimed. Figure 9 shows local velocities in a cross section of the closed

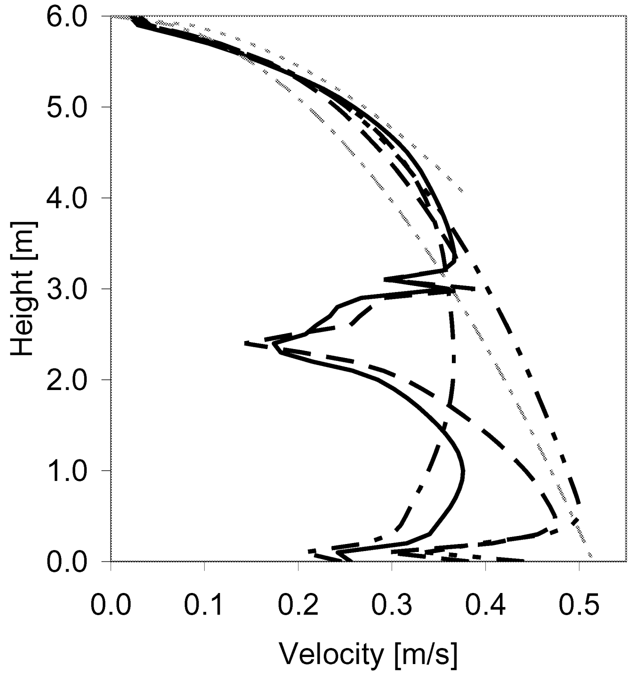


Figure 7. Max velocity along cold wall from simulations
With spoiler: closed room ——— , open room – – –
Plain wall: closed room — · — , open room – · · –
Empirical (plain wall) : turbulent , laminar

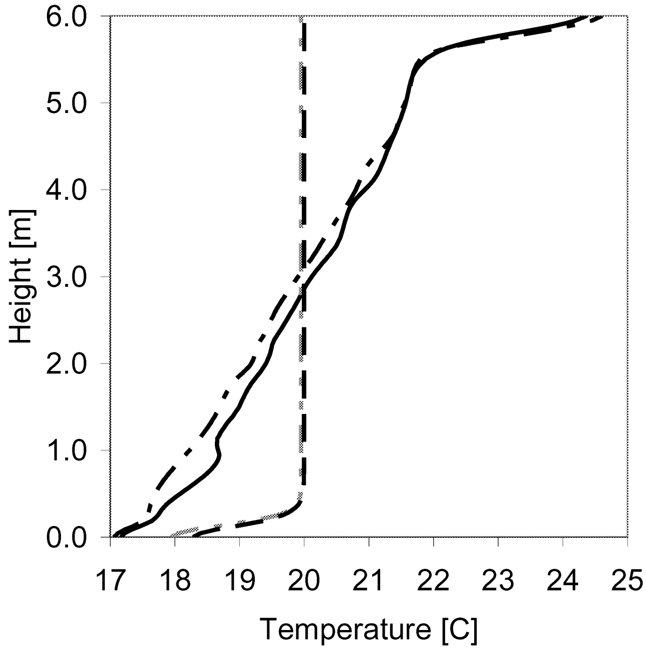


Figure 8. Temperature 1 m from cold wall, simulations.
With spoiler: closed room ——— , open room – – –
Plain wall: closed room — · — , open room [1)]
(Plain wall open room is shifted slightly, -0.05°C, to distinguish from spoiler case)

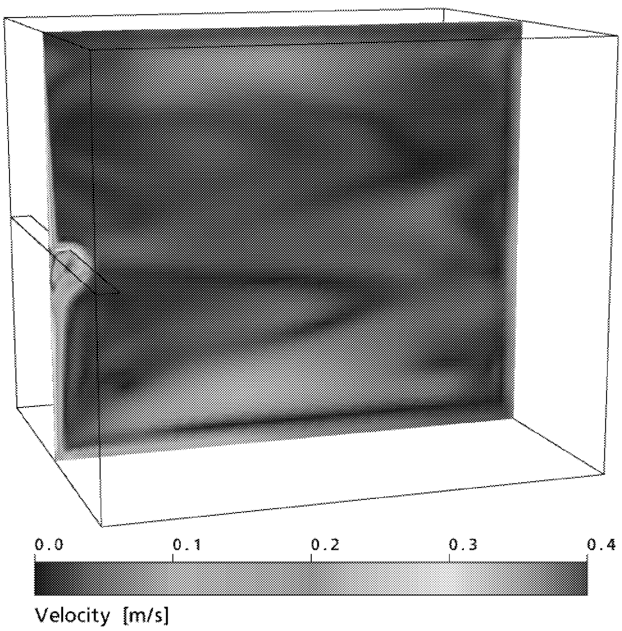


Figure 9. Velocities in cross section of (closed) test room with downdraft spoiler.

room, and how the cold convective flow is reattaching to the wall, a distance below the spoiler. The vertical velocity "structures" seen in the figure, are likely a consequence of the stably stratified conditions in the room, and that this tends to dampen vertical movements of air.

4. CONCLUSIONS

The simulation of a large atrium with downdraft spoilers suggests that these have little or no effect, at least in this case, where there is an unfavorable heat balance in the room. The circulating motion of room air seems to be much stronger than effects of obstructions in the convective layers along the façades.

Results from detailed simulations, for small one- and two-story test rooms, indicate that the developing thermal stratification has more effect on the downdraft velocities, than the downdraft spoiler. The exact values of the downdraft velocities are, however, likely to be very dependent on the boundary conditions of the room, both in experiments, simulations - and in real life.

REFERENCES

Heiselberg, P. (1994). Draught Risk From Cold Vertical Surfaces, Building and Environment, Vol. 29, No.3, pp. 297-301.

Heiselberg, P., Murakami, S. and Roulet, C.-A. (ed) (1998). Ventilation of Large Spaces in Buildings: Analysis and Prediction Techniques. IEA Energy Conservation in Buildings and Community Systems, Annex 26.

Esch T., Menter F. and Vieser W. (2003). Heat Transfer Predictions Based on Two-Equation Turbulence Models, 6th ASME-JSME Thermal Engineering Joint Conference, March 16-20, 2003.