Isolation Rooms - CFD Simulations of Airborne Contamination Through Doors During Passage

Harsem, Trond Thorgeir*[1]; Venås, Bård[1]; Vikan, Anders Welde[1]; Lind, Merethe Cecilie[1]; Kalliomäki, Petri[2]; Koskela, Hannu[2]

1) Norconsult AS
Vestfjordgaten 4
1336 Sandvika, Norway
**Corresponding author: tth@norconsult.no*

2) Turku University of Applied Sciences,
Indoor Environment Research Group
Lemminkäisenkatu 12-14
20520 Turku, Finland

ABSTRACT

There has been an increase in diseases caused by airborne infections such as influenza A/H1N1 or SARS in the recent years. Airborne infection isolation rooms are commonly used to limit the spread of airborne infections. The challenge today is that there is only a limited number of airborne infection isolation rooms in each hospital (class P4). The rooms are expensive to build and airflow control to avoid contamination is often complicated. Pressure difference between the isolation rooms and the corridor ensures that persons who stay outside the isolation are protected against the infected person. An ongoing research project aims to achieve a 90% reduction of transferred particles using 10% of the cost compared to the strictest class of airborne infection isolation rooms.

As part of this project CFD (Computational Fluid Dynamics) simulations have been performed for the “baseline cases”, i.e. air flows induced by opening and closing a hinged door between a patient room and an anteroom with balanced ventilation. The results are compared with laboratory experiments. The Overset Mesh method was used to model the door and the person moving between the rooms.

The simulations and the laboratory experiments for “door movement only” and “person exiting” the patient room showed very similar results. From the simulations for a person entering the patient room it was found that the air exchange is very sensitive to the velocity and the actual moving pattern of the person. The results show that the simulation model will enable reliable CFD simulations for more complicated cases in the future.

The simulation model will be used in the further work, which will include pressure differences between rooms, to investigate the sensitivity of walking speed and the movement of persons and to create simplified solutions to limit contamination from patient rooms.

KEYWORDS

Isolation rooms, Airborne infections, Contamination control, CFD simulation, Overset Mesh methods

1 INTRODUCTION

Airborne infection isolation rooms (AIIR) with pressure difference between the isolation rooms and the corridor ensures that persons who stay outside the AIIR are protected against the infected persons. Today there is only a few number of airborne infection isolation rooms in each hospital. The rooms are expensive to build and airflow control to avoid contamination is often complicated. Several studies in the last few years have examined air flow movements causing containment failures in AIIRs. Although the rooms are in negative pressure related to adjacent spaces, the door opening removes the pressure difference and hence the air and airborne contaminants are free to escape to the surrounding space. It has been estimated that door opening and passage through the doorway is among the main factors causing containment failures (Tang et al. 2006). For instance, Hayden et al. (1998), Saarinen et al. (2015) and Kalliomäki et al. (2016) have mapped air flows generated by a door opening with and without passage and showed that opening of a hinged door can induce a substantial air volume transfer across the isolation room doorway.

This ongoing research project aims to find simplified solutions that can be implemented in normal patient rooms to create negative pressure in the patient room and the anteroom compared to the surrounding areas. Also, limiting the initial air volume transfer can reduce the risk of contaminant failure. The overall aim is to significantly reduce the amount of contaminated air that escapes from the patient room to a fraction of the cost.

CFD (Computational Fluid Dynamics) simulations of the baseline cases are presented in this paper. The baseline consists of door opening of a hinged door in a normal bed ward with balanced ventilation. To ensure and verify the results, it was considered necessary to carry out laboratory measurements to validate the CFD simulation model. The laboratory methods are presented in detail by Harsem et al. (Harsem et al., 2018).

2 METHOD

2.1 Simulation model

Numerical simulations have been performed using the commercial CFD code ANSYS Fluent. The model represents a full-scale patient room with an anteroom, and the simulations aim at describing the air migration when people move between these two rooms.

Figure 1 shows the layout of the rooms, with closed door and the person standing in the centre of the patient room. The geometry, diffusors, and heat sources are the same as in the laboratory experiments with which the results are compared (Harsem et al., 2018).

The patient room has dimensions: 4 m × 4.7 m × 2.6 m, and the anteroom: 2.4 m × 2.4 m × 2.45 m (depth × width × height). The door is 1.22 m × 2.04 m × 0.04 m (width × height × thickness) with a 2 cm gap below. There are four heated plates in the patient room to represent internal heat gains (solar heating and equipment). In the laboratory a heated mannequin was lying the bed, but this geometry was not included in the CFD-model. There are two air extracts in the patient room and one in the anteroom.

The air ventilation inlets in both rooms are from Halton DHN nozzle diffusers. These were modelled using the "circular opening model" described by Kanerva et al. (Kanerva, et al., 2018). This methodology implies that each nozzle in the diffuser is represented by a circular inlet boundary condition with a horizontal flow component along the direction of the nozzle, in addition to the vertical velocity defining the volume flow into the domain. Based on results from the reference a flow angle of 30° between ceiling and the inlet velocity vector was used.

The ventilation rates were 50 l/s in and 57 l/s out from in the patient room, 14 l/s in and 15 l/s out from the anteroom. In addition, 7 l/s came from an inlet representing the gap below the door to the corridor (the corridor itself was not included in the CFD model). The inlet temperature was 18.5 ºC. All heat sources were assumed to release 50 % of its heat as radiation, this radiation was assumed distributed across all surfaces in the room, and the heat re-entered the room as convective heat from walls, floor and ceiling. In total 75 W were added to the anteroom and 240 W in the patient room, yielding a room temperature of ca. 22.5 ºC.

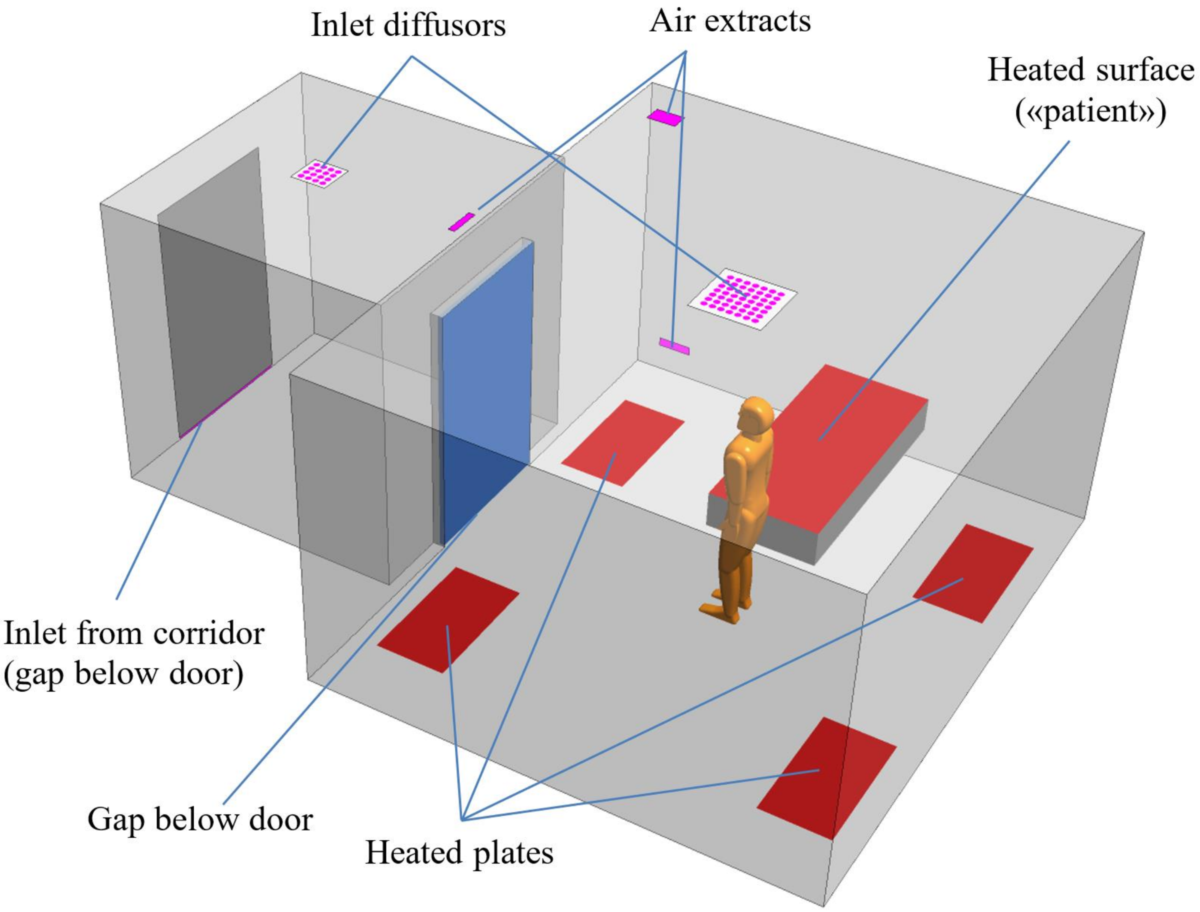

Figure 1: Lay-out of patient room and anteroom (ventilation and heating as in the laboratory).

2.2 Modelling moving objects

There are several methods available to model moving objects inside a CFD domain. Some methods, referred to as Immersed Boundary Methods, model objects through volumetric algoritms without the surfaces of the objects actually being resolved by the mesh. The main disadvantage with these methods are that they may produce less accurate solutions for complex geometries and frictional forces compared to methods that resolve the object surfaces. However, such methods have been used in very interesting work, e.g. by Saarinen et al. (2017) who used the Immersed Solid model in ANSYS CFX to perform LES simulations of door passages.

In methods that resolve the object, the mesh structure has to change during the simulation to follow the moving object at all times. Such methods include Mesh Deformation, i.e. simulations where flexible meshes accomodate movement by expanding and compressing the mesh cells, while maintaining the same overall mesh topology. Another option, Moving Domain methods, notably used for turbomachinery, splits the total domain into subdomains (e.g. for rotor and stator) which move relatively to each other, passing information across sliding mesh interfaces connecting the domains. Both of these methods, however, put large constraints on the changes in topology that they can represent and are difficult to use for the topic at hand.

An approach, where changes in topology actually can be achieved, is automatic remeshing of the geometry during simulation. The main disadvantage with this technique is that automatic meshing algoritms often are far inferior to manual methods, with regard to mesh quality.

A third alternative, which has more recently been included in commercial CFD software packages, is the Overset Mesh method, often referred to as Chimera grid methods. In this approach modelling of changing geometry is achieved using separate meshes for the "background" and the moving objects. The simulation is performed for all the meshes simultaneously and information is passed between the meshes in the zones where they overlap.

The Overset Mesh method has a combination of advantages that makes it suitable for the current simulation of a rotating door and person moving: there are no limitations on the changes in topology that can be included, while at the same time, the surface boundaries of the moving objects can be resolved with high detail.

A real person walks in a very complicated manner, moving the body and limbs in a coordinated way. Although such movement is possible to mimic with the Overset Mesh method (using one mesh for each body part) – it was chosen to implement a simpler "sliding" representation of the person at this time. The model of the person, and the Overset Mesh surrounding it, is illustrated in Figure 2.

The Overset Mesh method in Fluent divides the cells in all the meshes into one of the following types: donor, solve, receptor, orphan, or dead. Figure 2 shows the active overset meshes for door (blue) and person (green), as well as, for the background mesh (grey). The outer extent of the two overset meshes are shown as blue and green lines. The regions between these lines and the similarly coloured meshes are inactive (receptor, orphan or dead) at this specific time.

All the meshes were hexahedral with cut cell refinements. Cell counts were 3.7 million for the background mesh containing the two rooms, and respectively 1.1 million and 400,000 cells for the overset meshes door and person.

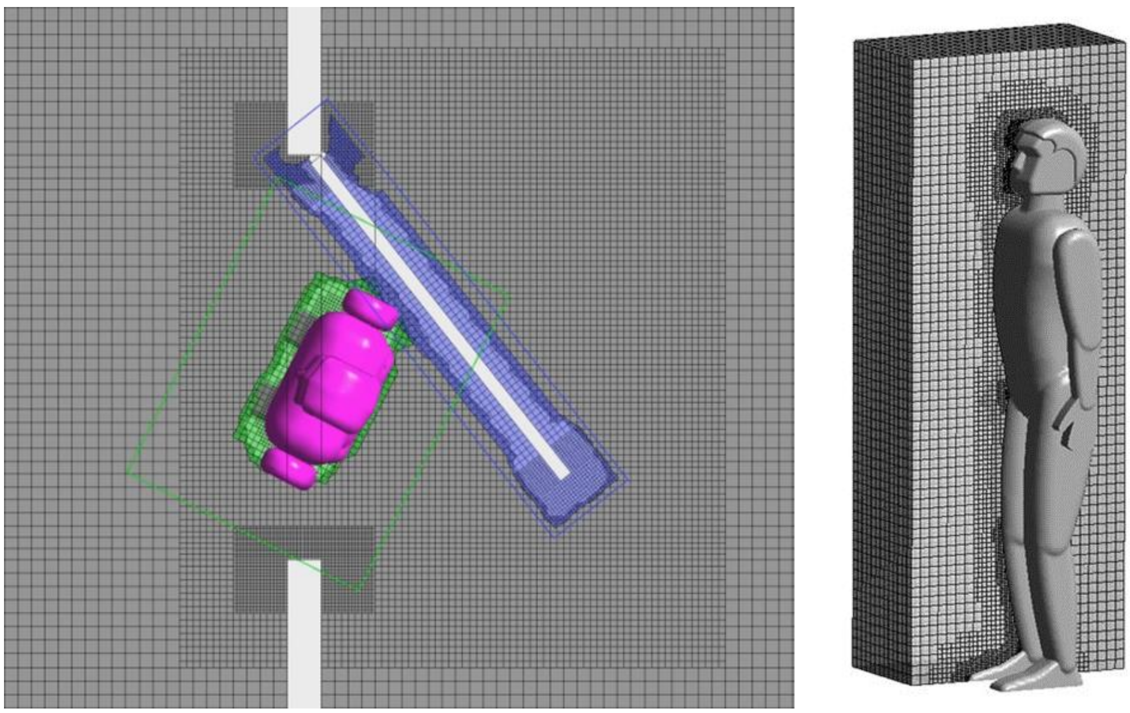

Figure 2: Active cells for the three meshes during passage through the door (left) and mesh for person (right).

2.3 Modelling of passage through door

Simulations were performed for passage of a person moving from anteroom to patient room, and in the opposite direction, from patient room to anteroom. Standardized motion sequences, approximating the movement of a real person in the laboratory were implemented for door and person. The path and timing were defined by tabulated waypoints, given in Table 1 and Table 2, and shown Figure 3. The door opening and closing sequence is 2 seconds for opening, 1 second hold, and 4 seconds for closing.

Table 1: Waypoints describing the person and door when moving from anteroom to patient room.

Waypoint no.	Time [s]	Action
1	0.0	Person starts to walk from anteroom.
2	2.2	Person stops, pushes opener, and door starts to open.
3	3.0	Person starts walking
4	4.8	Change of walking direction
5	6.0	Person stops in patient room
6	9.2	Door closes.

Table 2: Waypoints describing the person and door when moving from patient room to anteroom.

Waypoint no.	Time [s]	Action
1	0	Person starts to walk from centre of patient room.
2	2.4	Person stops, pushes operator, and door starts to open.
3	3.9	Person starts walking again.
4	4.8	Change of walking direction.
5	5.3	Change of walking direction.
6	6.7	Person stops in anteroom.
7	9.4	Door closes.

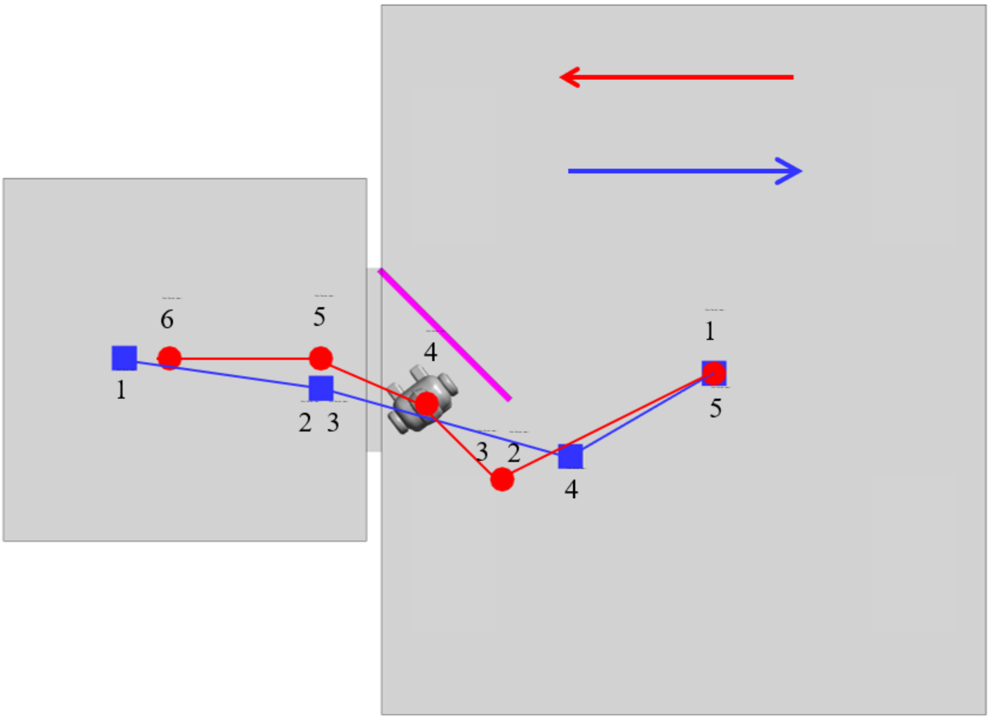

Figure 3: Waypoints for exiting (red circles) and entering (blue squares) the patient room.

It should be noted that, in the simulation model the movement is approximated through accelerations, in the two horizontal directions, up to a maximum walking speed (roughly, between 1.2 m/s^2 up to 0.6 m/s, and 1.8 m/s^2 up to 0.9 m/s). The model is defined to constantly rotate into the direction of travel, i.e. with the chest normal to the direction of travel. The implementation of the method, using accelerations, meant that the changes in direction was smoothed somewhat.

2.4 Simulations

Three different cases were simulated using CFD and tested experimentally in the laboratory. All cases where with balanced ventilation and isothermal conditions:

1. Door opening without passage
2. Door opening with passage from anteroom to patient room ("entry")
3. Door opening with passage from patient room to anteroom ("exit")

All the simulations were started using a fully developed ventilation and temperature simulation as initial conditions. In addition, the patient room was initially filled with a passive scalar. The amount of scalar inside the anteroom at the end of the simulation thus represents the air transfer during the passage.

Usually, in CFD simulation one would perform a volumetric integration to determine how much of the scalar has ended up in the anteroom. However, the Overset Mesh model in Fluent has not yet implemented volume integrals correctly. This is to say, the volumes which are active in the overlap between the meshes are not handled correctly, resulting in some "double counting". To overcome this limitation, a second model ventilated the anteroom separately and measured the concentrations in the extract air at the end of the simulations - in other words similar to the method used in the laboratory experiments. The ventilation rates were, however, drastically increased after the door was closed to keep the simulation time down.

2.5 Laboratory experiment

The experiments were carried out in an isolation room model built into a ventilation laboratory at Turku University of Applied Sciences. Tracer gas measurements using SF6 were carried out to quantitatively assess the air volume exchange between the patient room and the anteroom generated by the door opening and passage. The air volume transferred from the patient room to the anteroom was calculated by integrating the area under the tracer gas decay curve in the anteroom exhaust and by multiplying it with effective exhaust flow rate. The air volume transfer was calculated as:

$$V = Q_{eff} \frac{\int_0^\infty C_e(t)dt}{C_0} \tag{1}$$

The complete description of the methods for the laboratory experiments are presented in detail by Harsem et al. (Harsem et al., 2018).

3 RESULTS AND DISCUSSION

Four simulations have been performed and compared to laboratory measurements. The air transfer calculated from these are listed in Table 3.

Table 3: Air transfer results from CFD simulations and laboratory experiments.

Simulation no.		**1**	**2**	**3**	**4**
Description		Door only	Exit	Entry	Entry (alt.)
CFD model		755 litres	781 litres	1098 litres	682 litres
Experiments	Average	765 litres	729 litres	802 litres	
	Max	810 litres	751 litres	843 litres	
	Min	720 litres	684 litres	770 litres	
Difference (exp. vs. CFD)		1 %	- 7 %	-27 %	+ 18 %

First, one can notice that Simulation 1 (door only, i.e. without passage) gave very similar results in the CFD model and in the laboratory, respectively 755 and 765 litres. The laboratory tests varied between 720 litres and 810 litres over 8 experiments.

In the case where the person exits the patient room (Simulation 2), the CFD model found that 781 litres of patient room air followed the person to the anteroom. The experiments gave an average of 729 litres, spanning between 684 litres and 751 litres over four repetitions. The deviation between Simulation 1 (door only) and Simulation 2 (exit) is considered relatively small (7 %). Figure 4 seems to illustrate that moving through a door that opens towards oneself induces limited amounts of extra air exchanged compared to door only situation.

The difference between the CFD model and the experiments for the entry case, however, was found to be more significant (Simulation 3). The experiment gave about 25 % less transfer of air than the simulation. The variation within the four repetitions of the experiments is also quite small compared to this difference.

An earlier simulation of the air exchange (Simulation 4), with a less “accurate” representation of the passage, may shed some light on the origin of the difference. In that simulation, the “person” held a velocity of ca. 0.6 m/s through the door opening compared to ca. 0.9 m/s in this simulation. This simulation (Simulation 4 in Table 3) produced only 682 litres of air exchange, i.e. considerably lower than the experiments. The velocity 0.6 m/s is roughly the velocity held when exiting through the door (lower velocity due to the changes in walking direction).

The difference between Simulation no. 3 and 4 suggests that the aerodynamic drag forces, at least for the person entering the patient room, is very important. As a first approximation (without considering the door induced air velocity) a 50 % increase in speed roughly doubles the drag force and the slipstream behind the person.

Figure 5 shows the wake behind the person entering the room (from Simulation 3) and that the wake affects the flow out of the anteroom even after the person has entered the room. Due to continuity, a similar volume air flow is transported in the opposite direction, from the patient room to the anteroom.

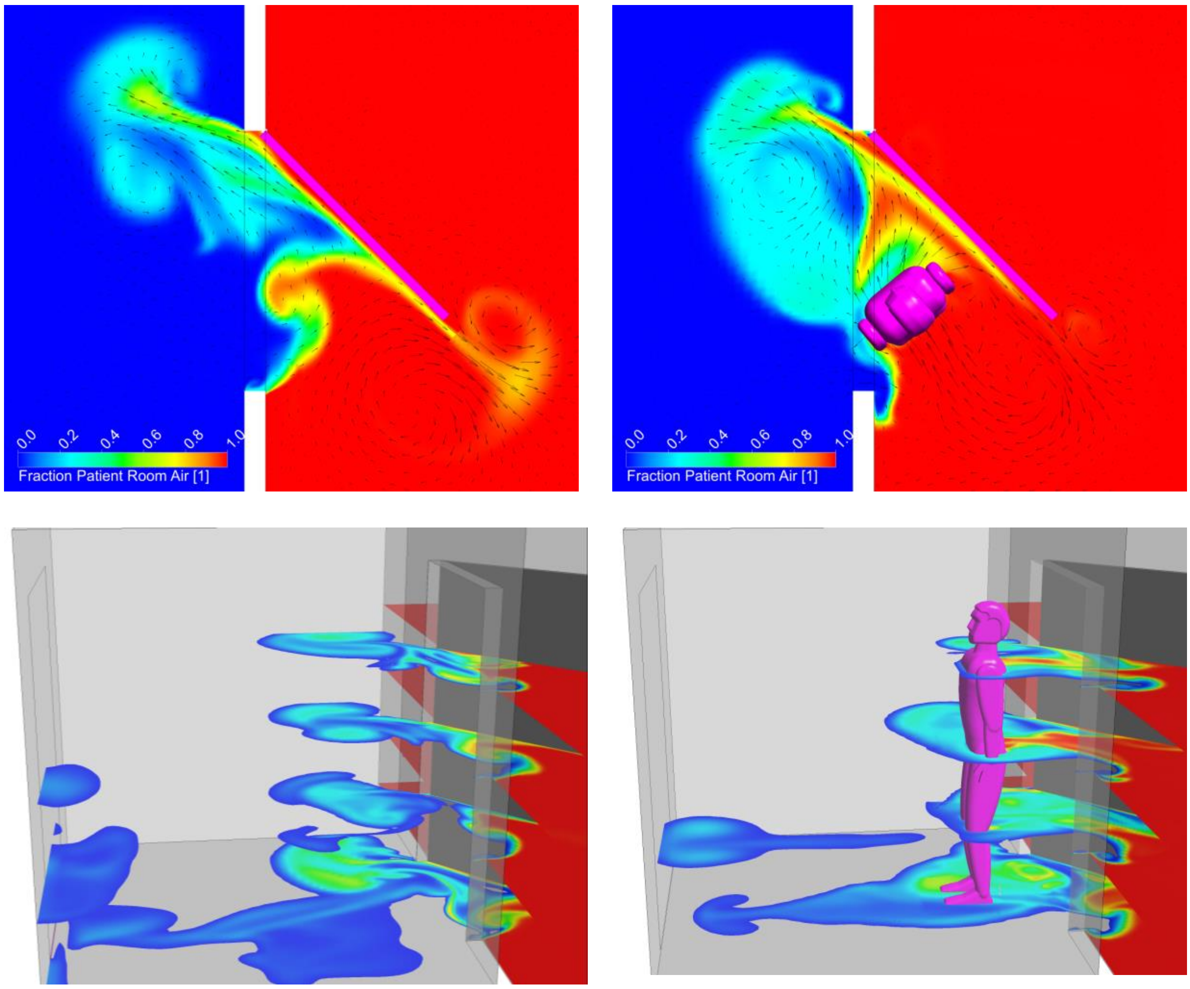

Figure 4: Exchange of air from patient room to anteroom; door only moving (left) and during exit (right).

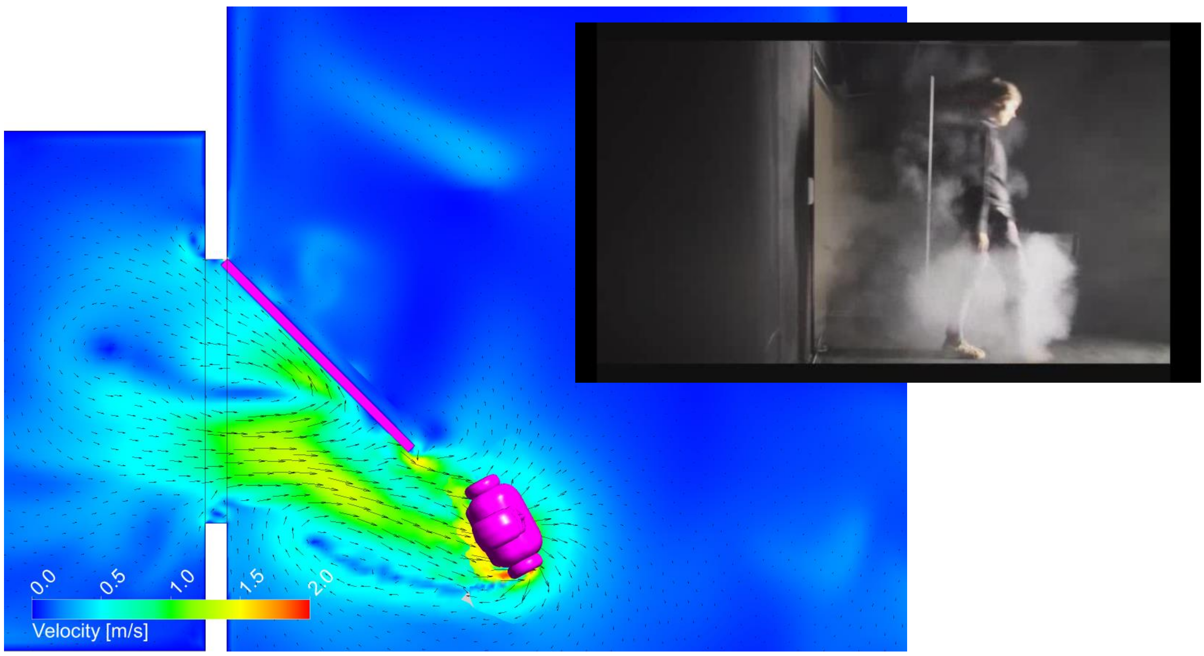

Figure 5: Person moving into the patient room illustrating strength of the wake. Top right: smoke visualization.

Many aspects of the movement differ between the CFD model and the experiments. A video study of a person walking in the lab, revealed several interesting features. First, the acceleration during start of walking was difficult to define as it involves just leaning forward and then just putting the first foot in front of oneself to stop “the fall”. In addition, the changes in walking direction involves standing on one leg, leaning to the side, and then putting the other leg in the new direction, producing a quite sudden change in direction of the torso.

The results show that both the velocity of the person, the actual moving pattern and the drag coefficient of the body influences the air volume transfer between an anteroom and a patient room. The model highlights the sensitivity of air exchange on the movement of the person, showing that the behaviour of the health care worker can contribute to the transference of contaminants, although the door opening of a hinged door seemed to be the major contributor.

4 CONCLUSIONS

CFD simulations of air flows induced by opening and closing a hinged door between a patient room and an anteroom have been examined and the results compared to laboratory experiments.

The simulation and the experiments for “door only” movement, i.e. without the passage of a person through the door, matched each other closely. The simulation and experiment with a person exiting the room, showed some difference, but within quite acceptable limits (7 %) for using the model in future predictive analyses.

The simulations with a person entering the patient room from the anteroom showed larger deviations. The experiments were 27 % lower than the simulations. However, an alternative simulation with roughly 30 % lower “walking speed” gave the opposite effect – the simulation was 18 % below the laboratory. The conclusion from this comparison, is that the air exchange, at least walking in this direction, may be very dependent on the effective drag force produced by the person.
The simulation model will continue be used to investigate the sensitivity of walking speed and the movement of persons. The model will be used in future work in this research project, which will include pressure differences to create simplified solutions to limit contamination from patient rooms.

5 ACKNOWLEDGEMENTS

The project is funded by the Research Council of Norway.

6 REFERENCES

Tang, J W, Li, Y, Eames, I, Chan, P K S, and Ridgway, G L. (2006). Factors involved in the aerosol transmission of infection and control of ventilation in healthcare premises. Journal of Hospital Infection, 64, pp 100-114.

Hayden, C. S., Johnston, O. E., Hughes, R. T., & Jensen, P. A. (1998). Air volume migration from negative pressure isolation rooms during entry/exit. *Applied Occupational and Environmental Hygiene*, *13*(7), 518-527.

Saarinen, P, Kalliomäki, P, Koskela, H, and Tang, J W. (2017). Large-eddy simulation of the containment failure in isolation rooms with a sliding door – An experimental and modelling study. Building Simulation, https://doi.org/10.1007/s12273-017-0422-8.

Kalliomäki, P, Saarinen, P, Tang, J W, and Koskela, H. (2016). Airflow patterns through single hinged and sliding doors in hospital isolation rooms – Effect of ventilation, flow differential and passage. Building and Environment, 107, pp 154-168.

Harsem, T T, Lind, M C, Venås, B, Kalliomäki, P, and Koskela, H. (2018). Door and passage induced air exchange across hinged door, Proceedings. 14th SCANVAC Conference Roomvent & Ventilation 2018, 02-05 Jun 2018, Espoo, Finland.

Kanerva P, Koskela H, Saarinen P, Kalliomäki P, Mustakallio P (2018) Simulation of the airflow pattern from multi-nozzle diffuser. Roomvent & Ventilation 2018 conference, 2.-5.6.2018, Espoo, Finland.