Numerical Simulation of Airflow in Livestock Buildings with Radial Inlet

Bjarne Bjerg[1], Lars C. Sørensen[2]

[1] The Royal Veterinary and Agricultural University, Groennegaardsvej 2, DK-1870 Fredriksberg C, Denmark

[2] SKOV A/S, Hedelund 4, Glyngøre, DK-780 Roslev, Denmark, lcs@skov.dk

[1] bb@kvl.dk

ABSTRACT

The purpose of this work is to inquire the potentials of a simplified method of modelling radial inlets in numerical simulation of airflow in mechanically ventilated livestock rooms. This simplified method is based on simulation of an angle section of the entire air volume with starting point in the centre line of the inlet. Using symmetry planes as boundary condition on the sides of the angle section, and grids including only one cell in the width direction, it is possible to include three-dimensional geometry in simulations nearly equally uncomplicated as two-dimensional simulations.

Inside and close to the inlet the angle section method generates practically the same air velocities and pressure conditions as simulation of the entire room. At floor level the angle section methods generates air velocities that can be regarded as typical for the variety for velocities that occurs in a corresponding entire room simulation. Compared to entire room simulation the angle section methods reduce the required modelling time and calculation time with at least 80 and 95 percent, respectively, and consequently, it becomes practicable to inquire a relative large number of alternative solutions in order to develop suitable layout of radial inlet armatures. Further on the angle section method gives much better possibilities to increase grid density in order to investigate for possible grid dependency on simulation results.

Keywords: Radial Inlet, numerical simulation, CFD, livestock buildings, ventilation, airflow, Denmark.

1. INTRODUCTION

This paper describes Computational Fluid Dynamics (CFD) modelling methods used in development of a new armature for radial air inlet in livestock buildings. Radial air inlet armatures have been available since the nineteen sixties, but the use in livestock buildings has been limited. Compared with other types of inlets used in livestock houses, radial inlets have the advantages that they can be used in rooms without access to outer walls or attics, and they are suitable to take fresh air from above the roof in order to reduce infection risk. Due to these advantages, the Danish climate control company SKOV A/S decided to develop a new radial armature that fulfilled modern demands for capacity, energy efficiency, air distribution, control, and air velocity in the animal occupied zone. The basic ideas were to obtain large capacity (up to 18.000 m^3 h^{-1} per armature at -40 Pa) and high energy efficiency by smoothing the air passage through the armature, and to limit air velocity in the animal occupied zone by distributing the air inlet on a number of controllable air distribution plates at medium and high ventilation yield, see the left side figure 1. At winter condition - which means low ventilation demand and low inlet temperature - optimal air distribution requires a large inlet velocity and a large air momentum, which is obtained by leading the air through one narrow slot created by elevation of the bottom plate and successively the air distribution plate(s), see the right part of figure 1.

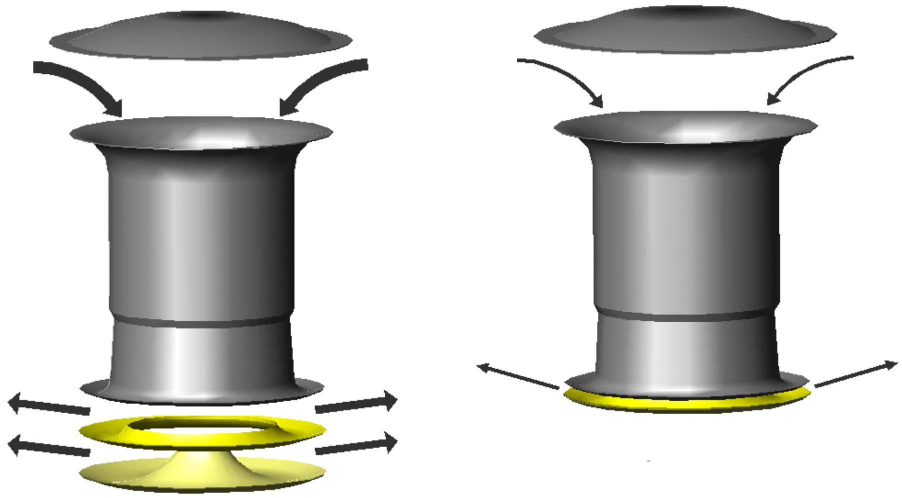

Figure 1. Radial air inlet armature. Summer adjustment to the left and winter adjustment to the right.

CFD have been used in studies on airflow in connection to livestock rooms for several years (Harral and Boon, 1997, Bjerg et al., 1999, 2000, 2002a., Sun et al., 2004, Wagenberg et al., 2004), but very few published works have focused on modelling of inlet armatures used in livestock ventilation. Bjerg et al (2002b) analyse deferent methods to model a prefabricated adjustable wall inlet used for livestock ventilation, but the radial inlet assumed in this work consist of a much more complicated geometry including double curved faces. And in addition the three-dimensional properties of the radial jet make it pointless to consider two-dimensional simulations. Therefore, in this work a simplified method is suggested and investigated. This simplified method is based on simulation of an angle section of the air volume with starting point in the centre line of the inlet, see figure 2. Using symmetry planes as boundary condition on the sides of the angle section, and grids including only one cell in the width direction, it is possible to include three-dimensional geometry in simulations nearly equally uncomplicated as two-dimensional simulations. The purpose of this work is to inquire the potentials of this angle section method in comparison with fully 3D simulations and measurements.

2. METHODS

A radial inlet armature - DA 50/920 from SKOV A/S - was installed above the centre of a 10 m long 8.6 m wide and 3.3 m high empty test room, see figure 3. During the measurement the inlet was equipped with one air distribution plate which divided the incoming air into two radial jets, see geometry A in figure 4. The heights of the bottom plate and the air distribution plate were controlled by a wire system (shown in figure 3) and both the distance between the bottom plate and the air distribution plate, and the distance between the air distribution plate and the fixed part of the inlet were adjusted to an opening height of 0.11 m. The diameter of both plates was 1.24 m and on the bottom plate the air was entering the room in a height of 0.52 m beneath the ceiling. The inner diameter of the air distribution plate was 0.75 m and at its narrowest spot the inner diameter of the shaft was 0.88 m.

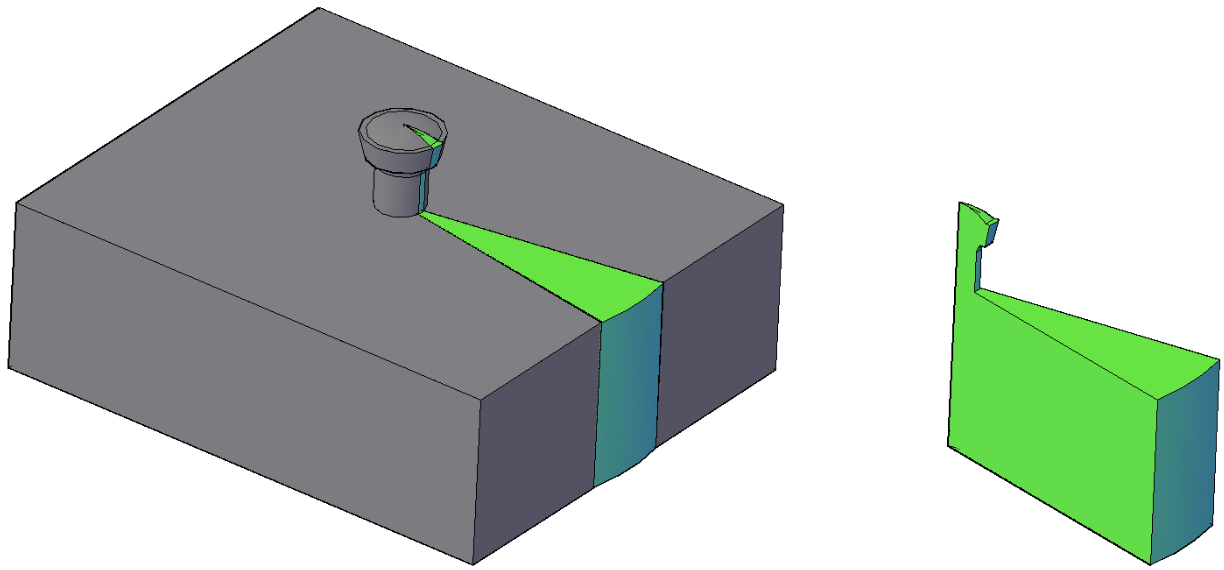

Figure 2. The entire air volume - to the left - and an angle section of the volume - to the right.

Two 0.6 m wide exhaust fans, Figure 3, located 2.1 metre above floor in one side wall was used to maintain a desired pressure drop over the inlet. In the measurements, a pressure drop of 20 Pa was measured with a Micatrone MicaFlex MF-PD pressuretransmitter. This corresponded to an air change of 3.67 m^3s^{-1}. The air exchange rate was calculated by knowing the capacity of the standard fan (tested at Research Centre Bygholm, Denmark). Room leakage at -20 Pa was measured and subtracted.

Air speeds were measured 0.1, 0.2 and 0.3 m above floor in the assumed return flow 1 m from one end wall, see figure 3. Dantec Low Velocity transducer 54R10 and Dantec Low velocity flow Analyzer 54N50 was used. Measurement frequency was1 Hz and the reported measuring values consists of average value of two measurement periods of 180 seconds. Difference between average air velocity in the two measuring periods were 0.02 - 0.04 ms^{-1}

CFD airflow modelling was performed with the commercial numerical simulations code Fluent 5 (Fluent Inc). The K-e turbulence model (Launder and Spalding, 1974), wall functions at surfaces, pressure inlet, velocity outlet boundaries, and steady state and isothermal conditions were assumed in all simulations. To include flow conditions into the inlet armature an air volume in front of the inlet was included in the simulations. The simulations were performed for either the entire room, or an angle section of the room, see figure 2.

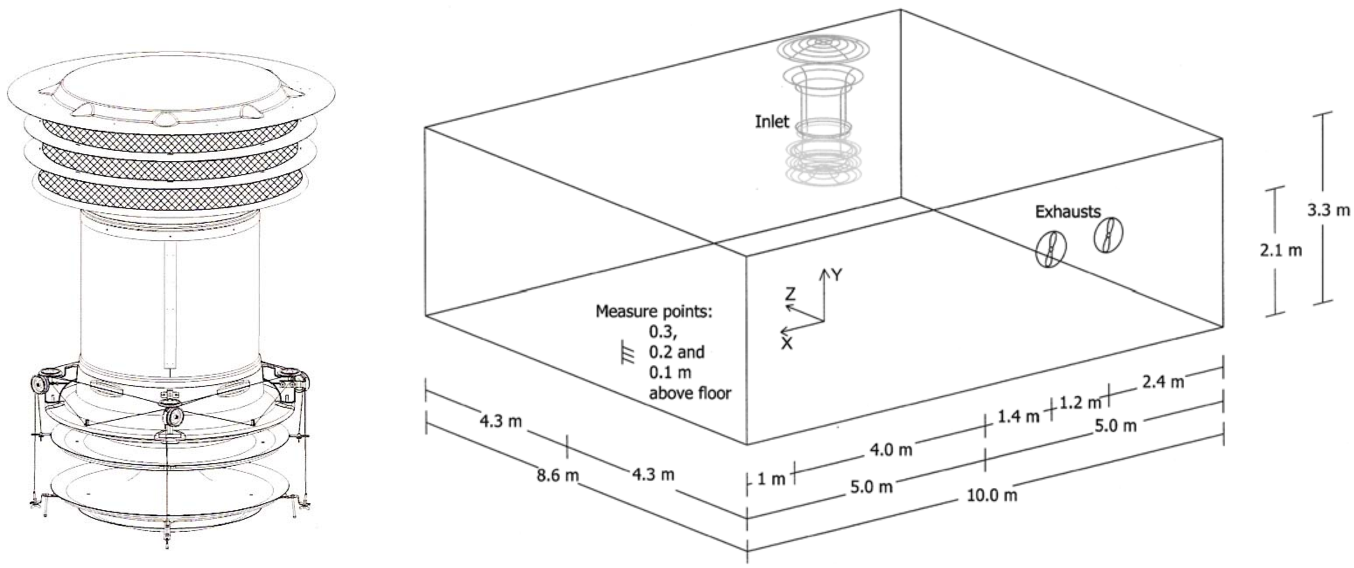

Figure 3. Radial inlet DA 50/920 and test room used in measurements and simulations.

The outlet needed to obtain mass balance in the angle section simulations were located on a four angular face in the middle of the cut face on one side of the model, see figure 5. This solution was chosen to reduce the influence of the outlet on the airflow. The remaining cut faces in the angle section simulations were modeled as symmetry boundaries.

Seven different inlet geometries were investigated in the simulations and entitled from A to G, see figure 4 and table 1.

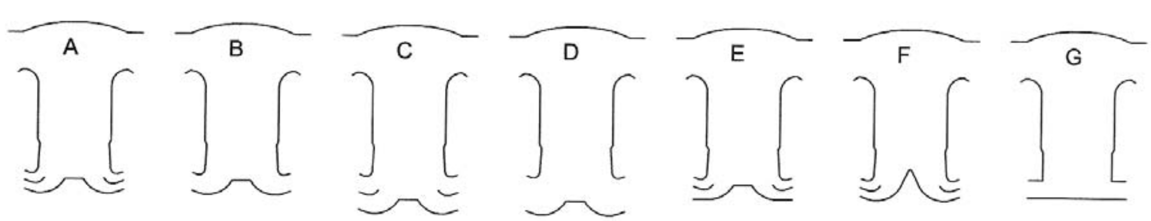

Figure 4. Cross section of examined inlet geometries.

The grid used in the simulation of the entire Room (A and B) consisted of about 190.000 hexagonal cells and in addition about 5000 tetrahedral cells where used in order to fit the geometry close to the outlets. Basically the flow domain used in angle section simulations consisted of a cut that included 10 degree on both sides of the positive x direction and a distance of 5 meter from centre to the side wall were assumed (the orientation of the used system of coordinates appear from figure 3). This scale and location of the angle section model ensured that the measuring points would be located in the centre plane of the model. The volume located closer than 0.04 metre from the centre line of the inlet where converted to a four angular shape in the x-z plane and this way it became possible to create less complicated grids that consisted of orthogonal cells only. Except small differences close to room centre, the grid used in simulation A1 was equal to the grid used in the y- and positive x direction in the simulations on the entire room. Grid on surfaces used in simulation A and A1 is shown in figure 5.

Table 1. Overview over simulation and measurement.

	Air Speed at x=4 and z=0, m/s height above floor (y), 0.10 m	0.20 m	0.30 m	Pressure drop over inlet, Pa
Measurement (geometry A)	0.53	0.55	0.56	20
Simulations with geometry as measurement				
A (entire room, 194.949 cells)	0.62	0.57	0.53	27
A1 (angle section, 3.938 cells, 1 cell in z direction)	0.87	0.79	0.71	31
A2 (angle section) 15.620 cells, 1 cell in z direction)	0.87	0.82	0.75	28
A3 (angle section) 43.318 cells, 11 cells in z direction)	0.91	0.83	0.75	31
Simulations without distribution plate				
B (entire room, 194.949 cells)	0.57	0.54	0.51	30
B1 (angle section, 3.938 cells, 1 cell in z direction)	0.80	0.73	0.67	35
Simulation with double distance between plates				
C1 (angle section, 3.938 cells, 1 cell in z direction)	0.67	0.61	0.56	25
As C1 but without distribution plate				
D1 (angle section, 3.938 cells, 1 cell in z direction)	0.60	0.56	0.50	26
Simulation with diverging angle of plates				
E1 (angle section, 3.938 cells, 1 cell in z direction)	0.71	0.65	0.59	27
Simulation with optimized bottom plate				
F1 (angle section, 3.938 cells, 1 cell in z direction)	0.87	0.79	0.71	31
Simulation with sharp edged inlet				
G1 (angle section, 3.597 cells, 1 cell in z direction)	0.85	0.78	0.71	60

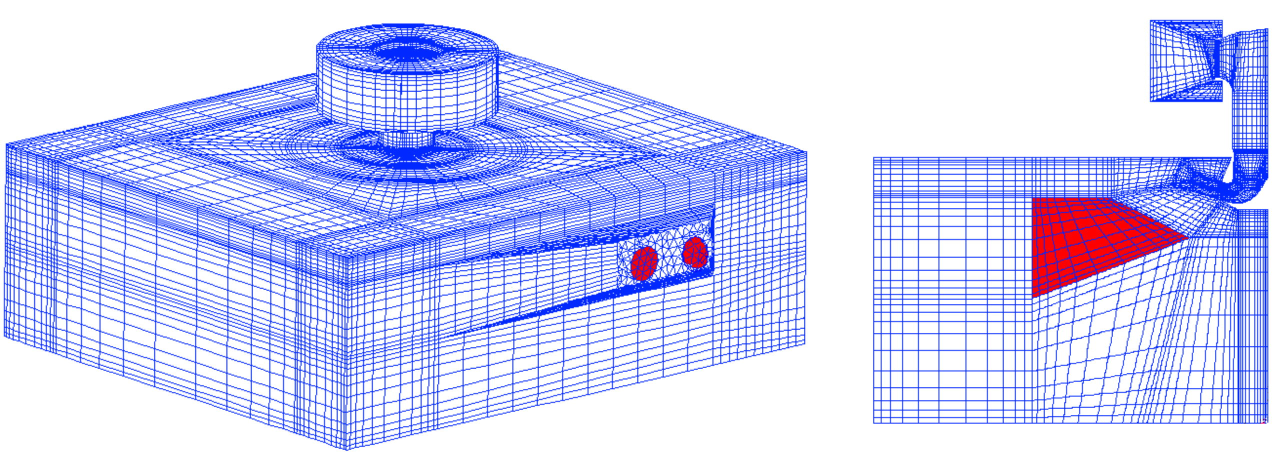

Figure 5. Grid on surfaces used in simulation A (Left) and A1 (right), red faces shows outlets.

To investigate possible grid dependency on simulations results the grid density in both x- and y-direction where doubled in simulation A2 compared to simulation A1. In the angle section simulation was generally use one cell in the z direction only, but a single exception was made in simulation A3 where 11 cells in the z-direction were assumed. Simulation B and B1 were equal to A and A1, respectively, except that the air distribution plate were excluded. Specifications of simulations C1 to G1 appears from table 1 and figure 4.

All simulation converged into steady state solution and was carried out on a 2.7 GHz double processor Windows XP system.

3. RESULTS AND DISCUSSION

Measured and simulated pressure drop over the inlet and air velocity 0.1, 0.2, and 0.3 m above the floor at x=4 m and z=0 m is included in table 1. In figure 6 is shown simulated air velocity distribution in the entire room (Simulation A) 0.2 m above floor and in vertical planes at z=0 m and z= -0.8 m.

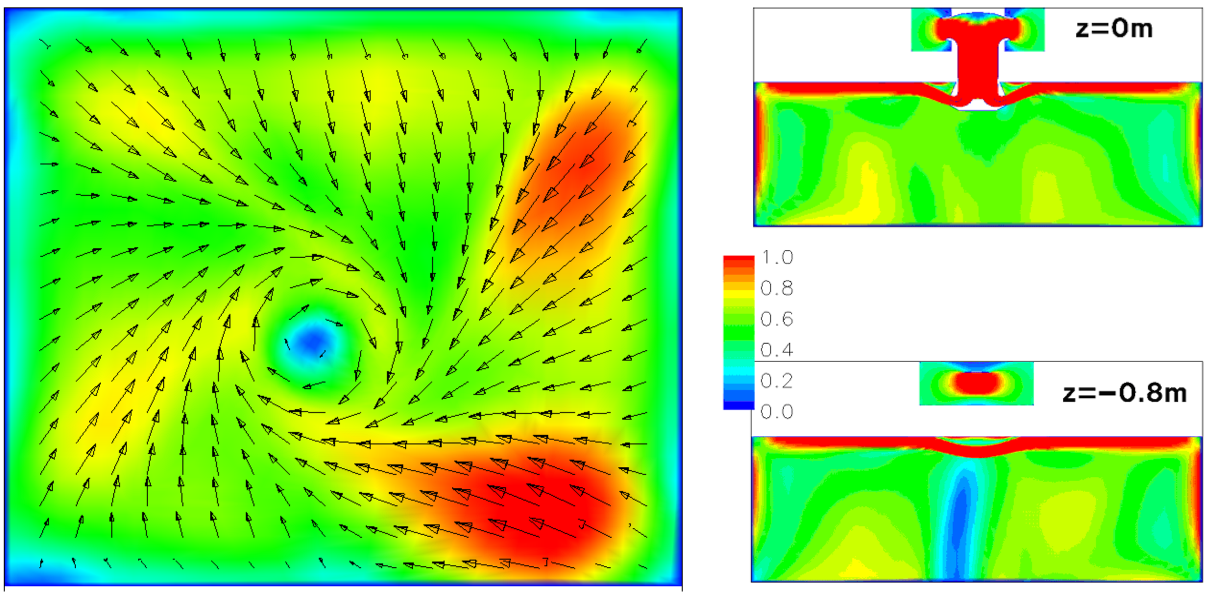

Figure 6. Simaulted air velocity distribution in the entire room (Simulation A) 0.2 m above floor (picture to the left) and in vertical planes at z=0 m and z= -0.8 m. Red colours shows air velocities above 0.8 ms^{-1}.

The pictures in figure 6 shows that the incoming air follows the ceiling and sides walls and further on the floor where it, to some extends, turns of to the left before it reaches a circulating up going stream located with a centre about 0.8 m from the centre of the room.

Figure 7 shows air speed 1 m from side walls and 0.2 m above floor in measurement and in simulation A and B. From the figure it appears that the simulated air speed in simulation A was very close to the measurement 0.2 above the floor, and that the air speed pattern 1 metre from the side walls was very similar in the simulations with and without guiding plate – simulation A and B respectively. The guiding plate increased the air speed with between 0 to 15 percent at the profiles shown in figure 7. In the vertical profiles of air velocity and turbulence (turbulent kinetic energy) at two distances from the inlet is shown in figure 8.

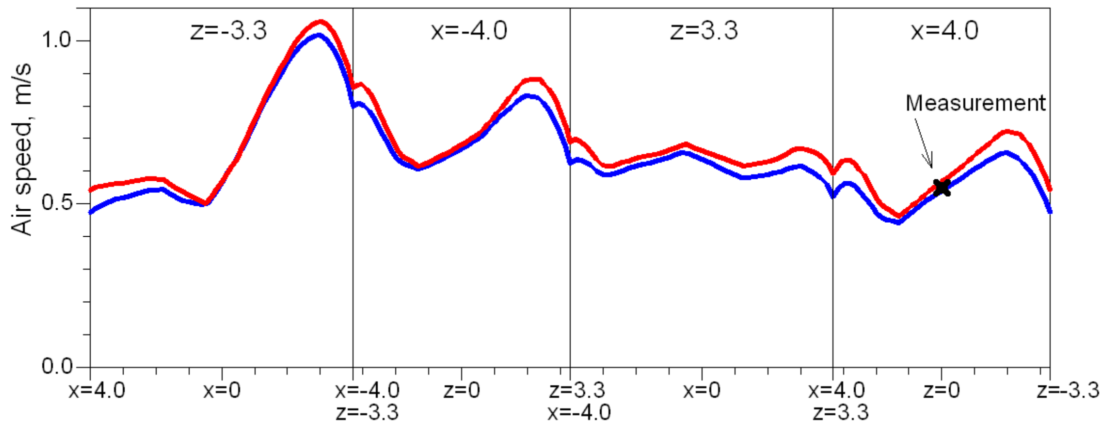

Figure 7. Air speed 1 metre from side walls and 0.2 m above floor in measurement and in simulation A (red) and B (blue).

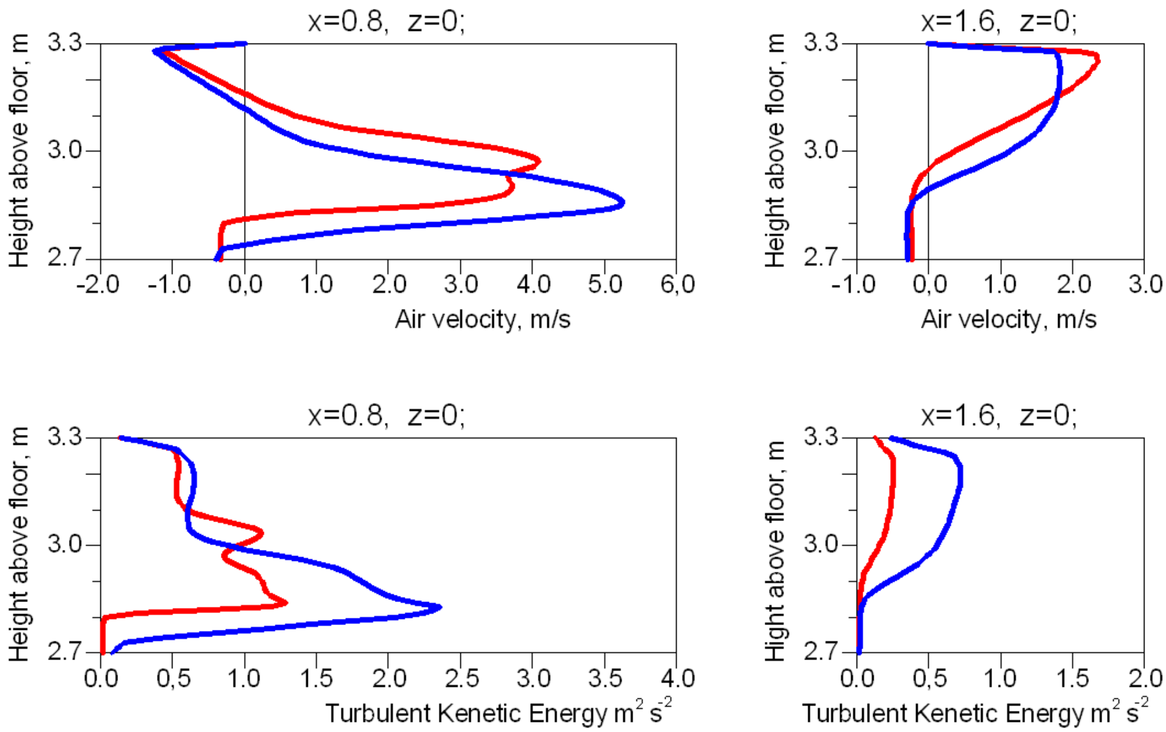

Figure 8. Vertical profiles of air velocity and turbulence (turbulent kinetic energy) at 0.8 m and 1.6 m from the inlet centre line. Simulation A shown as red line and simulation B shown as blue line.

The profiles at X=0.8 are located 0.18 m from the edge of the inlet and from figure 8 it appears that the guiding plate reduces the highest values of both air velocity and turbulence. The upper right graph in figure 8 shows that the guiding plates cause a slower air speed decrease with increased distance from the inlet. This phenomenon can be explained by less mixing of air between jet and surroundings due to the lower turbulence generation when the guiding plate is used. Further on the simulations showed that the guiding plate reduced the pressure drop over the inlet with about 10 percent corresponding to an increased flow rate of about 5 percent if the pressure drop were held at equal level.

The entire room simulation (A) and the angle section simulation (A1) resulted in very similar air velocity distribution inside and close to the inlet, see figure 9.

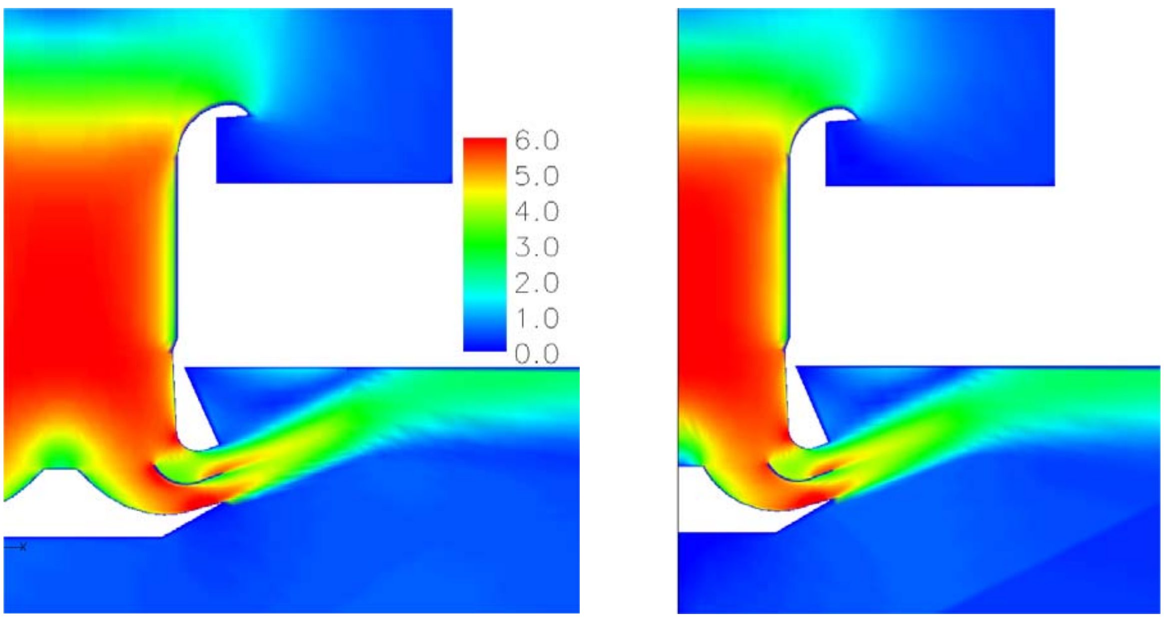

Figure 9. Simulated air velocity (ms^{-1}) inside and close to the inlet. Entire room simulation to the left (Simulation A), and the angle section simulation to the right (Simulation A1).

Figure 10 compares simulated velocities at x=0.8 for simulations of the entire room (Simulation A) and of an angle section with either 1 or 11 cells in the z-direction (simulation A1 and A3 respectively).

The figure confirms that the difference between the entire room simulation and the angle section simulation is small but it also reveals that the simulated air velocity direction is different just beneath the jet, which may be related to the assumed exhaust condition in angle section simulation. In addition figure 10 shows the influence of increasing the number of cells (from one to 10) transverse to inlet direction had an insignificant influence on the results of the angle section simulation only.

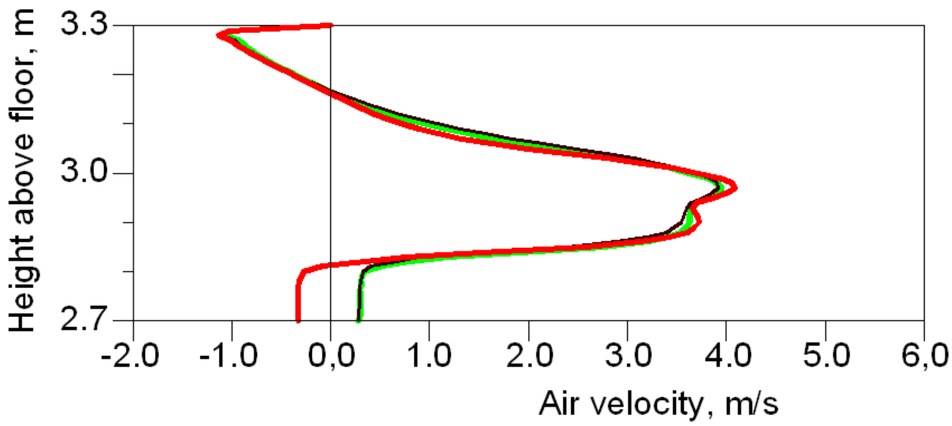

Figure 10. Air velocity in a vertical profile 0.8 m from the inlet centre line. Simulation A (entire room) shown as read line, simulation A1 (angle section with one cell in z direction) shown as green line and simulation A3 (angle section with 11 cell in z direction) shown as black line.

At the measuring point at 0.2 m above floor the angle section simulation resulted in air velocity of 0.79 ms^{-1} which is nearly 40 percent higher than the value in the entire room simulation. But if the angle section simulation is regarded as typical for all flow directions it should be compared with all air velocities simulated 1 m from side wall (and 0.2 m from floor) in simulation A. From figure 7 it appears that this value is varying between 0.45 and 1.05 ms^{-1}, and related to that the result of the angle section simulations seams satisfactory.

Comparison of results from Simulation A1 and A2 showed that a significant increase of grid density had a minor influence on simulated air velocities above floor and pressure drop over inlet, see table 1. This observation indicates that it might be relevant to make simulation with even finer grids, which easily could be done with the angle section method but would be very difficult in an entire room simulation.

Simulation C1 and D1 showed that a larger inlet area reduced pressure drop over the inlet and air speed at floor level. Generally, the effect of the guiding plate was similar to the effect it had at the smaller inlet area.

The original assumption of introducing air distributions plates was that it should fulfil a demand for reducing air speed in occupied zone at high ventilation rate. In geometries A to D (see figure 4) the guiding did not have that influence, but comparison of simulation E1 with Simulation A1 and B1 shows that a reduced inlet angle at the bottom plate can reduce air speeds at floor level.

Analyses of turbulence distribution in the A geometry (see figure 4) showed high turbulence level above the horizontal central area of the bottom plate. Simulation F1 showed that a cone shaped fill out of this volume significantly reduced the highest turbulence values inside the inlet but the influence on pressure drop over the inlet and on the air velocities at floor level were insignificant.

Simulation G1 illustrates that a sharp edged profile require a significant higher pressure drop to allow the same flow through the inlet. Because the airflow in simulation G1 is concentrated on the lower part of the inlet area the air velocity is up to about 7.9 ms^{-1} when it leaves the inlet.

This is 22 percent higher than in simulation A1 were the airflow is more evenly distributed over the inlet area. But due to high turbulence generation in the sharp edged geometry air velocity at floor level was lower in simulations G1 than in simulation A1 (see table 1)

The angle section methods were suggested in order to reduce modelling and calculation time compared with entire room simulation. Roughly estimated the method reduced modelling time with at least 80 percent. The entire room simulations required about 4000 iterations corresponding to a calculation time of about 6 hours in order to obtain convergence. In section angle simulations (A1 to G1) 400 iterations where sufficient which were reached after 2 minutes of calculation time only.

4. CONCLUSION

Due to low time consumption the angle section method used in this study makes it practicable to inquire a relative large number of alternative solutions in order to develop suitable layout of radial inlet armatures. Close to the inlet the angle section methods generates practically the same air velocities as simulation in the entire room. At floor level the angle section methods generates air velocities that can be regarded as typical for the variety of velocities that occurs in a corresponding entire room simulations. Further on the angle section method gives much better possibilities to increase grid density in order to investigate for possible grid dependency on simulation results.

The CFD analyses of 7 inlet profiles illustrated the potentials of optimising the inlet layout in order to reduce energy and material consumption and optimizing control of airflow at floor level.

The air distributions plate used in the studied armature reduced turbulence generation in the inlet, and consequently, the required pressure drop needed to ensure a certain airflow through the inlet were also reduce. Due to reduced turbulence in the incoming air the air distributions plate increased air speed at floor level. A changed inlet direction on the bottom plate would make it possible to reduce air velocity at floor level at high ventilations rate, without increasing the pressure drop over the inlet.

5. LITERATURE

Bjerg, B., Morsing, S., Svidt, K., Zhang, G., 1999. Three-dimensional air flow in a livestock test room with two-dimensional boundary conditions. Journal of Agricultural Engineering Research. 74(3): 267-274.

Bjerg, B, Svidt, K, Zhang, G, Morsing, S. 2000: The Effects of pen partitions and thermal pig simulators on airflow in a livestock test room. Journal of Agricultural Engineering Research. 77, (3): 317-326.

Bjerg, B., K. Svidt, G. Zhang, S. Morsing, and J.O. Johnsen. 2002a. Modelling of air inlets in CFD prediction of airflow in ventilated animal houses. Computers and Electronics in Agriculture. 34: 223-235 .

Bjerg, B., K. Svidt, S. Morsing, G. Zhang and J.O. Johnsen. 2002b. Modelling of a Wall Inlet in Numerical Simulation of Airflow in Livestock Buildings. Agricultural Engineering

International: the CIGR Journal of Scientific Research and Development. Manuscript BC 01 001. Vol. IV. March, 2002.

Harral, B. B. and C. R. Boon. 1997. Comparison of predicted and measured air flow pattern in a mechanically ventilated livestock building without animals. Journal of Agricultural Engineering Research. 66: 221-228.

Launder, B., D. B. Spalding. 1974. The numerical computation of turbulent flows. Computer Methods of applied Mechanical Engineering, 3, pp 267-274.

Sun, H., H. Keener, W. Deng and F. Michel, Jr. 2004. Development and validation of 3-D CFD models to simulate airflow and Ammonia Distribution in a high-rise hog building during simmer and winter conditions. Agricultural Engineering International: the CIGR Journal of Scientific Research and Development. Manuscript BC 04 004. Vol. VI. December, 2004.

Wagenberg, A. V. van, B. Bjerg and G. Bot. 2004. Measurement and Simulation of Climatic Conditions in the Animal Occupied Zone in a Door Ventilated Room for Piglets. Manuscript BC 03 020. Vol. VI. April 2004.