

CFD SIMULATION OF AN OFFICE HEATED BY A CEILING MOUNTED DIFFUSER

Bård Venås*, Trond Thorgeir Harsem, and Bent A. Børresen

*Norconsult AS
Vestfjordgaten 4
1338 Sandvika, Norway*

**Corresponding author: bard.venas@norconsult.com*

ABSTRACT

The paper investigates the possibility for using a traditional ventilation system with ceiling mounted diffusers to provide heating under winter time conditions in relatively cold climates – in buildings with low transmission losses such as “passive houses”. The analysis is done through a number of CFD simulations of a simplified office. It is shown that even small over-temperatures reduce the Air Change Efficiency substantially. On the other hand even very small internal heat sources increase the efficiency. It is concluded that the principle as it works fairly well when the building is occupied, i.e. for providing peak load heating, but quite poorly without internal heat sources.

KEYWORDS

Ventilation, Heating, Air Change Efficiency, Passive House, CFD

1 INTRODUCTION

It is a general conception that using ventilation for room heating may lead to low ventilation efficiency and reduce the quality of the indoor climate. This, as buoyancy will tend to gather the introduced warm air below the ceiling with limited mixing into the occupied zone.

In most cases ventilation air is therefore introduced with lower temperature than the one that is desired for the room. Heating is instead provided by a separate system, traditionally being radiators mounted below windows in the exterior walls. In addition to the heating of room air, radiators are located in such positions to improve indoor climate, by 1) reducing cold drafts, and 2) counter the effect of cold windows on operative temperature.

However, new buildings for instance adhering to “the passive house standard” have highly insulating and airtight building envelopes. Standards Norway NS 3701 (2012) states that for an office building the U-values of windows should be lower than $0.8 \text{ W/m}^2\text{K}$, including the frame, and the remaining wall below $0.12 \text{ W/m}^2\text{K}$.

From simple hand calculations one may conclude that internal heat loads (from occupants, lightning, computers and other technical equipment) are likely to provide most of the heating during the day when office buildings are occupied. In addition to the specific use of the building and the heat loads, this evidently also depends on the ambient temperatures in the region.

This paper investigates the possibility for using a traditional ventilation system, with ceiling mounted diffusers, to provide heating under typical winter time conditions in relatively cold climates. The work is part of the research and development project “ForKlima” funded by the Research Council of Norway.

2 METHOD

2.1 Test room model

A Computational Fluid Dynamics (CFD) analysis has been performed for an “office”, based on a set-up in a laboratory test, performed by another group in the project (data have not yet been studied in detail and comparisons with measurements are not presented in this paper).

The test room has a floor area of 4 m × 4 m and a nominal height of 3 m. The walls are highly insulated and equipped with a cooling system able to control the different wall temperatures. In the laboratory test, suspended ceiling plates were mounted 2.4 meters above the floor, and the 0.6 meter high volume above the plates used for the ducting and mounting the diffuser. Experiments were then performed by introducing air of varying temperature from the diffuser, and cooling down one of the walls in the room to represent an exterior wall.

2.2 Computational Fluid Dynamics

For simplicity, and generality, the suspended ceiling was set as the upper boundary for the CFD model, and the cavity above the ceiling plates neglected.

All walls, including the suspended ceiling, were defined as adiabatic. Adiabatic conditions imply that there is no conduction through the wall, and thus, local balance along the wall between convective heat flux and radiation heat flux:

$$\text{Convective Wall Heat Flux} + \text{Radiative Wall Heat Flux} = 0 \quad (1)$$

This is a simplification, as there in practice always will be some thermal interactions with the surrounding volumes (such as the cavity above the suspended ceiling). For non-stationary conditions thermal storage in the building itself will also be paramount.

Figure 1 shows the computational domain and the numerical mesh along the surfaces. There are in addition internal heat loads in the form of a “laptop” (a point heat source, not represented by the mesh) and a thermal dummy representing a person in the laboratory tests.

As in the experiments, to mimic an exterior wall, one of the surfaces was chilled to a specified temperature which was kept constant for each simulation. All other walls were adiabatic.

The simulations were performed using ANSYS CFX 14.5. Due to the unsteady nature of the flow the simulations were run transient to achieve proper convergence. The time steps were chosen in order to reach convergence within three inner-loop iterations for every time step. The simulations were allowed to run until sufficiently stationary conditions were judged to have been achieved. This took three hours room model time or more, depending on the ventilation rate and heat loads in the room (typically 3-5000 time steps).

The mesh is a variable density hexahedral mesh consisting of a total of 460.000 cells. The cell sizes expand from 1 mm in the normal direction close to the walls, through an inflation layer of 500 mm, up to 10 cm × 10 cm in the core of the room.

Turbulence was modelled using the two-equation SST (Shear Stress Transport) model and automatic wall functions accounting for viscous effects in low Reynolds number regions. The fine mesh resolution close to the walls was chosen to obtain y_+ values of the order of 1, to

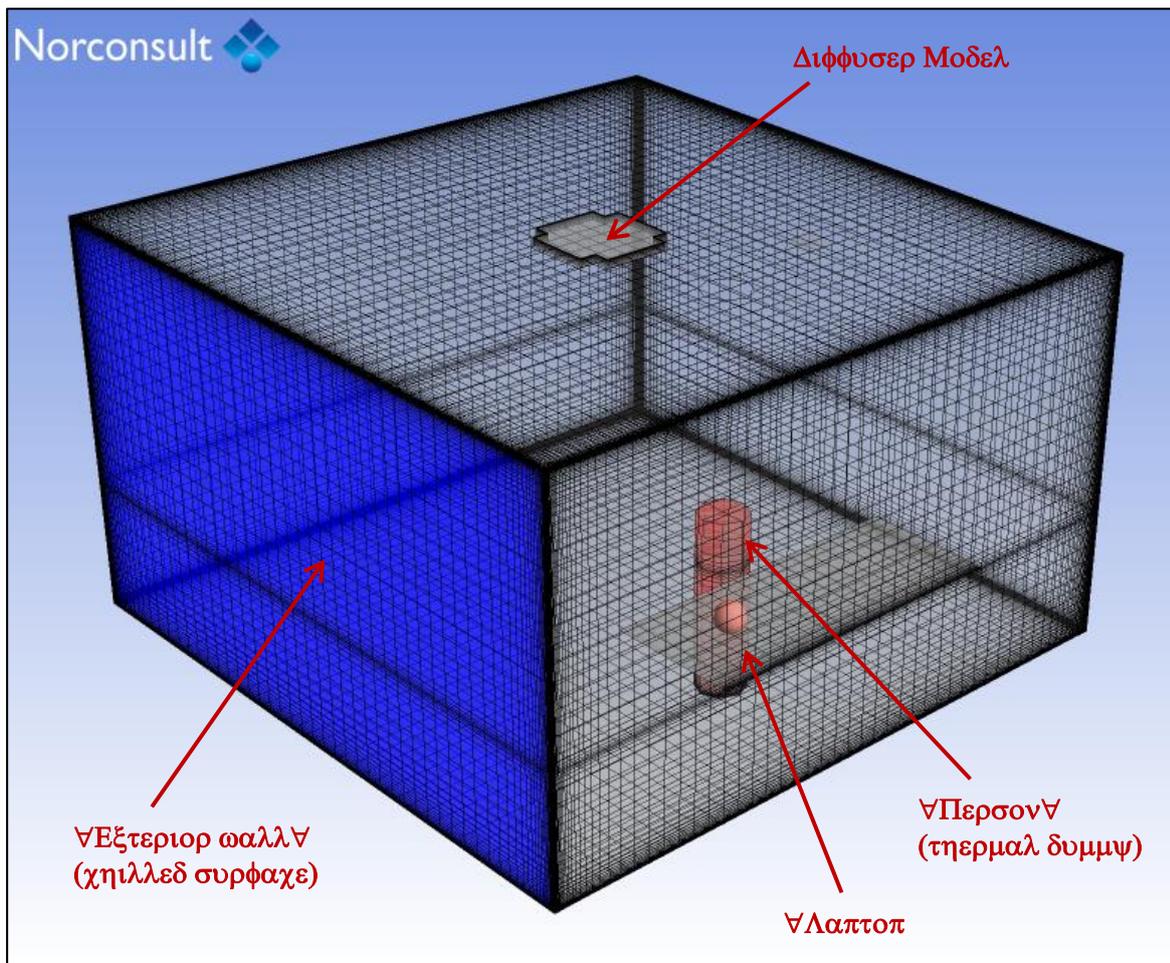


Figure 1: Geometry, boundary conditions and mesh used in the simulations. Grey walls are adiabatic.

better represent the “non-boundary layer” structure of natural convection (see e.g. Venås, B.& Børresen, B.A. (2008)) and wall-jets, flow features that are very important for the case under consideration.

Thermal radiation was modelled using the Discrete Transfer Model with 32^2 rays for ca. 13000 separate radiation elements. The turbulent Schmidt number was kept at its default value of 0.9 for all scalars.

2.3 Diffuser model

A specific type of radial VAV diffuser was mounted in the test room. This diffuser consists of a number of parallel baffles whose internal separation is controlled by the unit as a function of desired air flow rate and air pressure in the supply duct.

Due to the complexity of this geometry the CFD simulations were performed in two steps. First initial runs were performed with a highly resolved 2D diffuser-only model. These results were then loaded into the test room model to provide "effective" diffuser boundary conditions for the room model.

Figure 2 shows results from the diffuser model. One can notice how the exit jets between the individual baffles, first joins together, and then attaches as one to the ceiling due to the

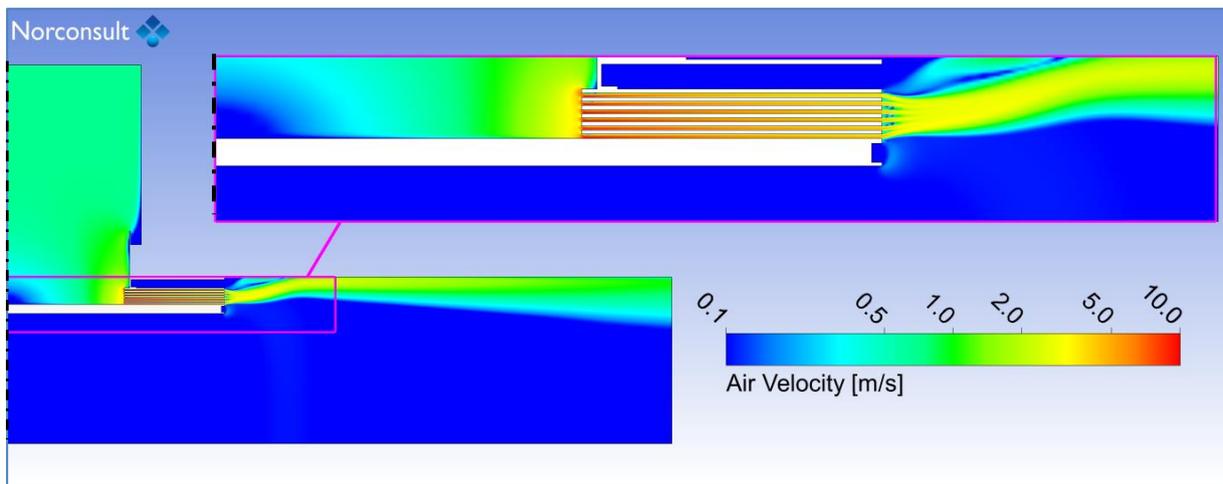


Figure 2: Air Velocity from CFD diffuser model. The centreline of the diffuser (rotational symmetry axis) follows the left edge of the figure. Note that a logarithmic velocity scale is been used.

Coanda effect. The flow then continues outwards, as a radial wall-jet, sticking to the ceiling as it expands away from the diffuser.

Two volume flows have been investigated in this work, respectively 18 litres per second (l/s) and 34 litres per second (the case shown Figure 2).

The diffuser models were run isothermally and the mixing of inlet air and entrained air was represented based on the concentration of an inlet air scalar at the diffuser model interface. This concentration marker was used to evaluate the scalar quantities in the effective diffuser inlet for the room model (temperature and age of air), i.e. determining the mixing between inlet air and entrained air near the diffuser.

In the current simulations the individual baffle separation was kept at “82.5 % opening ratio” (equal to 1.23 mm), as this was typically the opening ratio in the lab measurements.

Figure 3 shows air velocity in a cut through the room for an isothermal case. In this situation there were no heat loads or wall cooling, so that the room temperature perfectly equals the inlet air temperature. The inserted lower left frame shows how the diffuser model is interfaced with the room model.

In the figure one can notice how the air flow from the diffuser spreads out along the ceiling, as in the diffuser model, but is diverted downwards when the flow hits the vertical walls. A distance down along the wall a fluctuating separation of the flow from the wall takes place, and large vortices are formed in the room. The air velocities are generally low, typically less than 5 cm/s, except for close to the ceiling and the upper parts of the walls.

2.4 Air Change Efficiency

To investigate the efficiency of the ventilation system in replacing the air in the room – while simultaneously providing heating – the *Air Change Efficiency* has been evaluated. This measure represents how efficient the ventilation works; i.e. relative to the specific amount of ventilation.

Air Change Efficiency (or *Ventilation Efficiency* in general) can be evaluated in different ways. The method employed here is to calculate the local *Mean Age of Air* based on the CFD data for flow pattern advection and turbulent mixing in the room. In practice this is done by introducing a constant scalar flux throughout the volume inside the room, and letting the scalar accumulate until stationary conditions are achieved. The scalar then represents the time the air has been in a certain position.

The nominal time constant τ_n is used as reference. This time always corresponds to the mean age of the air exiting the room

$$\tau_n = \frac{V}{Q} \quad (2)$$

The overall room Air Change Efficiency is then found as

$$\varepsilon_{\text{room}} = \frac{\tau_n}{2 \cdot \langle \tau \rangle} \quad (3)$$

where $\langle \tau \rangle$ is the volumetric average of the mean age of air. The Air Change Efficiency in a specific location equals

$$\varepsilon_{\text{point}} = \frac{\tau_n}{2 \cdot \tau_{\text{point}}} \quad (4)$$

where τ_{point} is the local mean age of air in that location.

The factor “2” in the denominator of eq. (2) and (3) normalizes the expression so that perfect “piston flow” corresponds to $\varepsilon = 100\%$, whereas “full mix” equals $\varepsilon = 50\%$.

In theory the full mix definition stems from an idealized situation where the mean age of air is equal in the whole room (and thus also at the outlet). This is typically assumed to be the best practically achievable value for small rooms where there is not any possibility of obtaining efficient displacement ventilation.

The scale to the left in Figure 4 shows the mean age of air for the isothermal simulation in Figure 3. As the Air Change Efficiency simply is an inverse of the mean age of air it can be represented by the “inverse scale” which is shown to the right in the figure.

In the figure one can notice that new air inserted from the diffuser flows along the ceiling and “ages” with distance from the diffuser due to the elapsing time, and due to mixing with older air in the central parts of the room.

For more on Air Change efficiency see e.g. Novoselac & Srebric (2003).

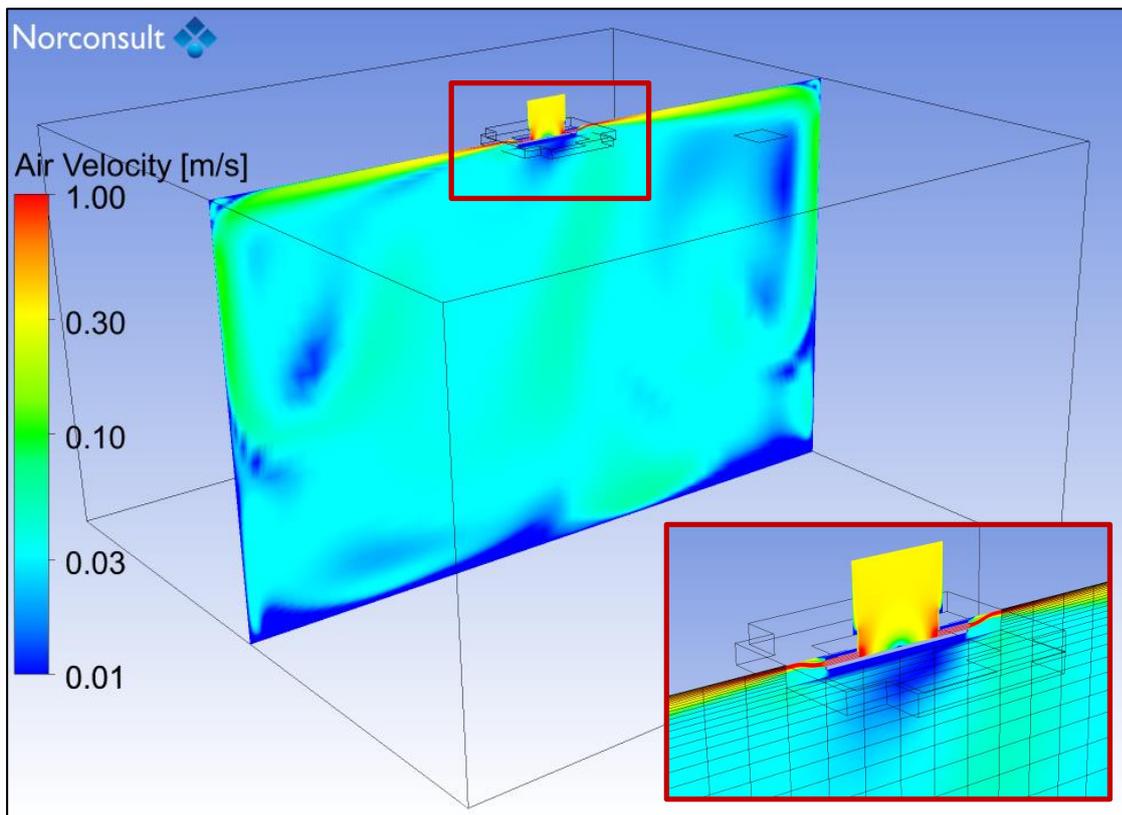


Figure 3: Air velocity in a cut through the room (note: logarithmic scale). Isothermal case with 18l/s ventilation. Close-up on diffuser model inserted lower right.

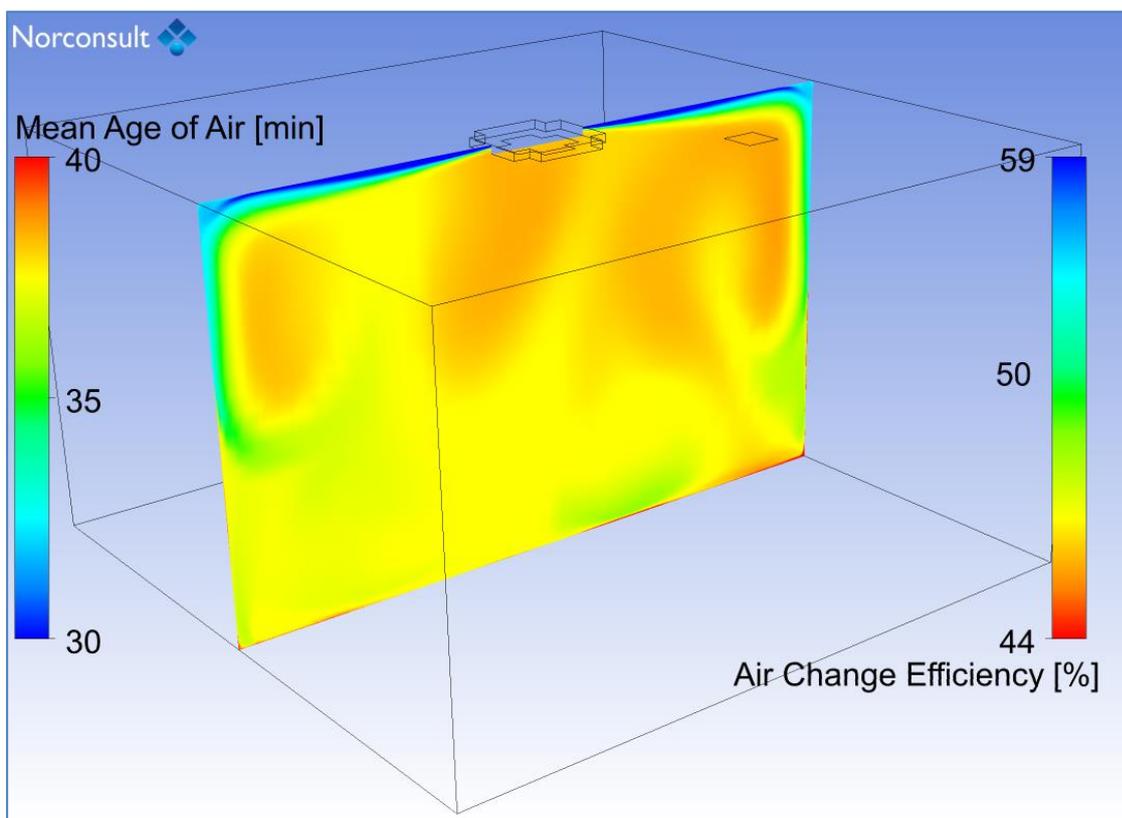


Figure 4: Local Mean Age of Air and Air Change Efficiency (shown by inverse scale to the right). Isothermal case with 18l/s ventilation.

3 RESULTS

A total number of 30 simulations have been performed, combining varying inlet temperatures, with different internal temperatures on the “exterior wall” and different internal heat loads.

Figure 5 shows Air Change Efficiency, as function of over-temperature, for the simulations performed without any internal heat loads. The isothermal cases here correspond to the leftmost measuring points in the figure. The fully drawn lines represent the efficiency in a measuring point 1.1 m above the floor next to the “desk” in model, whereas the dashed lines represent results for the room as a whole.

We can see that the efficiency is 46 to 47 % for the isothermal cases, both for the occupied zone and the whole room. Thus all being close to the full mix value.

When increasing the over-temperature (by increasing inlet temperature and decreasing the temperature of the chilled surface) the efficiency drops rapidly. For the 18 l/s ventilation case just 1 K over-temperature is enough to reduce the efficiency considerably. For 34 l/s one degree does not significantly change the efficiency, but it starts dropping for 2 K.

Another way of interpreting these results, which may be even more relevant for the problem at hand, is to calculate which ambient temperatures the total heating corresponds to. This has been done by using the upper limiting U-values for a passive house (see page 1), combined with an assumption that 2/3 of the exterior wall consists of windows.

The upper part of Figure 6 shows the same results as Figure 5 presented in this way. One can see that when heating using only ventilation at ambient temperatures of e.g. -10°C or -20°C the efficiency is low. Increasing the ventilation rate to 34 l/s improves the results, and the natural trend seems to be that using more ventilation at lower temperature is more efficient.

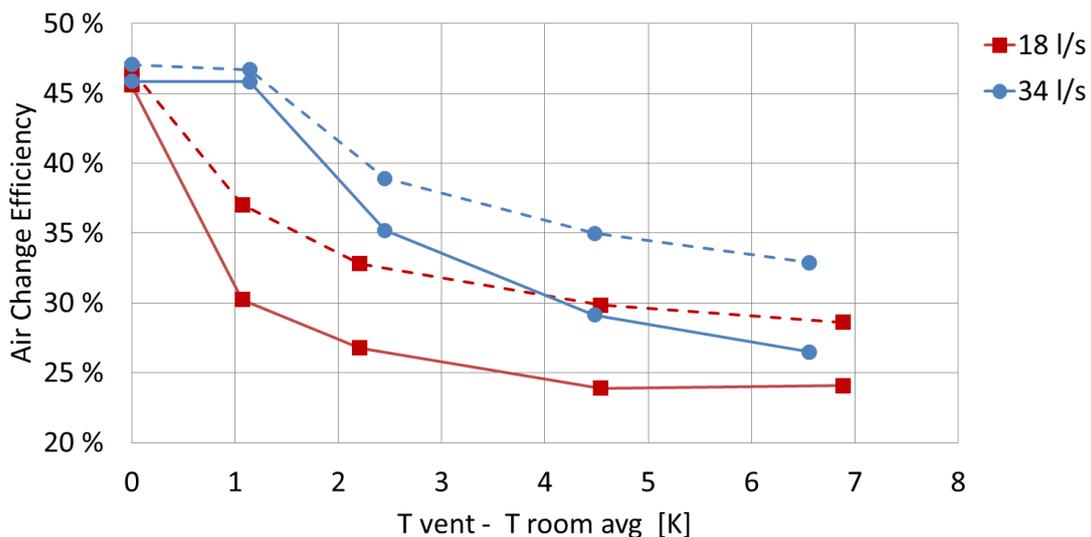


Figure 5: Air Change Efficiency without any heat load in room. Dotted lines are room averages.

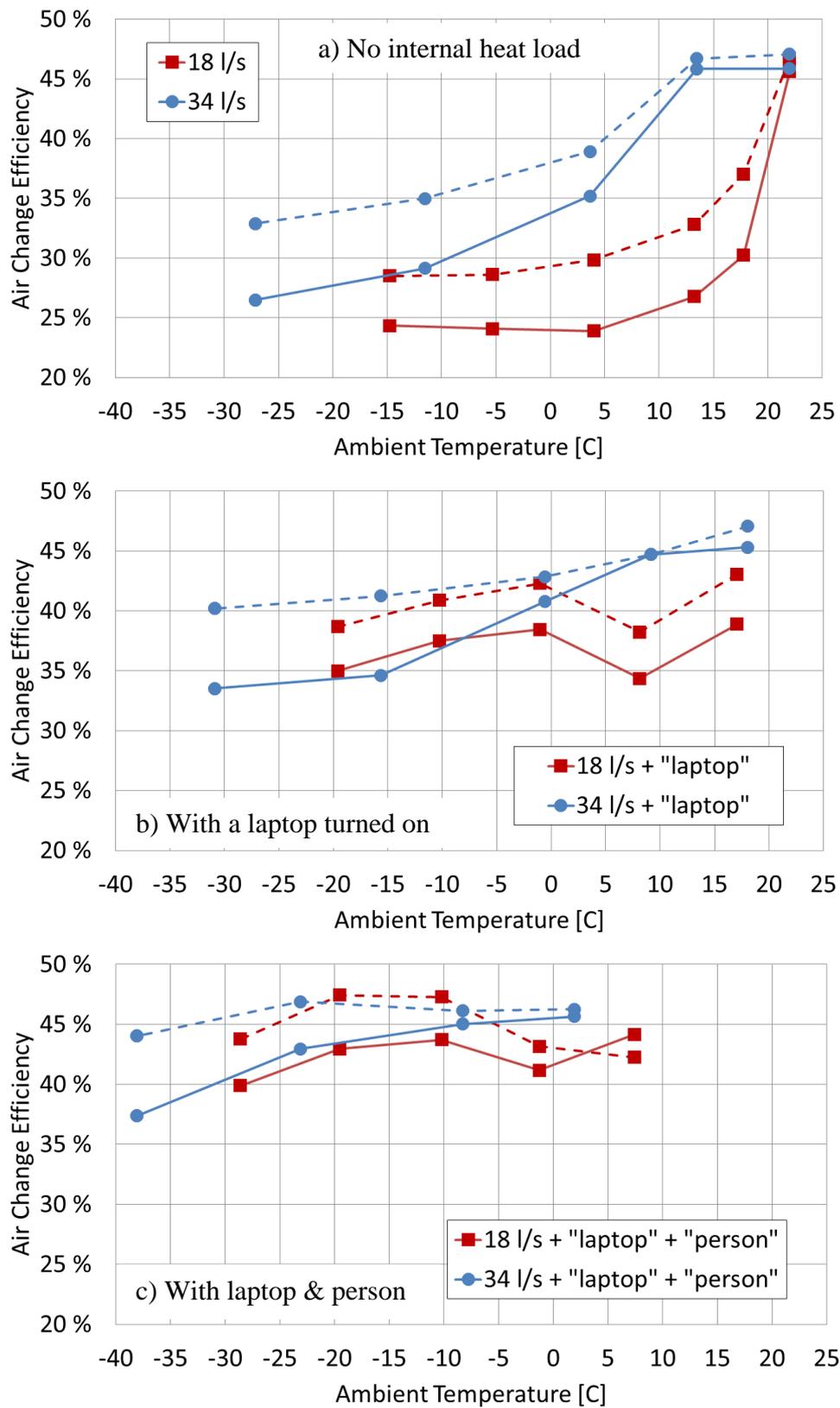


Figure 6: Air Change Efficiency at function of ambient temperatures when ventilation is used for heating – and for situations with respectively: a) no internal heat load (upper); b) with a laptop turned on (middle); and c) with laptop and person (bottom). Fully drawn lines are results from the occupied zone, and dotted lines room averages.

The middle and lower parts of Figure 6 show the corresponding curves when introducing, first a “laptop” computer releasing 40 Watt of convective heat, and then adding a person releasing 35 Watt convective and 35 Watt radiative heat.

One can see that the Air Change Efficiency increases substantially as more of the heat is added from internal sources. It is considered probably to be an important factor that the heat is introduced at low heights, as the rising plume in this case entrains more room air and thus increases the circulation in the occupied zone, than if they were located high (e.g. lights).

Figure 7 shows plane cuts of Air Change Efficiency through the location of the “laptop” and the “thermal dummy” for different simulations with 18 l/s ventilation rate.

The upper left part (a) shows the results for the fully isothermal case: the flow is circulating and being distributed in the whole room (same as Figure 4 but shown with a different scale).

The upper right part (b) represents the situation for an over-temperature of ca. 2 K without internal heat gains. One can see that there is high efficiency near the ceiling and low efficiency in the occupied zone. The downdraft along the chilled wall brings some new air into the occupied zone.

In the lower left part (c) the plume caused by the 40 W released by the laptop increases the mixing in the occupied zone considerably, bringing old air upwards, and a consequently new fresher air into the zone. The Air Change Efficiency thus increases.

The lower right figure (d) exhibits even more improvement. It is mostly the convective part of the heat that sets up the plume updraft, and this is roughly doubled by introducing a person in addition to the laptop.

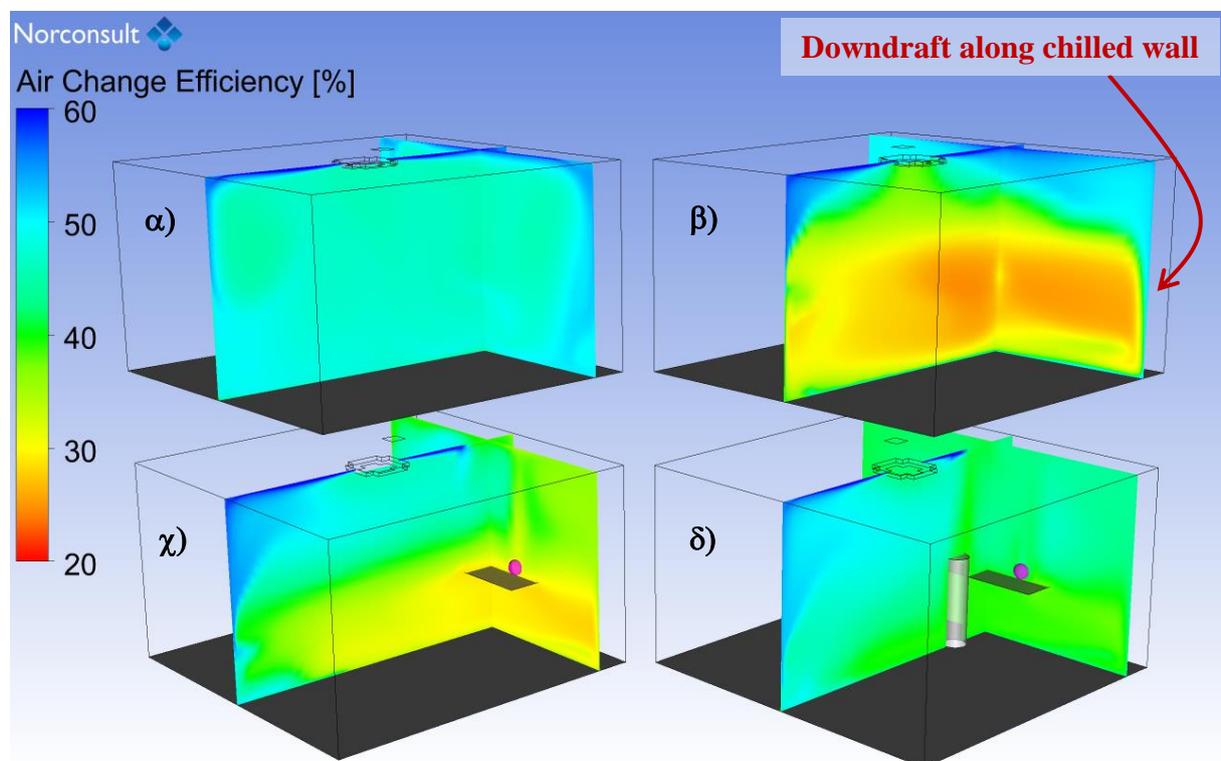


Figure 7: Comparison of Air Change Efficiency with 18 l/s ventilation, combined with respectively: a) isothermal; b) $\Delta T \approx -2\text{K}$, no internal loads; c) $\Delta T \approx -2\text{K}$; “laptop” only; and d) $\Delta T \approx -2\text{K}$, “laptop”+ “person”.

4 DISCUSSION AND CONCLUSION

The main findings considering the flow physics are summarized in Figure 8. The radial wall-jet flows outwards and is bent downwards by the side walls. The downwards “penetration depth” is reduced when thermal stratification in the room increases. At the same time all simulations with a cold wall exhibit a low-velocity downdraft that flows down the entire wall and into the room along the floor, and thus brings fresh air to the occupied zone.

Introducing internal heat loads, which represent low-level heat sources, increase the mixing by setting up thermal plumes which improve the ventilation efficiency considerably. A laptop computer or a person is enough.

When it comes to conclusions for using this type of ventilation for heating it seems like adding additional heat for peak loads is fully possible when the building is occupied, i.e. during work days and work hours for an office.

On the other hand the efficiency is quite low when using over-temperature in a situation without internal heat loads. Work continues on finding ways to improve this.

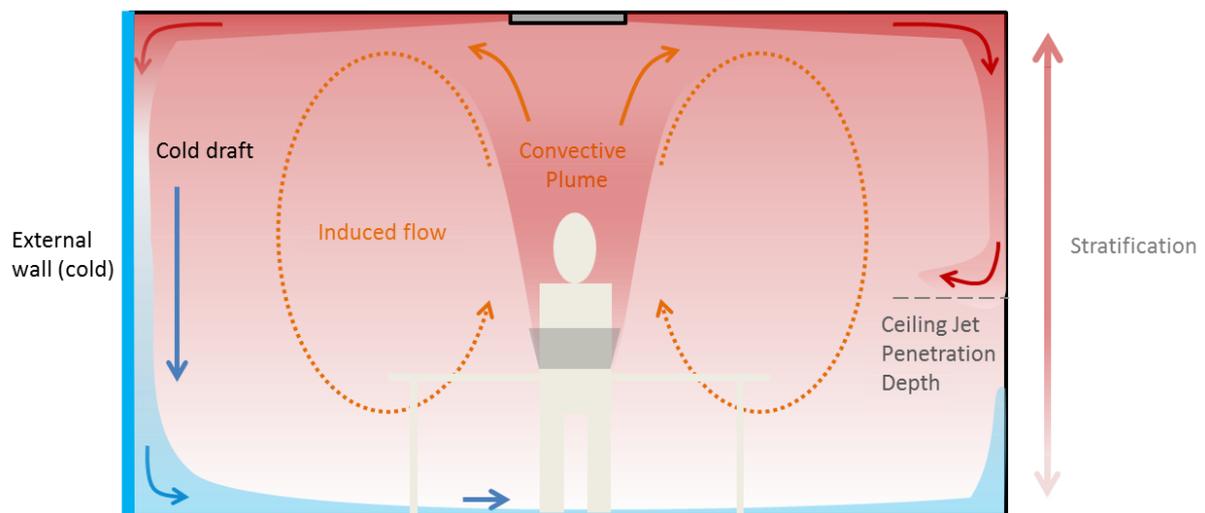


Figure 8: Sketch showing the interpretation of working principle of the studied flows.

5 ACKNOWLEDGEMENTS

This paper was written in the context of the research and development project ForKlima, funded by the Research Council of Norway. RCN as well as the partners of the project are gratefully acknowledged.

6 REFERENCES

- Novoselac, A., & Srebric, J. (2003). Comparison of Air Exchange Efficiency and Contaminant Removal Effectiveness as IAQ Indices. *ASHRAE Transactions* 109 (2), pp. 339-349.
- Standards Norway NS 3701. (2012). *Standards Norway NS 3701:2012: Criteria for passive houses and low energy buildings - Non-residential buildings* (in Norwegian).
- Venås, B., & Børresen, B. (2008). On the Use of Downdraft Spoilers in Glazed Atria. *29th AIVC Conference, Kyoto, Japan, 14-16 October 2008*.