Wageningen UR Livestock Research

Partner in livestock innovations

Report 664

Use of computational fluid dynamics (CFD) modelling to improve tracer gas techniques in very open naturally ventilated livestock buildings

September 2013

LIVESTOCK RESEARCH
WAGENINGEN UR

LIVESTOCK RESEARCH

WAGENINGEN UR

Colophon

Publisher
Wageningen UR Livestock Research
P.O. Box 65, 8200 AB Lelystad
Telephone +31 320 - 238238
Fax +31 320 - 238050
E-mail info.livestockresearch@wur.nl
Internet http://www.livestockresearch.wur.nl

Editing
Communication Services

Abstract
To identify the possibilities of CFD modelling of naturally ventilation patterns and emission levels a literature review was done. Although CFD results are not always validated the technique itself can be of use to predict ventilation and improve emission measurements.

Keywords
Computational Fluid Dynamics (CFD), natural ventilation, dairy housing, ventilation rate, modelling

Reference
ISSN 1570 - 8616

Author(s)
H.J.C. van Dooren
A. Sapounas

Title
Use of computational fluid dynamics (CFD) modelling to improve tracer gas techniques in very open naturally ventilated livestock buildings

Report 664

Report 664

Use of computational fluid dynamics (CFD) modelling to improve tracer gas techniques in very open naturally ventilated livestock buildings

H.J.C. van Dooren
A. Sapounas

September 2013

Dit onderzoek is uitgevoerd binnen het Beleidsondersteunend onderzoek van het ministerie van Economische Zaken, BO-20-004-021

Foreword

One of the possibilities to evaluate and improve sampling strategy in very open naturally ventilated (dairy) buildings is to use computational fluid dynamics (CFD) techniques to model air flow pattern in and around buildings. CFD are extensively used for all kind of purposes but use in naturally ventilated buildings for livestock is not so common. It is for that reason that the Dutch ministry of Economic Affairs commissioned this research to identify existing research and research groups that has been done or are active in this field, to summarize the existing CFD-models and their quality and to list the possible further possibilities for CFD modelling.

The authors

Summary

Ventilation rates from naturally ventilated livestock buildings are commonly measured using CO_2 produced by animals or SF_6 injected in the building acting as a tracer gas. Crucial assumption for this method is that the tracer gas is completely mixed through the whole building and behaves comparable with the target gasses like ammonia (NH_3). In very open livestock buildings like modern dairy barns this assumption is under pressure due to the relatively increasing share of cross ventilation to the total ventilation rate. Analyses of emission measurements using CO_2 as a tracer gas have shown that with the used sampling pattern the variation between farms is reduced to an acceptable level. The analyses also showed that more emphasis should be paid to the accuracy and precision of the measurements.

The use of models can help to gain insight in ventilation rates, flow patterns and internal concentration differences at different outdoor conditions.

Air flow patterns and ventilation rates are commonly studied using Computational Fluid Dynamic (CFD) techniques. This CFD software (models) provides commonly 3D airflow patterns. Results however are not always validated and also the representativeness of the outcomes in other barns or under other circumstances is not always clear. There is therefore a need for an overview of suitable and used CFD models or calculations and the possibilities to use these model in research on emissions from naturally ventilated buildings.

Objectives of this study are:

- To identify CFD model calculations of air flow patterns in or emissions from naturally ventilated barns that preferably have been validated
- To investigate the possible use in modern, very open dairy barns
- To investigate the options to validate (if not already done) CFD calculations in a cost effective way.

The following activities were performed:

- Identification of relevant literature concerning CFD and naturally ventilated livestock housing with emphasis on dairy barns.
- Identification of active research groups in the field of application of CFD calculation in livestock housing, with emphasis on naturally ventilated housing.

CFD models have been extensively used to investigate the natural ventilation of livestock buildings, specially the last three years. There are many research groups worldwide dealing with this scientific topic. The most active one is the research group from the Department of Engineering in the University of Aarhus in Denmark. The main research targets of these groups are to investigate the emission of pollutants from livestock buildings and to improve the ventilation performance of these buildings by suggesting design modifications.

In most of the studies found in the literature the validation of CFD models has been carried out using experimental data from wind tunnels, which means that almost only scale livestock buildings have been considered. In general the agreement between measured data and computational results is good. There is not any information if the conclusions provided by the research studies, have been used by the companies to improve the design of their buildings.

Till now there are few handbook with guidelines and best practices for developing CFD models for modelling the air flow around and inside buildings. There are not any particular guidelines for livestock buildings, even if the last years few review papers have been published. This indicates that each research group uses its own approach to simulate the environment of livestock buildings.
There are many CFD commercial codes available. The most common one is the ANSYS Fluent. Since the CFD models are totally deterministic models the same numerical methods, sub models and boundary conditions should provide the same results for a given computational domain and the choice for one or the other commercial codes could not make any difference.

Tracer gas techniques have been used to calculate the ventilation rate of livestock buildings but CFD models have not been used in advance to design the experimental setup. Usually the CFD models are used to validate the experiments and not to design them. CFD models can be a powerful tool to effectively design experiments with tracer gas or to setup sampling strategies. Validated CFD models can be used for further research and as tool to improve the design of current livestock buildings. There is a necessity for developing a methodology (protocol) for CFD simulation of livestock buildings. By using the same methodology the results obtained by different studies can be compared. Finally the CFD can be used to introduce new quality parameters for livestock buildings like those related to the spatial uniformity of air temperature, gas concentration of air velocity (uniformity index). The developing of these indexes will be useful both for the companies, farmers and research.

Referring to the objectives it can be concluded that several CFD-models are available for livestock building but only a few for naturally ventilated dairy barns. The majority of these studies has only be validated with data from windtunnel experiments. None of them appeared to be validated with real scale data. Validation with these data can be costly if new equipment has to be bought and installed. Combination with existing projects like the one on comparing the SF_6 traces method with the CO_2 balance method is possible and less costly. This can only be a validation on gas concentration predictions as no air velocity equipment is installed. It is therefore recommended to adapt the already exciting CFD model for dairy barns (Sapounas et al., 2012) and combine it with the data collected in the this project on tracer gas techniques.

Samenvatting

Ventilatiedebiet van natuurlijk geventileerde stallen wordt meestal bepaald door gebruik te maken van tracergas technieken of de zogenaamde CO_2 balans methode. Cruciale aanname daarbij is dat het tracergas en doelgas zich gelijk gedragen en tracergas volledig gemixt is in de stal. In zeer open natuurlijk geventileerde (melkvee)stallen staat met name dat laatste uitgangspunt onder druk.

Uit analyse van eerdere emissiemetingen waarbij CO_2 als tracergas is gebruikt blijkt dat de bij deze methode gehanteerde bemonsteringsstrategie voldoende ruim is opgezet om de variatie tussen bedrijven voldoende te reduceren. Meer aandacht zou moeten besteed worden aan de precisie (de systematische juistheid) en de nauwkeurigheid van de meting (de meetfouten met een toevalskarakter). Dit geldt met name voor de in de melkveehouderij in toenemende mate in gebruik zijnde zeer open stallen. De kwetsbaarheid van de huidige tracergasmethodes is hier vooral gelegen in onvoldoende menging van tracergas en de onnauwkeurige meting van de gasverhoudingen als gevolg van sterke dwarsventilatiepatronen.

Het modelleren van luchtstromingen kan helpen om inzicht te krijgen in het ventilatiedebiet, stromingspatronen en concentratieverschillen bij verschillende weersomstandigheden.

Luchtstromingspatronen worden meestal gemodelleerd door gebruik te maken van computational fluid dynamic (CFD) technieken. Deze worden buiten de landbouw voor allerlei doeleinden ingezet maar zijn binnen de veehouderij nog niet gebruikelijk. Daarnaast heeft een goede validatie van modeluitkomsten aan meetwaarden uit de praktijk meestal niet plaatsgevonden en is de geldigheid van een model voor andere omgevingen dan waar het voor ontwikkeld is onbekend. Er is daarom behoefte aan een overzicht van beschikbare CFD modellen en de geschiktheid daarvan voor toepassing in natuurlijk geventileerde omgevingen.

Deze studie heeft laten zien dat CFD modellen de laatste jaren steeds meer gebruikt worden om inzicht te krijgen in natuurlijke ventilatie. Verschillende onderzoeksgroepen richten zicht op dit onderwerp. De meest actieve daarvan is die verbonden aan de Aarhus universiteit in Denemarken. Belangrijkste doel van deze en andere groepen is het inschatten van gasvormige emissies en het verbeteren van ventilatie.

In de meeste literatuurbronnen heeft validatie van het CFD-model plaatsgevonden met behulp van gegevens uit windtunnelexperimenten. Dat betekent dat alleen schaalmodellen van stallen zijn gebruikt. Validatie met behulp van deze data laat meestal een goede overeenkomst zien tussen experimentele gegevens en modeluitkomsten. Het wordt niet duidelijk uit betreffende studies of de uitkomsten van deze modellering ook daadwerkelijk gebruikt zijn voor de uiteindelijke doelen: inschatting emissie en verbeteren ventilatie.

Er is geen algemene richtlijn of aanpak beschreven voor toepassing van CFD voor stallen. De beschikbare handboeken met richtlijnen en ‘best practices’ zijn algemeen van aard. Dat betekent dat de actieve onderzoeksgroepen op dit gebied elk hun eigen aanpak hanteren.

Software voor CFD berekeningen is ruim beschikbaar zowel als freeware als in de vorm van commerciële producten. Het meest bekende en gebruikte pakket is ANSYS fluent maar omdat de onderliggende numerieke methoden gelijk zijn zouden verschillende pakketten bij dezelfde uitgangspunten een vergelijkbaar resultaat op moeten leveren. Verschillen bestaan wel op het gebied van gebruiksvriendelijkheid en gebruikte submodellen voor, bijvoorbeeld, turbulentie.

Er zijn geen studies bekend waarin CFD modellen gebruikt zijn om experimenten met tracergas te optimaliseren. Uitkomsten CFD modellen worden meestal gebruikt om te vergelijken met experimentele gegevens maar niet om de opzet van die experimenten te beïnvloeden. Dit kan echter juist wel een krachtige toepassing van CFD technieken zijn hoewel de CFD modellen dan wel voldoende gevalideerd moeten zijn.

Gebruik van CFD modellen biedt ook de mogelijkheid om nieuwe parameters te introduceren die de ruimtelijke variatie in een gebouw ten aanzien van temperatuur, luchtvochtigheid, luchtsnelheid of gasconcentraties beschrijven.

Om resultaten van verschillende studies onderling vergelijkbaar te maken is er wel behoefte aan een afstemming van aanpak en methode.

Teruggrijpend naar de doelen van deze studie kan geconcludeerd worden dat CFD modellering voor natuurlijk geventileerde (melkvee)stallen relatief nieuw is, dat de beschikbare modellen alleen gevalideerd lijken te zijn op basis van windtunnelexperimenten en dat daarom validatie op basis van gegevens uit echte stallen nog steeds noodzakelijk is. Deze validatie kan kostbaar zijn als speciaal hiervoor meetapparatuur aangeschaft en geïnstalleerd moet worden.

Combinatie met lopende onderzoeken zoals het project waarbij de SF_6 tracergasmethode vergeleken wordt met de CO_2 balansmethode bij zeer open melkveestallen is daarom kosteneffectiever. Nadeel daarvan is wel dat alleen gasconcentratie gegevens voor de validatie gebruikt kunnen worden en geen data over luchtsnelheid in de stal beschikbaar zijn. Een combinatie met al bestaande en voor dit doel aangepast CFD modellen voor natuurlijk geventileerd melkveestallen (Sapounas et al., 2012) heef t uit kostenoverweging de voorkeur.

Table of contents

1 Introduction

Ventilation rates from naturally ventilated livestock buildings are commonly measured using CO_2 produced by animals or SF_6 injected in the building acting as a tracer gas. The source strength is either controlled by a mass flow controller (SF_6) or calculated based on CIGR equations for heat and CO_2 production of livestock (Pedersen and Sällvik, 2002). Crucial assumption is that the tracer gas is completely mixed through the whole building and behaves comparable with the target gasses like ammonia (NH_3), methane (CH_4) or nitrous oxide (N_2O). In very open livestock buildings like modern dairy barns this assumption is under pressure due to the relatively increasing share of cross ventilation to the total ventilation rate. The validity of the assumptions can be checked using two traces gasses at the same time or by increasing the number of sampling points inside the barn. Both options are under research but are too costly to implement in more dairy barns.

Analyses of emission measurements using CO_2 as a tracer gas have shown that with the used sampling pattern (Ogink et al., 2011) the variation between farms is reduced to an acceptable level. The analyses also showed that more emphasis should be paid to the accuracy and precision of the measurements.

Model calculations of air flow can also help to improve a sampling strategy for gas concentrations in dairy barns. Air flow patterns and ventilation rates are commonly studied using Computational Fluid Dynamic (CFD). This CFD software (models) provides commonly 3D airflow patterns. However, results are not always validated and the representativeness of the outcomes in other barns or under other circumstances is not always clear. There is therefore a need for an overview of suitable and used CFD models or calculations and the possibilities to use these models in research on emissions from naturally ventilated buildings.

1.1 Objectives

Objectives of this study are:

- To identify CFD model calculations of air flow patterns in or emissions from naturally ventilated barns that preferably have been validated
- To investigate the possible use of CFD models in modern, very open dairy barns
- To investigate the options to validate (if not already done) CFD calculations in a cost effective way.

1.2 Activities and bookmarks

The following activities were performed:

- Identification of relevant literature concerning CFD and naturally ventilated livestock housing with emphasis on dairy barns.
- Identification of active research groups in the field of application of CFD calculation in livestock housing, with emphasis on naturally ventilated housing.

In chapter 2 the background of CFD modelling is described and the different elements and steps to come to model calculations are explained. Chapter 3 describes the possible use of CFD in naturally ventilated livestock buildings for the early mentioned purposes. In chapter 4 an overview of relevant literature, software and research groups is given. In chapter 5 some conclusions are drawn and possible further steps are recommended.

2 Background of Computational Fluid Dynamics

2.1 Computational Fluid Dynamics in general

Computational Fluid Dynamics (CFD) aims to obtain numerical solution to fluid flow problems using a computer. The increase in computer speed and memory availability made is possible to obtain solutions to many flow problems. A variety of reasons can be mentioned for the increased importance of CFD simulation techniques in recent years:

- The need to forecast performance
- The wish to prevent costly or impossible experiments
- The desire for increased insight

The dynamics of fluid flow is governed by a set of equation describing the conservation of mass, energy and momentum (Navier-Stokes). These equations form a system of coupled non-linear partial differential equations (PDEs). Because of the non-linear terms in PDEs, analytical methods can yield very few solutions. In general, closed form analytical solutions are possible only if these PDEs can be made linear, either because non-linear terms naturally drop out or because non-linear terms are small compared to other terms so that they can be neglected. If the nonlinearities in the governing PDEs cannot be neglected, which is the situation for most engineering flows, numerical methods are needed to obtain the solutions.

With CFD the differential equation governing the fluid flow are replaced with a set of algebraic equations. This process is called discretization. These algebraic equations can be solved with the aid of a digital computer to get an approximate solution. The best-known discretization methods used in CFD are Finite Difference Method (FDM), Finite Volume Method (FVM), Finite Element Method (FEM), and Boundary Element Method (BEM).

The main advantages of CFD over experiments are (Versteeg and Malalasekera, 1995):

- Reduction of lead time and costs of new designs
- Ability to study systems where controlled experiment are difficult of impossible or under conditions that are hazardous or beyond normal conditions
- Practical unlimited level results and detail.

2.2 Elements of Computational Fluid Dynamics

There are essentially three elements to every CFD simulation process: pre-processing, solving and post-processing.

Preprocessing
This is the first step in building and analysing a flow model. It includes building the model within a computer aided design (CAD) package (the *domain*), creating and applying a suitable computational *mesh* or *grid*, selection of fluid properties and phenomena that need to be modelled and entering the flow *boundary conditions*. The accuracy of the solution is defined by the number of cells in the grid (mesh). The larger the number of cells the better the accuracy but also the higher the necessary amount of calculation time. Optimal grids are therefore non uniform: finer at points of interest of in situations where large variations are expected (Versteeg and Malalasekera, 1995). There are large numbers of commercial CAD packages for creating complex 3D geometries.

Solution
The CFD solver does the flow calculations and produces the results by solving the discretised form of governing equation. This stage needs clear understanding of flow physics involved in the problem like phenomena related with heat transfer, mass transfer, multispecies flow, multiphase flow, reacting flow, turbulence, radiation, etc. In most of the commercial CFD packages there are different options regarding the numerical methods can be used to solve the governing equations. As a solution of all the governing equations, the flow parameters, like velocity, pressure, density, temperature, concentration are calculated at each grid point.

Postprocessing
The enormous amount of data generated by CFD solver cannot be analysed by just looking at the numerical values. The final step in CFD analysis involves the organization and interpretation of the predicted flow data and the production of CFD images and animations. Different post-processing tools like colour plots, contour plots and vector plots are used to go into the problem. The interpretation of these results plays important role in determining the performance of any system being studied.

2.3 The Computational Fluid Dynamics process

The main stages of a CFD analysis process are:

- Problem statement (information about the flow)
- Mathematical model (define the governing equations and numerical method)
- CAD model (define the geometry of the solution domain)
- Mesh generation (define methods, schemes, and boundary zones)
- Turbulence model definition (for turbulent flows)
- Definition of boundary conditions
- Time discretization (for unsteady flows)
- Iterative solver (discrete function values)
- CFD software (implementation, debugging)
- Simulation run (parameters, stopping criteria)
- Postprocessing (visualization, analysis of data)
- Verification model (validation / adjustment)

Developing a CAD model and during mesh generation it is good to bear in mind that the computing times for a flow simulation depend on:

- The choice of numerical algorithms and data structures
- Linear algebra tools, stopping criteria for iterative solvers
- Discretization parameters (mesh quality, mesh size, time step)
- Cost per time step and convergence rates for outer iterations
- Programming language (most CFD codes are written in Fortran)
- Hardware, vectorization, parallelization

The quality of simulation results depends on:

- The mathematical model and underlying assumptions
- Approximation type, stability of the numerical scheme
- Mesh, time step, error indicators, stopping criteria

Whether or not the results of a CFD simulation can be trusted depends on the degree of uncertainty and on the cumulative effect of various errors:

- Uncertainty is defined as a potential deficiency due to the lack of knowledge (turbulence modelling is a classic example)
- Error is defined as a recognizable deficiency due to unknown other reasons

2.4 Verification and validation of Computational Fluid Dynamics results

Results obtained from CFD models, apart from those that have been validated, always have to be considered with scepticism. Verification and validation of the results are very important process steps. The goal of verification and validation is to ensure that the CFD code produces reasonable results for a certain range of flow problems.

Verification aims to looking for errors in the implementation of the models by:

- Examine the computer programming by visually checking the source code, documenting it and testing the underlying subprograms individually
- Examine iterative convergence by monitoring the residuals, relative changes of integral quantities and checking if the prescribed tolerance is attained
- Examine consistency (check if relevant conservation principles are satisfied)

- Examine grid convergence: as the mesh and/or and the time step are refined, the spatial and temporal discretization errors, respectively, should asymptotically approach zero (in the absence of round-off errors)
- Compare the computational results with analytical and numerical solutions for standard benchmark configurations (representative test cases)

Validation aim to checking if the model itself is adequate for practical purposes:

- Verify the code to make sure that the numerical solutions are correct.
- Compare the results with available experimental data (making a provision for measurement errors) to check if the reality is represented accurately enough.
- Perform sensitivity analysis and a parametric study to assess the inherent uncertainty due to the insufficient understanding of physical processes.
- Try using different models, geometry, and initial/boundary conditions.
- Report the findings, document model limitations and parameter settings.

3 Computational Fluid Dynamics and natural ventilation

3.1 Natural ventilation and tracer gas techniques

Natural ventilation itself is not new. It is only in the past 150 years that mechanical ventilation has been used (Etheridge, 2011). Before, all enclosures occupied by humans and/or animals were naturally ventilated. The beginnings of natural ventilation design can perhaps be considered as the time when these enclosures started to become purpose-built. A detailed overview about natural ventilation of buildings is given by Etheridge (2011).

The mechanism of natural ventilation depends on wind effects, thermal buoyancy and the combination of both wind and buoyancy forces. Thermal buoyancy is less important than the wind effect for natural ventilation, especially in hot weather, wind speed and wind direction are the dominant factors for wind-induced effects (De Jong and Bot, 1992; Yu et al., 2002). Miguel et al. (2001) indicated that a full understanding of the relationship between wind characteristics (wind speed and wind direction) and ventilation characteristics (dimensions, inlet and outlet design, etc.) are required to achieve sufficient natural ventilation. To ensure adequate ventilation, it is important that the building is designed to:

- Remove excess heat, water vapor and gases
- Provide a uniform distribution of air at suitable air speeds for animal or human

Natural ventilation is the least troublesome, most efficient and least expensive system for providing an optimum environment within a building. The aim of the ventilation system must be to provide a continuous stream of fresh air to every housed animal at all times.

In general there is no accurate, reliable, online measurement method for estimating the ventilation rate (VR) throughout naturally ventilated livestock buildings. Kiwan et al. (2012) compared the values of VRs estimated for a naturally ventilated barn using three different methods: a tracer gas technique using ^{85}Kr, air velocity through the inlet openings and a natural ventilation method. The comparison was performed using Pearson correlation analysis and an ANCOVA model. The study was accomplished using measurements of the air velocity through the inlet openings of the barn, the natural ventilation method (outside wind velocity and thermal buoyancy) and the tracer gas technique (concentration decay method). The authors concluded that it is appropriate to summarize the data of all single measuring points to one decay curve and calculate one regression curve (from this decay curve), that represents the VR of the whole barn, instead of calculating single regression curves for each measuring point and then averaging the single VRs (to one value). For VR estimation based on air velocity measurements, it was concluded that it is more important to measure the air velocity at different points within an opening, to obtain representative data for the whole opening, than to measure at a high number of openings which are located in very similar positions. The three methods investigated showed a generally good correlation with each other. Only when the wind approaching the barn had to pass surrounding obstacles, there was no linear correlation between the air velocity measurements through the inlet openings and the tracer gas technique.

The tracer gas technique is based on a mass balance of a tracer gas in the building air. There are three methods of measuring ventilation and leakage rates with tracer gas techniques:

- the decay tracer gas method
- the method of constant injection
- the method of constant concentration

In the most popular one, the decay tracer gas method, the building is initially enriched with a quantity of tracer gas and allowed to become well mixed to get uniform concentration. Sampling is then performed over time to document the rate at which the tracer gas concentrations decreases. The ventilation rate, in air changes per hour, can then be determined from this tracer decay rate. For all techniques, selection of the tracer gas is very important. It should have the characteristics of being easy to measure at low concentrations, inert, non-toxic, non-flammable, not a natural component of air and with a molecular weight close to the average weight of the air components. Many gases have been used as a tracer gas, such as sulphur hexafluoride (SF_6), methane (CH_4), carbon dioxide (CO_2), hydrogen (H_2), nitrous oxide (N_2O), Argon 41 and Krypton 85. The two gases which are most frequently used are CO_2 and N_2O. The latter is best because it meets all the above requirements. Carbon dioxide can be used, but it is necessary to measure the concentration of CO_2 in the external

air and the rate of release from the animal or soil. In a livestock building, N_2O is better of the two because its concentration is not influenced by the animal breathing.
Using tracer gas is one of the approaches to determine the ventilation flow rates of livestock buildings. Application of an artificial tracer gas is complex, expensive, and may involve technical difficulties to make its distribution uniformly in the buildings. A feasible approach that has been applied by several researches is using the CO_2 produced by livestock as tracer gas to estimate the ventilation airflow in open buildings. Zhang et al. (2010) reviewed the CO_2 production model and the field measurements of five dairy cattle buildings to estimate the feasibility of applying CO_2 production model to determine the ventilation airflow rates in naturally ventilated livestock buildings. The method of using CO_2 production model was compared with that of using an artificial tracer gas and the uncertainty of determination of ventilation rates based on CO_2 production model was analyzed. In this study was assumed that the error in estimated ventilation flow for open buildings was in the range of +/- 50%. The authors indicated that the biggest challenge in using tracer gases for estimation of ventilation flow is to decide where the carbon dioxide concentration sensor should be placed. The potential error due to wrong sensor placing is increasing progressively with the size of wall and roof openings.

Tracer gas techniques can be used not only for ventilation rate measurements but also for:

- Identification and characterization of air movement pathways. By this testing method, air sampling is first performed to assess if there is any background levels of tracer gas, typically sulfur hexafluoride (SF_6). After documenting that no measurable quantities of tracer are present, a tracer gas release is initiated. Then if tracer gas is detected, it will be from this release, and a pathway between two or more locations in a building can be determined. If the measured tracer gas concentration builds rapidly to an equilibrium value after the release and drops rapidly after the release is terminated, then the pathway is short and direct. If however, the measured tracer persists, or even continues to increase, after the release has been terminated, then the pathway involves an intermediate significant volume that acts as a reservoir for this transfer of tracer.
- Determination of Volumetric Flow. In this determination a measured quantity is released into the air stream. Downstream of this release location, sampling is performed. A calculation is then performed to determine what volume of the tracer gas is needed to be diluted into the air to yield the tracer gas measurement obtained.
- Determination of Re-entrainment. In this determination it is the pathway of air movement from an external release, such as from a stack, which is evaluated to assess whether this discharge is re-entering the building.

3.2 Emissions from natural ventilated livestock buildings

Natural ventilation of livestock buildings plays a key role to the emission and dispersion of aerial pollutant from these buildings. The odorous compounds of livestock odour include ammonia (NH_3), amines, hydrogen sulphide (H_2S), volatile fatty acids, indoles, skatoles, phenols, mercaptans, alcohols, carbonyls, p−cresol and volatile carboxylic acids. The concentrations of some major compounds (H_2S, NH_3, volatile organic compounds (VOCs)) of livestock odour have been measured and large variations have been observed among different studies, (Yu, 2010). Although the concentrations of many individual compounds responsible for livestock odours are below the standardized odour detection thresholds, the intensity of the total mixture may be very strong because the intensity of odorous emission results not only from detectable individual compounds, but also from the aggregate effect of numerous odorous chemicals with concentration below detection threshold. There are several models such as ISC3 (Industrial Source Complex), AERMOD (AMS/EPA Regulatory Model), ADMS (Atmospheric Dispersion Modelling System), AUSPLUME (AUStralian PLUME dispersion model), INPUFF (Gaussian INtegrated PUFF model), CALPUFF (A Lagrangian Puff model), and others, that are commercially available and have been applied for modeling agricultural odour dispersion, (Yu, 2010).

The spatial and temporal distributions of ammonia concentrations and emissions in livestock buildings depends on the ventilation rate itself, the airflow patterns inside the building, floor construction, waste storage system, animal activity level and type of feed. The airflow pattern is an important factor inside the building; it is influenced by building geometry and construction, size and distribution of ventilation inlets and outlets, heat production, and particularly for the naturally ventilated buildings, the local wind environment. At given ventilation rate the size and location of emission areas such as feeders and

floor, also affect the aerial distribution of the ammonia gases.For a mechanically ventilated livestock building, ventilation rate can be monitored by a pressure nozzle equipped to exhaust duct or by recording the speed of the exhaust fan. The method cannot be applied directly to naturally ventilated buildings which possess large openings. The openings may act as inlet at one condition and as outlet at another condition according to the external wind directions.
It is more difficult to size a natural ventilation system than it is a mechanical ventilation system, because in the later, the designer has accurate information regarding the volume flow rates produces by items of equipment (fans, diffusers, duct combinations). In contrast, there is much more uncertainty regarding for the example the combination of opening sizes and buoyancy-producing heat loads required to drive a certain volume flow in a naturally ventilated regime. It is therefore important that some form of model, physical or theoretical, is used to optimise the key parameters such as opening sizes and their position, prior to construction.

A special issue consist of eight articles regarding the airborne pollutant emissions from naturally ventilated livestock buildings will be published in 2013 in Biosystems Engineering scientific journal. In this issue three methods for modeling the ammonia emission are presented, ammonia release modeling, air change modeling and CFD air change modeling, (Bjerg et al., 2013a; Bjerg et al., 2013b; Bjerg et al., 2013c).
Given the complexity and expensiveness of measuring ventilation rates, concentration differences and air flow patterns within a naturally ventilated building using a tracer gas technique, CFD calculations can contribute to determine the ventilations rate, emission level and internal airflow patterns and gas concentrations.

3.2.1 Role of Computational Fluid Dynamics in experiments using tracer gas

One of the biggest disadvantages of applying tracer gas techniques to measure the ventilation performance of a building is that these techniques are expensive and time consuming. This is the reason why a detailed experimental set up is needed. In addition, the measurements can be used for model development. The main question is how the application of CFD can be useful to perform, in the most beneficial way, experiments using tracer gas techniques. Since by using CFD models is possible to simulate any physical phenomenon related with flow of mass and/or energy, CFD can be used in order to:

- Design experiments where tracer gas will be used
- Interpret the measured data while they are obtained
- Develop and validate general models, based on experimental data

The design of the experiments can be done by performing preliminary calculations using CFD models. In these models the parameters influence the tracer gas distribution inside and around the building can be analysed. Through this approach can be determined:

- Order of magnitude regarding the ventilation rate of a particular building under different weather conditions, mainly wind speed and direction
- Locations that tracer gas can be measured without obtain data that don't represent the inside environment
- Order of magnitude of tracer gas concentration in certain building areas
- Number of sample points are needed or how many sensors
- Best time interval for obtaining and analysing the tracer gas
- Potential differences between the tracer gas concentration of different areas
- Relocation of the sensors can provide more data for further analysis
- Areas that the sensors must be installed, in the case that outside measurements have to be obtained

When a CFD model has been used to design an experimental setup with tracer gas, the same models can be used simultaneously while measured data are obtained. The main goals are to:

- Validate the CFD model
- Identify unrealistic measurement data (sensor failure)
- Calculate the values of parameters that can be easily checked like air temperature and humidity
- Decide if relocation of the sensors is needed
- Produce tracer concentration and temperature maps
- Determine any uncertain factor has not been considered during the design of the experiments

The ultimate task of every CFD model is to be validated with real data measured in situ. In most of the cases the validation of CFD models has been done by using data obtained from wind tunnels, where scale models are used. Since the fluctuations of boundary conditions in real buildings are much higher than the conditions occurred in wind tunnels, the validation of CFD models using data obtained in real buildings is very difficult. At this stage the most difficult task is to modify the CFD model in order to accurately describe the measured data and simultaneously keep the CFD model as general as possible, to be able to be used for future simulations. The CFD model developed at the phase of the experimental design and modified during the experimental process, must be validated according to spatial and temporary distribution of:

- Air temperature inside the building and mainly in the areas where the animals are located
- Air flow
- Tracer gas concentration inside and outside the building
- Target gas concentration measured inside and outside the building
- Pressure conditions in the case that have been measured
-

When a CFD model has been validated, preferably using measured data from a real building, it can be used to simulate practically unlimited ‘what if’- scenarios. In this phase special attention must not be given anymore to the design details of the certain building used for validation but to the sub models being used in the CFD model. These and other aspects concerning the development and use of CFD models is given in a comprehensive overview on this subject Tropea and others. (Tropea et al., 2007).

4 Overview of literature, software and research groups

4.1 Literature

4.1.1 Search criteria

A literature study related to the application of CFD techniques for the investigation of naturally dairy cow buildings was conducted using the following criteria.

- The relevance of scientific topic. Only studies related to naturally ventilated livestock buildings have been considered, mainly for dairy cows.
- The time of publication. The studies published the last 15 years.
- The publishing medium. References are mainly articles that have been published in international peer reviewed journals and have been registered in international databases, as well as dissertations.
- The distribution of institutions, universities and research centers. Studies from Europe, Asia and the USA have been addressed.
- The complexity of the CFD models used. Mainly 3D simulation models have been considered.
- The existence or not of experimental measurements used to validate the results provided by the simulation models.

Using the keywords 'CFD' + 'natural' + 'ventilation' + 'dairy' or 'cow' or 'livestock', 85 articles were found in the database of scopus (http://www.scopus.com/home.url). The articles were published from 1997-2012. From them, 16 were published in conference proceedings and 69 in scientific journals (figure 1). Almost half of the articles (47%) were published the last 3 years, from 2009-2012, indicating that there is an increasing interest from the scientific community to topics related with natural ventilation.

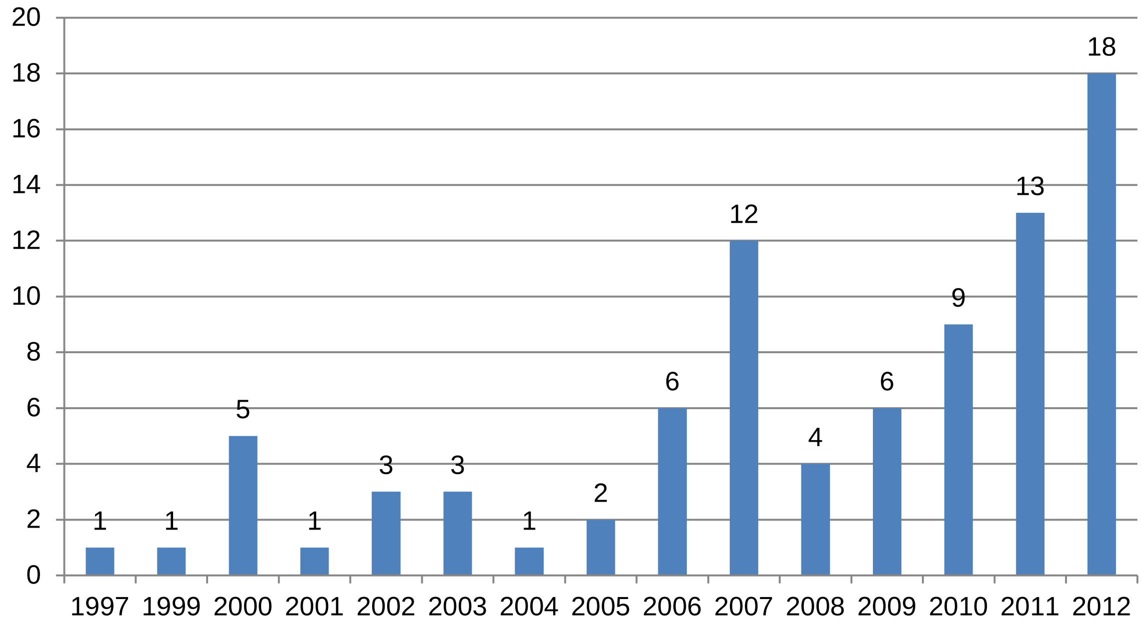

Figure 1: Number of publications related to CFD and natural ventilation in livestock buildings

4.1.2 Literature overview

An extensive literature review has been carried out focusing mainly in the most recent published articles in scientific journals.

Full-scale experimental data and computational fluid dynamics (CFD) methods were used by Bartzanas et al. (2007) to determine the accuracy of four different turbulence models [standard *k-ε*, *k-ε* renormalisation group (RNG), *k-ε* realizable and Reynolds stress model (RSM)]. The models were used to describe the turbulent part of air in problems concerning the natural ventilation of buildings.

Ventilation rates were measured in a livestock building using the decay tracer gas (CO_2) technique. In addition, airflow and temperature patterns were mapped out in a greenhouse with a tomato crop using a three-dimensional sonic anemometer and a fast-response temperature sensor and average values from experiments were used for boundary conditions. The numerical results were compared with the experimental data, showing a good agreement, especially when the *k-ε* RNG turbulence model was used. The computations of the flow field using the different turbulence models showed noticeable differences for computed ventilation rate, air velocity and air temperature confirming the importance of the choice of the closure model for turbulence modeling,(Bartzanas et al., 2007)

A computational fluid dynamics (CFD) model was developed by Norton et al. (2009) to investigate the natural ventilation of a climatic livestock building under different wind incidences (WIs) for three different inlet opening areas. A 1/2 scale experimental duo pitch building was employed to validate, both qualitatively and quantitatively, the CFD predictions of airflow distribution. To improve the applicability of CFD to building design, a thermal comfort index called the "minimum comfort temperature" was used in this study. Results showed that ventilation *rates* were not at their highest, when wind was blowing at moderate pace to the building because a considerable quantity of the flow exited the building via short-circuiting. However, the greatest ventilation *homogeneity* was experienced when the wind was blowing at moderate pace to the building, because of the formation of two wind-driven vortices within the building. Results also showed that the highest level of environmental heterogeneity occurs at WIs of 10-40° because the primary vortex only occupies a portion of the total building volume. It was also found that in some circumstances the ventilation rate determined from the flow rate through the building openings may not accurately represent the actual ventilation rate of a building, and measurements and simulations of contaminant decay may form a more accurate measure of ventilation rate (Norton et al., 2009).

Natural ventilation depends largely on temperature difference between inside and outside air and wind velocity and direction. Therefore, it is very important, already in early stages of building design, to provide orientation and accurate opening areas (Ecim-Djuric and Topisirovic, 2010). Natural ventilation and computation of fluid dynamics in livestock buildings can be usefully integrated in whole ventilation system optimization and related energy consumption decrease. CFD analysis is generally restricted to the study of buildings environment flows and space study, and the designer must supply boundary conditions in the form of external and internal buildings envelope (wall) surface conditions.

The use of computational fluid dynamics (CFD) to evaluate the climate distribution in agricultural buildings has grown in importance in recent years, (Norton et al., 2010c). Convection and radiation are the dominant forms of heat transfer from an animal's body. Accounting for animal heat flux during CFD simulations is necessary to achieve a good understanding of the livestock's thermal environment. Of the total heat flux leaving the animal, the convective part is regularly calculated using a predetermined ratio between the convective to radiative parts (C-R ratio). However, by employing this ratio the CFD modeller is essentially forcing the simulated animal to lose a certain amount of convective heat, irrespective of the environmental conditions at their location. Therefore, unless the indoor environment is known a priori to a CFD simulation, and unless this environment is totally uniform, the C-R ratio ascribed to some of the simulated animals will be less precise. In order to address this difficulty, Norton et al. (2010c) used a zero-dimensional calf heat transfer model to account for calf thermoregulation during CFD simulations. The effect of using this representation of calf heat flux was analysed for practical housing scenarios and differences in the predicted airflow distribution patterns were found to occur between the simulations that used a fixed and dynamic relationship for calf heat flux partitioning.

A new approach to express the ventilation rate using CFD is by calculating the “age-of-air”, (Kwon et al., 2011). This concept can be used to assess the ventilation efficiency of an agricultural facility. However, experimental research has been limited due to the indirect method of using unstable tracer-gas and limited instrument. To overcome limitations and increase applicability, CFD technique was employed from established methodology by user-defined functions, (Kwon et al., 2011). In this study a three-dimensional chamber was designed to accurately implement and verify the age-of-air through simulation and tracer-gas experiment under unsteady-state conditions. In validating the computations of the local-mean-age and local-mean-residual-lifetime, the results showed similar quantitative and qualitative distributions with average errors of 9 and 13%, respectively. The authors concluded that the method of realizing the age-of-air via CFD was reasonably well designed and capable of estimating ventilation efficiency of agricultural facilities under unsteady-state conditions. The results also showed that when air exchange rate (AER) increased in the target structure, the age-of-air values decreased. But when comparing air exchange efficiencies, the values had an opposite tendency. Through the methodology presented in this study, the feasibility of analyzing ventilation efficiency using age-of-air in agricultural facilities was confirmed and it will be upgraded for actual application considering characteristics of ventilation structure.

In a study contacted by Shen et al. (2012) computational fluid dynamics (CFD) and experiments, using the pressure method and tracer gas method, were compared. The simulations were based on five two-equation RANS turbulence models. A building model with two different sizes of the ridge openings was investigated in a wind tunnel. Results shows that the simulated overall ventilation rate through building space by standard *k-ω* model agreed the most with the experimental data. However, there lies a large discrepancy between simulation and experimental data as the ridge opening size become larger. The discrepancy may arise from shortcomings of the used method in calculating the ventilation rate in larger ridge opening. Also a lack of reliable technique to measure the wind pressure at the ridge opening may be an explanation. As for the tracer gas method, a generally good fit was found comparing with simulation results; however, the variance still exists as the wind direction was more perpendicular to the building sidewall. The discrepancy was expected to be reduced by the improvement of measurement accuracy of experiment,(Shen et al., 2012)

Ventilation is directly linked with emissions of gases with environmental impact, such as ammonia (NH_3), methane (CH_4) and nitrous oxide (N_2O). To acquire a better understanding of the complex natural ventilation process in and around animal houses, air velocity measurements were carried out in 1:60 scale models of a dairy barn placed in a wind tunnel, using a reference air velocity of 3.5 ms^{-1}, (De Paepe et al., 2012). In this study six different ventilation opening configurations were compared, in order to quantify their effect on internal and leeward air velocity profiles. The different scale model designs clearly gave rise to variable indoor and outdoor air velocities. Enlarging the inlet opening height led to lower velocities near the inlet, while higher velocities were measured at the outlet. The air velocities at the centre of the house were hardly affected by the inlet opening height, even with the front wall completely removed. Removing the outlet wall at the same time, however led to much higher velocities at the centre of the scale model (3-4 times higher). Higher airflow rates were found in scale models with larger ventilation opening surface areas. The presented results additionally provide a useful experimental basis towards validation of CFD simulations.

4.1.3 Computational Fluid Dynamics modeling of ammonia dispersion in natural ventilated livestock buildings

The application of CFD to model the dispersion and emission of ammonia within and around NV livestock buildings adds another layer of complexity to the situation, namely how to best represent the flux of ammonia from emitting sources under different convective conditions. Ultimately, CFD is expected to make accurate predictions of ammonia and to be subsequently used to reduce the emission of ammonia through design modifications, (Bjerg et al., 2013a). The main requirements of this approach are to:

- Ensure that the convective flow regimes in the building are accurate representations of reality. This requires understanding of the flow regime being modelled and the best turbulent models to accurately describe this regime;
- Ensure that the building geometry and surrounding environment reflect realistic conditions as accurately as possible, as without good understanding of the CFD modelling process for wind-building interactions it is possible to compromise the accuracy of solutions and reduce efficient simulation capabilities;
- Ensure that the release of ammonia is accurately modelled, i.e. this represents the ammonia production processes inside the manure as well emissions from the manure surface. This requires not only understanding of the ammonia production processes for manure stored in houses with different flooring systems (slatted, bedded, solid scraped floor) but also the attention to the mass transfer across the boundary layer of the exposed manure surface as well as influencing pressure drops associated with the floor (straw bedding, slatted floors).

From these three points it is evident that most of the elements that drive the dispersion and emissions of ammonia from NV livestock buildings can be quantified using CFD. In fact most of these issues have already been given significant consideration by researchers in the field of NV livestock building design. Therefore, in the following sections the state-of-the-art development pertaining to the main requirements for accurate ammonia emission modelling will be comprehensively reviewed.

In order to use CFD to model the dispersion of gases such as ammonia the Navier-Stokes equations has to be coupled with an associated species equation (for ammonia). In practice the accuracy of the species equation is based on a number of factors such as the ability of the CFD model to describe the nature of the flow-field via turbulence modelling, as well as the ability of the boundary conditions to accurately represent the quantity and spatial distribution of ammonia.

An overview of CFD models have been used to analyze the ammonia dispersion from natural ventilated buildings is given by Bjerg et al. (2013).

4.1.4 Computational Fluid Dynamics modelling techniques

An overview of the “Best Practice Guideline for the CFD Simulations in the Urban Environment” is given in Franke et al. (2007). This report provides best practice guidelines for undertaking simulations that are used to evaluate microscale obstacle-accommodating meteorological models. The different sources of errors and uncertainties that are known to occur in numerical simulation results are listed and defined. In this study the sources of error that can be controlled and quantified by the user are discussed in detail and best practice guidelines for their reduction and quantification are given. In addition, a review existing guidelines of best practices is given showing that there are several existing guidelines available, at least for the computation of the flow in the urban and industrial environment with the statistically steady RANS equations for neutrally stratified flow fields. Only general guidelines are provided as most parameters depend to a large extent on the details of the application problem, (Franke et al., 2007).

4.1.4.1 Geometrical design and mesh generation

The computational domain

The modelling of NV buildings requires the computational domains both inside and outside the buildings even though the area of interest is only the inside. In this case, the outside domain should be large enough not to interfere in simulation result inside the building. In wind engineering, the size of the outside domain is generally determined by the height of the building. The level of detail required for individual buildings is dependent on their distance from the central area of interest. Buildings further away may normally be represented as simple blocks (Franke et al., 2007).

The size of the entire computational domain in the vertical, lateral and flow directions depends on the area that shall be represented and on the boundary conditions that will be used. The extent of the built area (e.g. buildings, structures or topography) that is represented in the computational domain depends on the influence of the features on the region of interest. An experience from wind tunnel simulations is that a building with height H_n may have a minimal influence if its distance from the region of interest is greater than 6-10 times H_n. In case of uncertainty about the influence of distant features on the flow and dispersion in the area of interest it is recommended to perform simulations with and without the features, i.e. larger and smaller built areas. All recommendations presented in the following sections depend on the boundary conditions which are generally applied. For single buildings the top of the computational domain should be at least 5 times H_n above the roof of the building (Franke et al., 2007). The large distances from the building of interest are necessary to prevent an artificial acceleration of the flow over the building, as most boundary conditions applied at the top of the computational domain do not allow fluid to leave the domain, (Tominaga et al., 2008). For areas with multiple buildings the top of the computational domain should be 5 times H_{max} away from the tallest building with height H_{max}. Most details about best practices regarding the lateral extension of the domain and the extension of the domain in flow direction can be found in the report of Franke et. al. (2007) and Tominaga et al. (2008).

Computational grid

The computational results depend crucially on the grid that is used to discretise the computational domain. The grid has to be designed in such a manner that it does not introduce errors that are too large. This means that the resolution of the grid should be fine enough to capture the important physical phenomena like shear layers and vortices with sufficient resolution. Also the quality of the grid should be high. Ideally the grid is equidistant. Therefore, grid stretching/compression should be small in regions of high gradients, to keep the truncation error small. The expansion ratio between two consecutive cells should be below 1.3 in these regions.

The domain inside the building generally requires finer resolution meshing than the outside domain. In this case, increasing mesh density near buffer zone, such as ventilator and opening, with finer meshes inside the building and coarser meshes outside the building can be important. In the area of interest, at least 10 cells per cube root of the building volume should be used and 10 cells per building separation to simulate flow fields. This must be understood as an initial minimum grid resolution. The necessary resolution then will have to be analysed by using grid refinement which is available in most of commercial CFD codes (Franke et al., 2007).

Outdoor environments, such as terrain, obstacle, etc., are actually very important because internal air flow of the NV building is highly dependent on outdoor wind and pressure distributions. It follows that also the prediction of ammonia emission and dispersion out of the building requires detailed modelling of outside domain. For complex terrain features, generation of a computation grid is very complicated and it is also not possible to maintain high resolution within the overall area of interest. An overview about the grid size used in different CFD studies related to livestock buildings is given by Bjerg et al. (2013). The grid size used ranged from 10^4 cells for a 2D computational domain (Sun et al., 2004) to around $3*10^6$ cells (Seo et al., 2009).

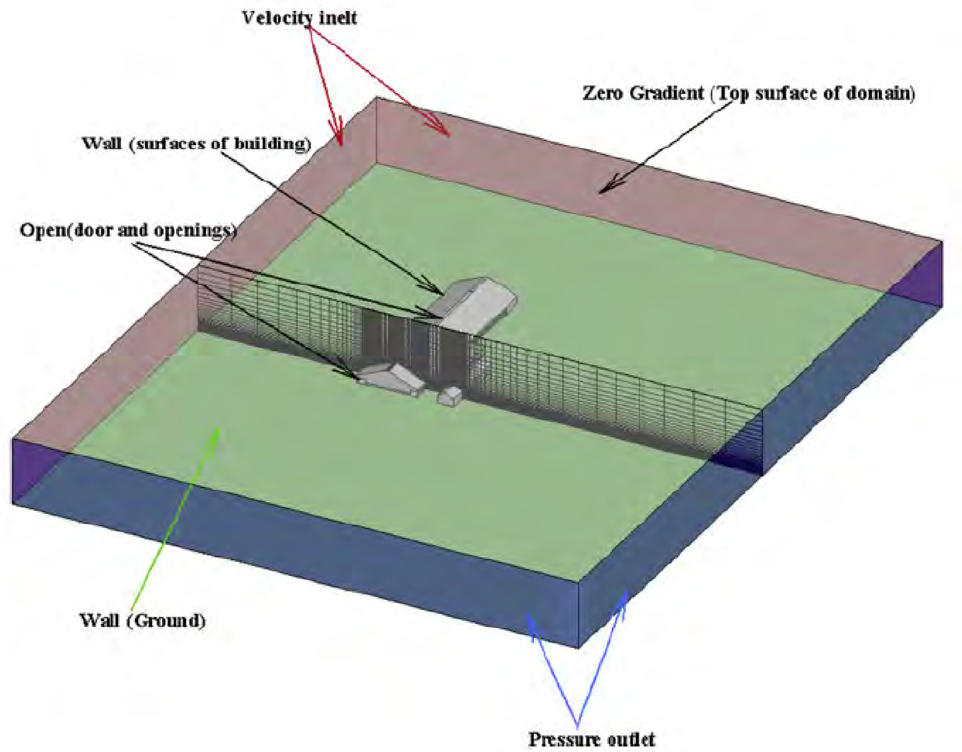

Figure 2: Mesh and boundary conditions of a dairy cattle building, (Wu et al., 2012a)

Geometrical design of the animals
In most of the CFD models that have been used to investigate the naturally ventilated livestock buildings, the presence of animals either is neglected or is simulated as area with higher resistance to air flow (porous media). There are only few studies where the exact design of the animal has been considered for CFD simulations (Figure 3, 4 and 5). In a study contacted by Sapounas et al. (2012), is described how a detail CAD design of a dairy cow can be used to calculate the potential resistance of the animals to the inside air flow.

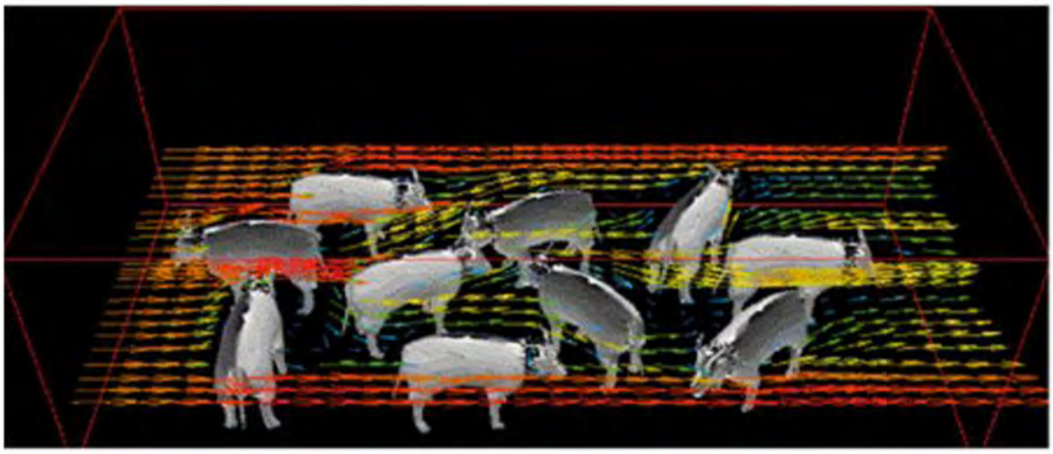

Figure 3: Flow field around a ventilated room of cows, (Gebremedhin and Wu, 2003)

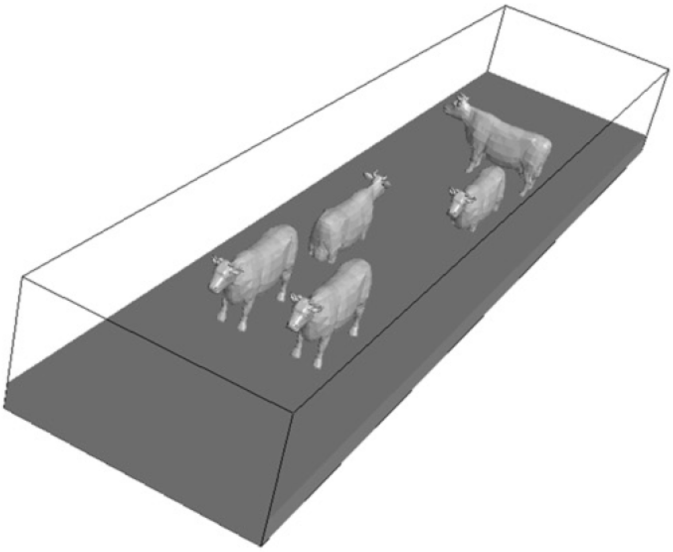

Figure 4: Cad design of dairy cows used as submodel for a CFD study, (Wu et al., 2012a)

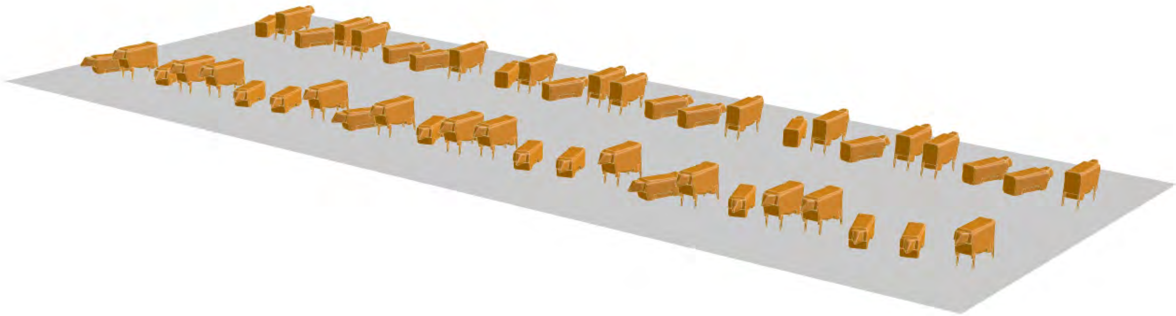

Figure 5: 3D model of a virtual dairy cow house with 48 cows (Sapounas et al., 2012)

4.1.4.2 Numerical methods, sub models and boundary conditions

Numerical method

The choice of the basic equations has the largest impact on the modelling errors and uncertainties. First it has to be decided whether the application requires an unsteady or a steady treatment. As the atmospheric boundary layer flow is turbulent, an unsteady treatment is required in principle. The turbulent flow within livestock buildings, in general is modelled by the Navier-Stokes equations. Temperature and humidity equations as well as equations for liquid and solid water compounds need to be considered if relevant for the simulations (e.g. non-neutral atmospheric stratification, small wind speeds and large temperature gradients). However, in many cases, approximations may be used that still ensure reliable model results. When pollution dispersion is taken into account, then one or more additional transport equations for the pollutant(s) have to be solved. Depending on the state of the pollutants (gaseous, liquid, solid) further physical complexities like chemical reactions, break-up, coalescence, evaporation and particle-particle interaction may have to be modelled. Neglecting liquid and solid pollutants one still has to decide whether the gaseous pollutant affects the flow field through a substantial change in density, (Franke et al., 2007).

Turbulent airflow
One of the most important aspects in CFD modelling is the choice of the proper turbulence model. Several studies have focused on the effect of various turbulence models on the final numerical solution, showing that no general rules could be applied in all simulation models. The most common used turbulence model is the standard *k-ε* which, has been tested in many cases describing the ventilation process in greenhouses. Despite it's widely usage the standard *k-ε* model has small accuracy especially in low velocity magnitudes and should be carefully used. In addition, it contains many empirical constants that have long been known to have an adverse effect on prediction performance (Versteeg and Malalasekera, 1995). In recent times engineers have turned to more complex turbulence models like the two-scale *k–ε* turbulence models such as the renormalisation group (RNG) and the realizable *k–ε*, which are not so reliant on empiricism. In a two-dimensional wind induced ventilation study (Mistriotis et al., 1997) showed that better qualitative agreement with experimentally observed flow patterns could be achieved with a two-scale *k–ε* turbulence model than with the standard *k–ε* one. In a similar study concerning the simulation of pollutant dispersion in an urban street canyon, carried out by Sapounas and Campen (2003) four turbulence models were validated against experimental data presented by (Gerdes and Olivari, 1999); the standard *k-ε*, the RNG *k-ε*, the realizable *k-ε* and the RSM. All the models were tested for two different boundary conditions concerning the turbulence Intensity 'I' in order to find the influence of Reynolds number, I=5.8% and I=0.1%, (Sapounas and Campen, 2003). In this study, the results concerning the absolute percentage variations between the experimental values and computational results, provided by four turbulence models ranged from 5.6% to 118.2%. The sensitivity analysis showed that the most appropriate turbulence model was the realizable *k-ε* one. The same turbulence model was proposed by Garcia Sagrado et al. (2002) in a similar study, (Garcia Sagrado et al., 2002). Although, the realizable *k-ε* turbulence model performs relative better according to above mentioned studies, recent ones proposed the RNG *k-ε* as the most accurate turbulence model, regarding natural ventilation problems (Bartzanas et al., 2007; Stavrakakis et al., 2008). These differences indicate firstly that indeed the standard *k-ε* turbulence model is not accurate enough, and secondly the necessity of performing preliminary calculations in order to determine the most appropriate turbulence model.

Heat release from animals
In NV buildings the heat production from animals may have a significant importance for the exact airflow and temperature condition when the outdoor wind speed is low or when the ventilation openings are small. Heat release from animal surface is depending on radiation and convection which is influenced by environmental conditions such as air velocity, temperature and other emission factors including body surface of animal and thermal insulation between body and tissue layer.

In a study contacted by Bjerg and Andersen (2010) the assumed animal density was $12m^2$ cow^{-1} corresponding to $9m^2$ HPU^{-1} (Heat Production Unit). The assumed sensible heat production was 700W HPU^{-1} of which 500W HPU^{-1} was distributed as convective heat in the animal occupied zone from 0-1.2 m above the floor in the barns (Bjerg and Andersen, 2010). The reason for assuming this relative high convective heat release was that it would secure a realistic temperature increase in the assumed well insulated building without modelling radiation heat transfer. The remaining part of the sensible heat – mainly radiation heat – is assumed to correspond to the heat loss through the building constructions, which the model does not take in to account. In a study contacted by Sapounas et al. (2012) the heat production was considered only as sensible heat at 700, 900 and 1200W per cow, when the outside air temperature was 20°C, 10°C and 0°C respectively. Norton et al., proposed a more accurately description of the animal heat release by using a zero-dimensional calf heat transfer model to account for calf thermoregulation during CFD simulations, (Norton et al., 2010c).

Boundary conditions for weather
In NV buildings, modification of the indoor environment is mainly achieved by the action of external factors (solar radiation, air temperature, wind speed and direction, etc.) which should be adequately simulated in CFD calculations through boundary conditions. The boundary conditions represent the

influence of the surroundings that have been cut off by the computational domain. As they determine to a large extent the solution inside the computational domain, their proper choice is very important. Often, however, these boundary conditions are not fully known. Therefore the boundaries of the computational domain should be far enough away from the region of interest to not contaminate the solution there with the approximate boundary conditions.

At the inflow an equilibrium boundary layer is usually prescribed, at a distance of at least 5 times H_{max}. The mean velocity profile is usually obtained from the logarithmic profile corresponding to the upwind terrain via the roughness length z_0 or from the profiles of the wind tunnel simulations. Available information from nearby meteorological stations is used to determine the wind speed at the reference height. For steady RANS simulations, the mean velocity profile and information about the turbulence quantities is required. Their profiles can be obtained from the assumption of an equilibrium boundary layer, (Franke et al., 2007).

At solid walls the no-slip boundary condition is used for the velocities. In most of the CFD codes the equations for the turbulence quantities contain damping functions to reduce the influence of turbulence in this region dominated by molecular viscosity. The low-Reynolds number approach requires a very fine mesh resolution in wall-normal direction. To reduce the number of grid points in the wall-normal direction and therefore the computational costs, wall functions are applied as an alternative approach to compute the wall shear stress. With the wall function approach, the wall shear stress is computed from the assumption of a logarithmic velocity profile between the wall and the first computational node in the wall-normal direction, (Franke et al., 2007).

At the boundary behind the obstacles, where all or most of the fluid leaves the computational domain, open boundary conditions are used in commercial CFD codes, which are either outflow or constant static pressure boundary conditions. With an outflow boundary condition the derivatives of all flow variables are forced to vanish, corresponding to a fully developed flow. Therefore this boundary should be ideally far enough away from the built area to not have any fluid entering into the computational domain through this boundary.

Thermal characteristics of the building

In general, the heat transfer through building constructions is calculated by assuming that the heat moves by conduction, while heat moves from warm areas to cold areas by three way; (1) conduction, (2) convection and (3) radiation. The heat flow by conduction is only dependent on thermal conductivity and thickness of the materials. In CFD simulation, this thickness of a wall, which is much smaller than the size of the whole domain in most cases, tend to disturb the overall mesh quality since thin wall requires very small meshes. For this reason, most CFD tools, and researches treat walls as zero thickness in pre-processing and specify the thickness for walls in main computation. This is a general and simplified method but conduction is considered only in the normal direction, i.e. across the wall thickness. However in cases where conduction in the planar direction of the wall is also important, other methods are available. A detailed description about thermal characteristics used in CFD models is given by Bjerg et al. (2013).

Simulating the tracer gas technique

Various techniques have been used to measure and predict ventilation rate such as tracer gas techniques, energy balance and measurements of pressure differences between inside and outside. Sapounas et al. (2012) calculated the ventilation rated of seven commercial dairy cow buildings (figure 6) by simulating the method of constant injection of a tracer gas. (Sapounas et al., 2012). In the simulation model the tracer gas was used as a virtual gas called air-tracer which has the same physical properties as normal air. The ventilation rate in $m^3\,s^{-1}$ is given by Equation 1.

$$\phi_{v,kelder} = \frac{\phi_{m,tracer}}{c_{tracer}} \qquad \text{Equation 1}$$

Where $\phi_{m,tracer}$ is the constant mass flux of the tracer gas (air-tracer) in kg s^{-1} and c_{tracer} is the concentration of the tracer gas in the building in kg m^{-3}.

The seven simulation models, each corresponding to a different barn type were solved for 27 cases concerning the wind speed (1, 4 and 8 m s^{-1}), wind direction (0°, 45° and 90°) and outside air temperature (0, 10 and 20°C). In total 189 cases were solved. The heat production of cows were specified only as sensible heat to 700, 900 and 1200 W cow^{-1}, when the external air temperature was

20°C, 10°C and 0°C respectively. Finally, the ventilation rate was calculated for each building but also for the area where the cows are standing or laying down.

The results of the model calculation showed that for all the cases the ventilation rate was influenced substantially by the building design followed by the wind characteristics and to less extend by the buoyancy effect. Higher ventilation rates for a given wind speed were calculated when the direction of the air flow is parallel to the ridge of the building. The influence of the wind direction decreases while the wind speed increases from 1 to 8 m s^{-1}. The roof shape influences the resistance of the building to the air flow. This is the reason why the lower ventilation rates were calculated for the buildings with Design 4 and Design 6 since these buildings have round cover (Figure 6). Another important factor was proven to be the ratio between the covered area (m^2), which is linearly correlated to the number of animals, with the volume of the building (m^3). These ratios were: 0.16, 0.12, 0.11, 0.12, 0.18, 0.16 and 0.23 m^2 m^{-3} for the designs 1-7 respectively. In general the ventilation rate becomes higher when this ratio is increased. The reason is that similar openings area used to ventilate much larger volumes. From the other side in buildings with relatively small volumes, the fluctuations of the ventilation rate due to differences in climate conditions are higher. This is the reason why the building with Design 7 has the highest ventilation rate but also is the most sensitive one to changes in climatic conditions. All the results from this study are depicted in Figure 7.

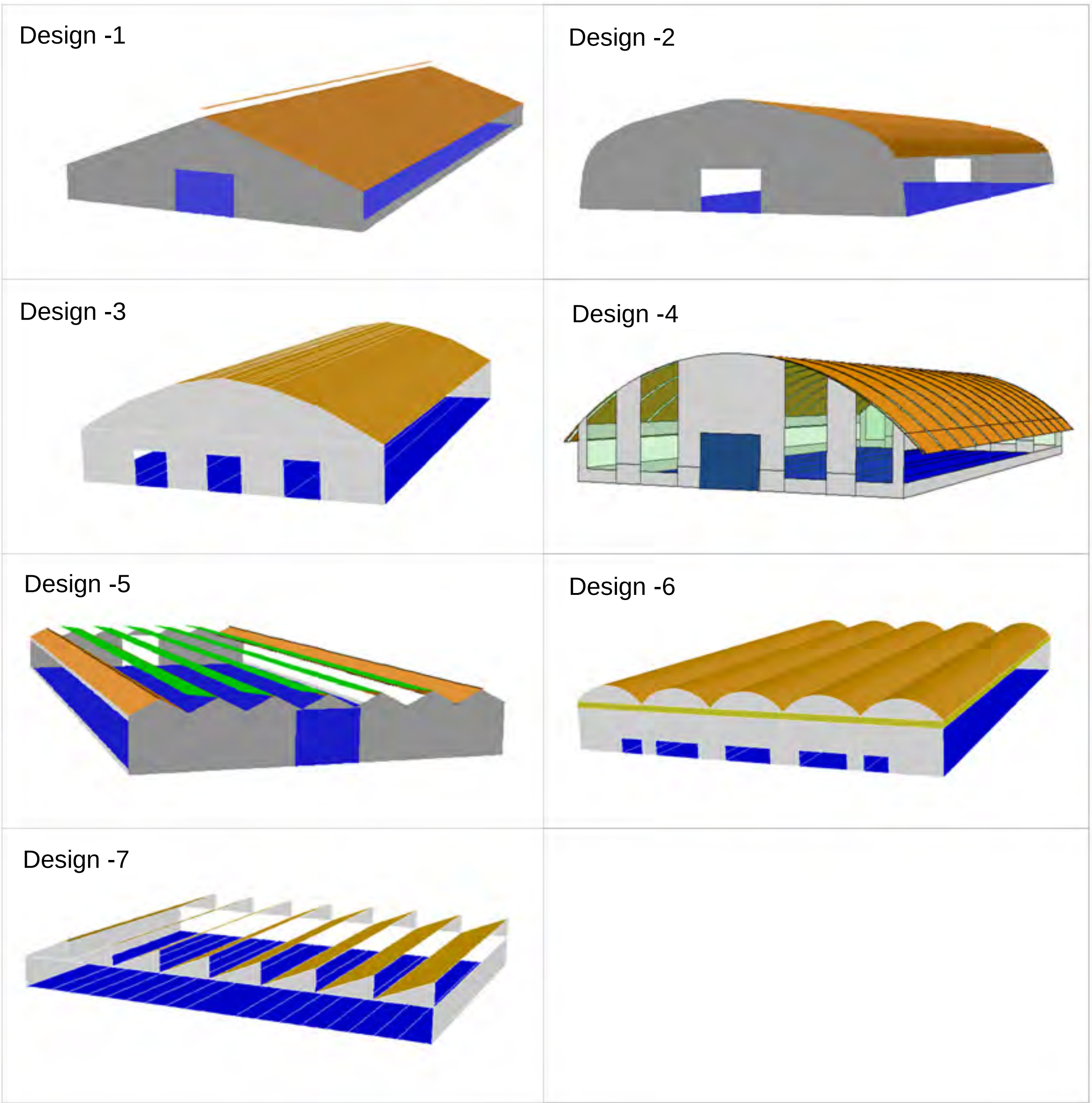

Figure 6: 3D CAD models of seven commercial dairy cow houses used for the CFD simulation models, (Sapounas et al., 2012)

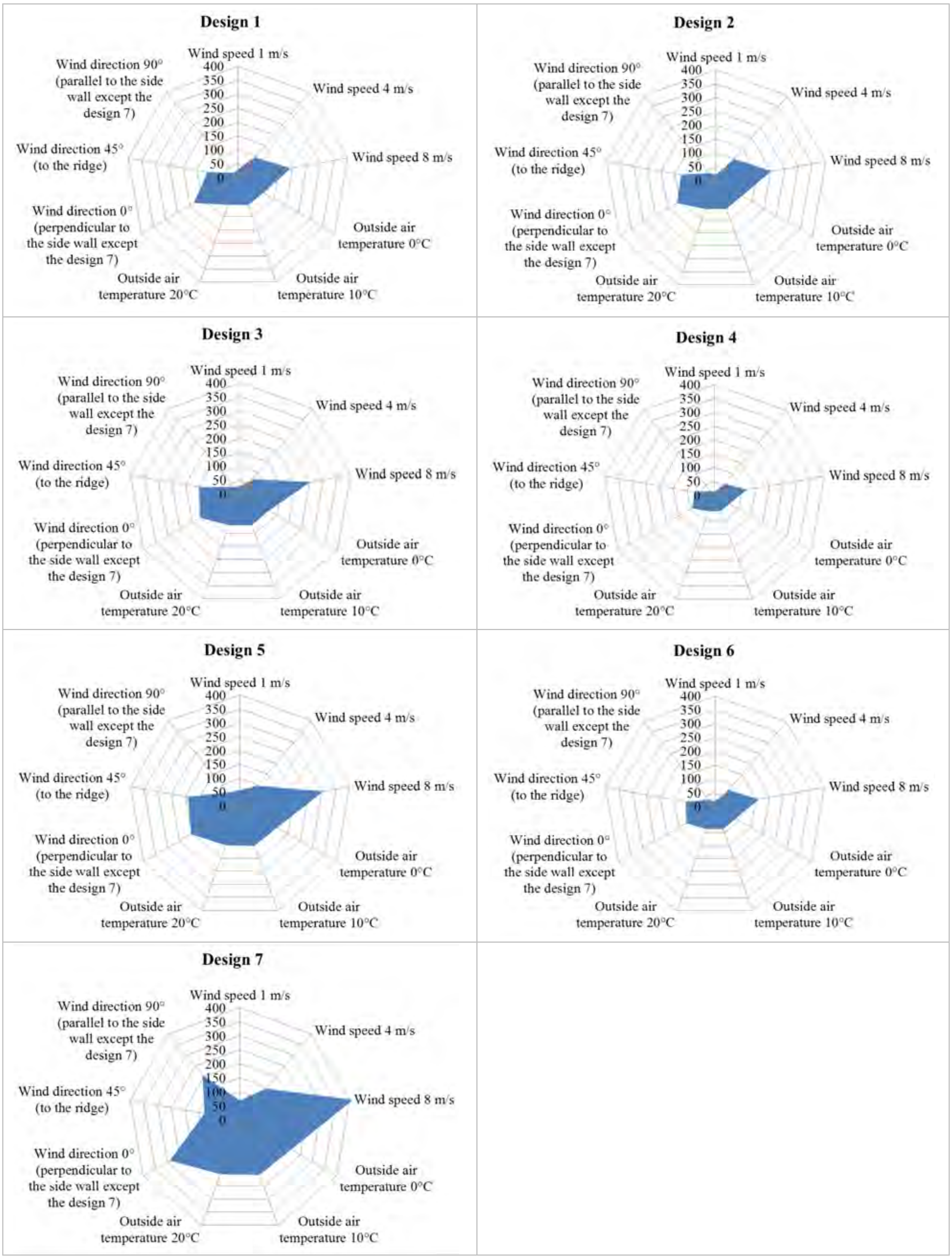

Figure 7: Ventilation performance (air changes per hour) of seven commercial dairy cow houses for different climatic conditions, (Sapounas et al., 2012)

4.2 Computational Fluid Dynamics software

4.2.1 Purposes for use

The primary applications of CFD to the design of naturally ventilated buildings are as follows (Etheridge, 2011):

- Calculation of velocity and temperature fields in rooms and buildings;
- Calculation of envelope flows;
- Calculation of surface wind pressure distributions;
- Whole-field calculations i.e. effectively the combination of (a), (b) and (c);
- Calculation of the flow characteristics of openings (e.g. discharge coefficients).

For mechanical ventilation, application (a) is the main one, with the envelope flows being specified as boundary conditions. For natural ventilation, a more demanding approach is desirable, in which the envelope flows are calculated i.e. a whole-field calculation should be carried out, whereby (a), (b) and (c) form part of the same calculation. Applications (c) and (e) are less demanding and are primarily used to provide information for envelope flow models.

As was mentioned in the introduction, a variety of reasons can be cited for the increased importance of using CFD techniques. The main advantage is that through this approach we have the ability to investigate practically unlimited number of what-if scenarios. CFD techniques have already been used in thousands of application in almost all engineering scientific fields.

4.2.2 Difficulties Associated with natural ventilation

Many problems and difficulties are produced from the large dimensions of buildings (and their three-dimensional nature), with the consequent requirement for a large number of cells, which may exceed the available computer memory or lead to unacceptably long processing times. Eventually developments in computing power will reduce this difficulty. There are other ways of doing so, some of which are more justifiable than others. One approach is to make use of symmetry, where possible, so that only the symmetrical part of the flow is calculated. Another approach is to assume two-dimensional flow. This certainly reduces the number of cells required, but only few buildings or rooms can genuinely be classed as having two-dimensional geometry. Breaking down the building into separate rooms and assuming that their internal flows are independent is an option that is justified when the rooms are isolated. This is more likely to be acceptable for mechanically ventilated buildings than it is for naturally ventilated ones. Ignoring the desirability for a whole-field calculation and restricting the domain to the envelope of the building is another, but at the expense of introducing problems with boundary conditions. Another option is to employ a grid that is coarser than what is generally accepted to be needed for accuracy. For internal flows, the importance of buoyancy to natural ventilation (compared with mechanical ventilation, where forced convection is usually present) introduces several difficulties. It introduces thermal boundary conditions that are difficult to specify. It raises the question of whether radiation should be included. There is also evidence that it can lead to problems with convergence (stability) of solutions. Furthermore, the velocities encountered in naturally ventilated rooms can be very small, leading to low Reynolds numbers and consequential difficulties associated with the simulation of transition and turbulence. At the more detailed level, buoyancy can have an impact on turbulence models. Difficulties also arise in connection with the inherently unsteady nature of natural ventilation and distinctions between different types of steady and unsteady flow can be made.

4.2.3 Available software

Commercial CFD codes are widely available. Most software packages consist of three distinct parts, namely the pre-processor, the solver and the post-processor. The pre-processor is where the user sets the boundary conditions for the solution, including the specification of the computation domain and grid and the number of cells. The number of cells is likely to be of order 10,000 for simple cases such as a small room and of order 1,000,000 for large buildings. The solver carries out the numerical solution. This is an iterative procedure, which continues until acceptable convergence is obtained. Solution times can vary from a few hours to weeks, depending on the complexity of the boundary conditions, the number of cells and the capacity of the computer. The post-processor is where the results are presented in a suitable and accessible form. Since the solution generates large amounts of data, it is important to be able to present it in a meaningful way. The post-processor can also be used to evaluate quantities directly from the computed velocity field e.g. the concentration field of a passive contaminant.

The purpose of the solver is to obtain a solution to a large number of algebraic equations arising from division of the flow domain into a large number of small cells and discretisation of the partial differential equations that govern the flow. Discretisation of the equations essentially means that the derivatives (rates of change) are expressed in terms of the local values of the variables. There are three methods of discretisation employed in CFD, namely finite volume, finite element and finite difference. The most common method is the finite-volume method, which allows more flexibility in the choice of grid. The conservation principle is applied to small volumes (the cells) and the resulting equations are discretised. The conservation equations are essentially those that are used to derive the transport equations. The most accurate one is considered the finite element method. The finite difference method corresponds to discretisation of the transport equations and is rarely used. With all three methods, the evaluation of derivatives becomes more accurate as the grid spacing (and time step) is reduced and the differences between them become less.

The CFD codes are divided in the following groups.

- Solvers. Software which are specialized in solving the numerical part of a CFD model, provide results and visualize the flow.
- Grid generation codes. Software which are used to generate the computational domain and the grid.
- Visualization codes. Software which are used only to visualize the results provided by a the solvers.

There are both free and commercial Software used for CFD simulation. A very detailed and updated list can be found in the web site: http://www.cfd-online.com/Wiki/Codes. The most common in engineering studies which have been used also for the CFD studies related to livestock buildings are:

- Fluent, (http://www.ansys.com/)
- CFX, (http://www.ansys.com/)
- CFD 2000, (http://www.adaptive-research.com/)

Nowadays the previous world leaders in CFD codes, Fluent and CFX are offered from the same company ANSYS CFD which provides access to ANSYS FLUENT and ANSYS CFX products (also available separately). These two core fluids simulation solvers represent more than 1,000 person-years of research and development. The most common used CFD code in the research groups regarding livestock buildings is the FLUENT.

4.2.4 Main characteristics

The three basic types of CFD model are described in the following, beginning with the most fundamental, DNS. The performance of three different numerical techniques, i.e. RANS, URANS and LES were compared by Salim et al., (2011) to determine their suitability in the prediction of urban airflow and pollutant dispersion process using the commercial CFD code Fluent. The CFD codes were evaluated against wind tunnel experimental data, and it was observed that LES although more computationally expensive, produces the most accurate and reliable results because it resolves the turbulent mixing process in the flow field. URANS, albeit solving for the transient solution, fails to account for the unsteady fluctuations and hence is not an appropriate replacement for LES (Salim et al., 2011).

Direct Numerical Simulation (DNS)

DNS refers to numerical solution of the discretised unsteady N–S equations. DNS is still a very long way away from being used for ventilation design, due to the extreme demands it places on computing power. This is mainly due to the relatively small time steps and cell dimensions that are required. In addition, very small time steps are needed to simulate the fine detail of temporal changes. The advantages of DNS are that it is nominally exact i.e. no empiricisms (turbulence models) are required and no similar assumptions about transition are needed. As far as is known, DNS has not yet been used specifically for ventilation studies. However, it has been used to simulate the unsteady flows generated from flow separation on bluff bodies which is one source of turbulence in rooms.

Large-eddy Simulation (LES)

Large eddy simulation (LES) is a technique in which the smallest scales of the flow are removed through a filtering operation, and their effect modelled using sub grid scale models (the Reynolds decomposition is used). This allows the largest and most important scales of the turbulence to be resolved, while greatly reducing the computational cost incurred by the smallest scales. This method requires greater computational resources than RANS methods. Several examples of its use for natural ventilation are given in (Etheridge, 2011). LES provides a direct unsteady solution for all but the smaller scales of the motion. The stresses that are directly calculated are known as the resolved stresses and those that are derived from semi-empirical equations are known as the modelled stresses. LES is also considerably more demanding on boundary conditions, since they should be specified in unsteady form.

Reynolds-averaged N–S Equations (RANS)
Reynolds-averaged Navier-Stokes (RANS) equations are the oldest approach to turbulence modelling. An ensemble version of the governing equations is solved, which introduces new apparent stresses known as Reynolds stresses. This adds a second order tensor of unknowns for which various models can provide different levels of closure. It is a common misconception that the RANS equations do not apply to flows with a time-varying mean flow because these equations are 'time-averaged'. In fact, statistically unsteady (or non-stationary) flows can equally be treated. This is sometimes referred to as URANS. There is nothing inherent in Reynolds averaging to preclude this, but the turbulence models used to close the equations are valid only as long as the time over which these changes in the mean occur is large compared to the time scales of the turbulent motion containing most of the energy.
The RANS equations are primarily used to describe turbulent flows. These equations can be used with approximations based on knowledge of the properties of flow turbulence to give approximate time-averaged solutions to the Navier–Stokes equations.
The main RANS-based turbulence models are:

1. Linear eddy viscosity models
   a. Algebraic models
      i. Cebeci-Smith model
      ii. Baldwin-Lomax model
      iii. Johnson-King model
      iv. A roughness-dependent model
   b. One equation models
      i. Prandtl's one-equation model
      ii. Baldwin-Barth model
      iii. Spalart-Allmaras model
      iv. Rahman-Siikonen-Agarwal Model
   c. Two equation models
      i. k-ε models
         1. Standard k- ε model
         2. Realisable k- ε model
         3. RNG k- ε model
         4. Near-wall treatment
      ii. k-ω models
         1. Wilcox's k- ω model
         2. Wilcox's modified k- ω model
         3. SST k- ω model
         4. Near-wall treatment
      iii. Realisability issues
         1. Kato-Launder modification
         2. Durbin's realizability constraint
         3. Yap correction
         4. Realisability and Schwarz' inequality
2. Nonlinear eddy viscosity models
   a. Explicit nonlinear constitutive relation
      i. Cubic k-epsilon
      ii. Explicit algebraic Reynolds stress models (EARSM)
   b. v2-f models
3. Reynolds stress model (RSM)

In studies related to livestock buildings the most common models used are the two-equation models. In studies contacted by Animal Science Group of WUR the models are used are both the Realisable *k-ε* and the RNG- *k-ε* in combination with wall treatment functions.

4.3 Research groups

There are many research groups worldwide that are using CFD techniques to investigate the natural ventilation of livestock buildings (table 1). It must be noticed that only in Denmark there are three different groups that are collaborating in this scientific field. One of the reasons for these extensive research takes place in Denmark is the state of the art experimental facilities and obviously the number of researchers. The work carried out in the University of Seoul in South Korea has to be mentioned as well since can be considered the most innovative one. Finally, since Dr. Norton moved from the UC Dublin to the Engineering Department of Harper Adams University College in United Kingdom it is expected that many research projects will be carried out in this University as well.

Table 1: Main research groups which are using CFD techniques to investigate natural ventilation of livestock buildings.

	Research or educational organization	**Country**
1	Technology and Food Science Unit, Agricultural Engineering, Institute for Agricultural and Fisheries Research (ILVO), Burg. Van Gansberghelaan 115 bus 1, 9820 Merelbeke, Belgium	Belgium
2	Department of Agricultural and Bioresource Engineering, University of Saskatchewan, Saskatoon, SK, Canada	Canada
3	Key Lab. in Bioenvironmental Engineering of Ministry of Agriculture, China Agricultural University, Beijing, China	China
4	Department of Civil Engineering, Aalborg University, Denmark	Denmark
5	Department of Engineering, Aarhus University, Blichers Alle 20, P.O. Box 50, 8830 Tjele, Denmark	Denmark
6	Royal Veterinary and Agricultural University, Gronnegaardsvej 8, DK-1870 Frederiksberg C, Denmark	Denmark
7	School of Agricultural Sciences, Department of Agriculture Crop Production and Rural Environment, University of Thessaly	Greece
8	Food Refrigeration and Computerised Food Technology (FRCFT), University College Dublin, National University of Ireland	Ireland
9	Institute of Agricultural Engineering, Faculty of Agriculture, Serbia	Serbia
10	Department of Rural Systems Engineering and Research Institute for Agriculture and Life Sciences	South Korea
11	Wageningen University and Research Center, Greenhouse Horticulture	The Netherlands
12	Cornell University, Ithaca, New York	USA
13	Department of Agricultural and Biosystems Engineering, University of Arizona, Tucson, Arizona	USA

4.3.1 Contact details

The contact details of the most important research groups regarding CFD studies of natural ventilated livestock buildings are given in Appendix 2.

5 Conclusions and recommendations

CFD models have been extensively used to investigate the natural ventilation of livestock buildings, specially the last three years. There are many research groups worldwide dealing with this scientific topic. The most active one is the research group from the Department of Engineering in the University of Aarhus in Denmark. The main research targets of these groups are to investigate the emission of pollutants from livestock buildings and to improve the ventilation performance of these buildings by suggesting design modifications.

In most of the studies found in the literature the validation of CFD models has been carried out using experimental data from wind tunnels, which means that almost only scale livestock buildings have been considered. In general the agreement between measured data and computational results is good. There is not any information if the conclusions provided by the research studies, have been used by the companies to improve the design of their buildings.

Till now there are a few handbooks with guidelines and best practices for developing CFD models for modelling the air flow around and inside buildings. There are no particular guidelines for livestock buildings, even if the last years few review papers have been published. This indicates that each research group uses its own approach to simulate the environment of livestock buildings.

There are many CFD commercial codes available. The most common one is the ANSYS Fluent. Of course since the CFD models are totally deterministic models the same numerical methods, sub models and boundary conditions should provide the same results for a given computational domain and the choice for one or the other commercial codes could not make any difference.

Tracer gas techniques have been used to calculate the ventilation rate of livestock buildings but CFD models have not been used in advance to design the experimental setup. Usually the CFD models are used to validate the experiments and not to design them. CFD models can be a powerful tool to effectively design experiments with tracer gas or to setup sampling strategies. Validated CFD models can be used for further research and as tool to improve the design of current livestock buildings. There is a necessity for developing a methodology (protocol) for CFD simulation of livestock buildings. By using the same methodology the results obtained by different studies can be compared. Finally the CFD can be used to introduce new quality parameters for livestock buildings like those related to the spatial uniformity of air temperature, gas concentration of air velocity (uniformity index). The developing of these indexes will be useful both for the companies, farmers and research.

Referring to the objectives it can be concluded that Several CFD-models are available for livestock building but only a few for naturally ventilated dairy barns. The majority of these studies has only be validated with data from windtunnel experiments. None of them appeared to be validated with real scale data. Validation with these data can be costly if new equipment has to be bought and installed. Combination with existing projects like the one on comparing the SF_6 traces method with the CO_2 balance method is possible and less costly. This can only be a validation on gas concentration predictions as no air velocity equipment is installed. It is therefore recommended to adapt the already exciting CFD model for dairy barns (Sapounas et al., 2012) and combine it with the data collected in the this project on tracer gas techniques.

Literature

Bartzanas, T.; C. Kittas; A. A. Sapounas and C. Nikita-Martzopoulou (2007). Analysis of airflow through experimental rural buildings: Sensitivity to turbulence models. *Biosystems Engineering* **97**(2): 229-239..

Bjerg, B. and M. Andersen (2010). NUMERICAL SIMULATION OF A PIT EXHAUSTS SYSTEM FOR REDUCTION OF AMMONIA EMISSION FROM A NATURALLY VENTILATED CATTLE BUILDING. XVIIth World Congress of the International Commission of Agricultural and Biosystems Engineering (CIGR), Québec City, Canada..

Bjerg, B.; G. Cascone; I.-B. Lee; T. Bartzanas; T. Norton; S.-W. Hong; I.-H. Seo; T. Banhazi; P. Liberati; A. Marucci and G. Zhang (2013a). Modelling of ammonia emissions from naturally ventilated livestock buildings - Part three: CFD modelling. *Biosystems Engineering* **will be published**..

Bjerg, B.; P. Liberati; A. Marucci; G. Zhang; T. Banhazi; T. Bartzanas; G. Cascone; I.-B. Lee and T. Norton (2013b). Modelling of ammonia emissions from naturally ventilated livestock buildings - Part two: Air change modelling. *Biosystems Engineering* **will be published**..

Bjerg, B.; T. Norton; T. Banhazi; G. Zhang; T. Bartzanas; P. Liberati; G. Cascone; I.-B. Lee and A. Marucci (2013c). Modelling of ammonia emissions from naturally ventilated livestock buildings - Part one: Ammonia release modelling. *Biosystems Engineering* **will be published**..

Bjerg, B.; K. Svidt; G. Zhang; S. Morsing and J. O. Johnsen (2002). Modeling of air inlets in CFD prediction of airflow in ventilated animal houses. *Computers and Electronics in Agriculture* **34**(1-3): 223-235..

De Jong, T. and G. P. A. Bot (1992). Air exchange caused by wind effects through (window) openings distributed evenly on a quasi-infinite surface. *Energy and Buildings* **19**(2): 93-103..

De Paepe, M.; J. G. Pieters; W. M. Cornelis; D. Gabriels; B. Merci and P. Demeyer (2012). Airflow measurements in and around scale model cattle barns in a wind tunnel: Effect of ventilation opening height. *Biosystems Engineering* **113**(1): 22-32..

Ecim-Djuric, O. and G. Topisirovic (2010). Energy efficiency optimization of combined ventilation systems in livestock buildings. *Energy and Buildings* **42**(8): 1165-1171..

Etheridge, D. (2011). Natural Ventilation of Buildings: Theory, Measurement and Design, John Wiley & Sons, Ltd..

Franke, J.; A. Hellsten; H. Schlünzen and B. Carissimo (2007) Best practice guideline for the CFD simulation of flows in the urban environment. University of Hamburg, Meteorological Institute, Centre for Marine and Atmospheric Sciences.

Garcia Sagrado, A. P.; J. van Beeck; P. Rambaud and D. Olivari (2002). Numerical and experimental modelling of pollutant dispersion in a street canyon. *Journal of Wind Engineering and Industrial Aerodynamics* **90 (4-5)**(4-5): 321-339..

Gebremedhin, K. G. and B. X. Wu (2003). Characterization of flow field in a ventilated space and simulation of heat exchange between cows and their environment. *Journal of Thermal Biology* **28**(4): 301-319..

Gerdes, F. and D. Olivari (1999). Analysis of pollutant dispersion in an urban street canyon. *Journal of Wind Engineering and Industrial Aerodynamics* **82**(1-3): 105-124..

Hong, S.; I. Lee; H. Hwang; I. Seo; J. Bitog; K. Kwon; J. Song; O. Moon; K. Kim and H. Ko (2011a). CFD modelling of livestock odour dispersion over complex terrain, part I: Topographical modelling. *Biosystems Engineering* **108**(3): 253-264..

Hong, S.; I. Lee; H. Hwang; I. Seo; J. Bitog; K. Kwon; J. Song; O. Moon; K. Kim; H. Ko and S. Chung (2011b). CFD modelling of livestock odour dispersion over complex terrain, part II: Dispersion modelling. *Biosystems Engineering* **108**(3): 265-279..

Kwon, H. J.; C. H. Lee; E. J. Choi; J. Y. Song; B. C. Heinze and J. Y. Yoon (2010). Optofluidic device monitoring and fluid dynamics simulation for the spread of viral pathogens in a livestock environment. *Journal of Environmental Monitoring* **12**(11): 2138-2144..

Kwon, K. S.; I. B. Lee; H. T. Han; C. Y. Shin; H. S. Hwang; S. W. Hong; J. P. Bitog; I. H. Seo and C. P. Han (2011). Analysing ventilation efficiency in a test chamber using age-of-air concept and CFD technology. *Biosystems Engineering* **110**(4): 421-433..

Li, Y. and H. Guo (2007). Evaluating the effect of computational time steps on livestock odor dispersion using a computation fluid dynamics model. *Transactions of the ASABE* **50**(6): 2199-2204..

Miguel, A. F.; N. J. Van De Braak; A. M. Silva and G. P. A. Bot (2001). Wind-induced airflow through permeable materials. Part II: Air infiltration in enclosures. *Journal of Wind Engineering and Industrial Aerodynamics* **89**(1): 59-72..

Mistriotis, A.; G. P. A. Bot; P. Picuno and G. Scarascia-Mugnozza (1997). Analysis of the efficiency of greenhouse ventilation using computational fluid dynamics. *Agricultural and Forest Meteorology* **85**(3-4): 217-228..

Norton, T.; J. Grant; R. Fallon and D. W. Sun (2009). Assessing the ventilation effectiveness of naturally ventilated livestock buildings under wind dominated conditions using computational fluid dynamics. *Biosystems Engineering* **103**(1): 78-99..

Norton, T.; J. Grant; R. Fallon and D. W. Sun (2010a). Assessing the ventilation performance of a naturally ventilated livestock building with different eave opening conditions. *Computers and Electronics in Agriculture* **71**(1): 7-21..

Norton, T.; J. Grant; R. Fallon and D. W. Sun (2010b). A computational fluid dynamics study of air mixing in a naturally ventilated livestock building with different porous eave opening conditions. *Biosystems Engineering* **106**(2): 125-137..

Norton, T.; J. Grant; R. Fallon and D. W. Sun (2010c). Improving the representation of thermal boundary conditions of livestock during CFD modelling of the indoor environment. *Computers and Electronics in Agriculture* **73**(1): 17-36..

Norton, T.; J. Grant; R. Fallon and D. W. Sun (2010d). Optimising the ventilation configuration of naturally ventilated livestock buildings for improved indoor environmental homogeneity. *Building and Environment* **45**(4): 983-995..

Norton, T.; D. W. Sun; J. Grant; R. Fallon and V. Dodd (2007). Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: A review. *Bioresource Technology* **98**(12): 2386-2414..

Ogink, N. W. M.; J. Mosquera and J. M. G. Hol (2011) Protocol voor meting van ammoniakemissie uit huisvestingssystemen in de veehouderij 2010. Rapport 454, Wageningen UR Livestock Research, Lelystad.

Pedersen, S. and K. Sällvik (2002) Heat and moisture production at animal and house levels. 4th Report of Working Group on Climatization of Animal Houses, International Commission of Agricultural Engineering, Section II.

Rong, L.; B. Elhadidi; H. E. Khalifa; P. V. Nielsen and G. Zhang (2011). Validation of CFD simulation for ammonia emissions from an aqueous solution. *Computers and Electronics in Agriculture* **75**(2): 261-271..

Salim, S. M.; K. C. Ong and S. C. Cheah (2011). Comparison of RANS, URANS and LES in the Prediction of Airflow and Pollutant Dispersion. World Congress on Engineering and Computer Science, October 19-21, 2011, San Francisco, USA..

Sapounas, A. A. and J. B. Campen (2003). Simulating the Pollutant Emissions from Greenhouse using CFD: Validation of four turbulence models. Wageningen, The Netherlands, IMAG..

Sapounas, A. A.; J. B. Campen; M. C. J. Smits and H. J. C. van Dooren (2009). Simulating the effect of forced pit ventilation on ammonia emission from a naturally ventilated cow house with CFD. 4th European Conference on Precision Livestock Farming, Wageningen, The Netherlands..

Sapounas, A. A.; H. J. C. van Dooren and M. C. J. Smits (2012). Natural ventilation of commercial dairy cow houses: simulating the effect of roof shape using CFD. International Conference of Agricultural Engineering. AGENG-2012, Valencia, Spain..

Seo, I.-H.; I.-B. Lee; S.-H. Kwon; H.-S. Hwang; S.-W. Hong; B. J.P.; J.-I. Yoo; K.-S. Kwon and Y.-H. Kim (2009). Numerical investigation of the pig house to prevent pig respiratory disease according to ventilation system. The 2009 CIGR International Symposium of the Australian Society for Engineering in Agriculture, The Chifley at Lennons, Brisbane, Queensland, Australia..

Shen, X.; G. Zhang and B. Bjerg (2012). Comparison of different methods for estimating ventilation rates through wind driven ventilated buildings. *Energy and Buildings* **54**: 297-306..

Stavrakakis, G. M.; M. K. Koukou; M. G. Vrachopoulos and N. C. Markatos (2008). Natural cross-ventilation in buildings: Building-scale experiments, numerical simulation and thermal comfort evaluation. *Energy and Buildings* **40**(9): 1666-1681..

Sun, H.; H. Keener; W. Deng and J. F. Michel (2004). Development and validation of 3-D CFD models to simulate airflow and ammonia distribution in a high-rise hog building during summer and winter conditions. *CIGR Journal of Scientific Research and Development* **Manuscript BC 04 004. Vol. VI.**.

Tominaga, Y.; A. Mochida; R. Yoshie; H. Kataoka; T. Nozu; M. Yoshikawa and T. Shirasawa (2008). AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of Wind Engineering and Industrial Aerodynamics* **96**(10-11): 1749-1761..

Tropea, C.; A. L. Yarin and J. F. Foss (2007). Handbook of Experimental Fluid Dynamics. Berlin Heidelberg, Springer..

Versteeg, H. K. and W. Malalasekera (1995). An introduction to computational fluid dynamics : the finite volume method. Harlow, Longman..

Wu, B. and K. G. Gebremedhin (2001). CFD development and simulation of flow fields in ventilated spaces with multiple occupants. *Transactions of the American Society of Agricultural Engineers* **44**(6): 1839-1850..

Wu, W.; J. Zhai; G. Zhang and P. V. Nielsen (2012a). Evaluation of methods for determining air exchange rate in a naturally ventilated dairy cattle building with large openings using computational fluid dynamics (CFD). *Atmospheric Environment* **63**: 179-188..

Wu, W.; G. Zhang; B. Bjerg and P. V. Nielsen (2012b). An assessment of a partial pit ventilation system to reduce emission under slatted floor - Part 2: Feasibility of CFD prediction using RANS turbulence models. *Computers and Electronics in Agriculture*..

Yu, H.; C. H. Hou and C. M. Liao (2002). Scale model analysis of opening effectiveness for wind-induced natural ventilation openings. *Biosystems Engineering* **82**(2): 199-207..

Yu, Z. (2010). Development of a Livestock Odour Dispersion Model. Division of Environmental Engineering. Saskatoon, University of Saskatchewan**:** 191..

Yu, Z.; H. Guo and C. Laguë (2010). Livestock odor dispersion modeling: A review. *Transactions of the ASABE* **53**(4): 1231-1244..

Zhang, G.; S. Pedersen and P. Kai (2010). Uncertainty analysis of using CO2 production models by cows to determine ventilation rate in naturally ventilated buildings. XVIIthWorld Congress of the International Commission of Agricultural and Biosystems Engineering (CIGR), Québec City, Canada, Canadian Society for Bioengineering (CSBE/SCGAB)..

Appendices

Appendix 1. Recent publications related to CFD applications in natural ventilation studies of livestock buildings from research groups

	Research or Educational organization	Country	Year of publication	Article title	Researchers	Reference
1	Technology and Food Science Unit Agricultural Engineering, Institute for Agricultural and Fisheries Research (ILVO), Burg. Van Gansberghelaan 115 bus 1, 9820 Merelbeke, Belgium	Belgium	2012	Airflow measurements in and around scale model cattle barns in a wind tunnel: Effect of ventilation opening height	De Paepe , M., Pieters, J.G., Cornelis, W.M., Gabriels, D., Merci, B., Demeyer, P	(De Paepe et al., 2012)
2	Department of Agricultural and Bioresource Engineering, University of Saskatchewan, Saskatoon, SK, Canada	Canada	2010	LIVESTOCK ODOR DISPERSION MODELING: A REVIEW	Z. Yu, H. Guo, C. Laguë	(Yu et al., 2010)
3	Key Lab. in Bioenvironmental Engineering of Ministry of Agriculture, China Agricultural University, Beijing, China	China	2007	Evaluating the effect of computational time steps on livestock odor dispersion using a computation fluid dynamics model	Y. Li, H. Guo	(Li and Guo, 2007)
4	Department of Civil Engineering, Aalborg University, Denmark	Denmark	2011	Validation of CFD simulation for ammonia emissions from an aqueous solution	Rong, L., Elhadidi, B., Khalifa, H.E., Nielsen, P.V., Zhang, G	(Rong et al., 2011)

5	Department of Engineering, Aarhus University, Blichers Alle 20, P.O. Box 50, 8830 Tjele, Denmark	Denmark	2012	Evaluation of methods for determining air exchange rate in a naturally ventilated dairy cattle building with large openings using computational fluid dynamics (CFD)	Wu, W., J. Zhai, G. Zhang and P.V. Nielsen	(Wu et al., 2012a)
6	Department of Engineering, Aarhus University, Department of Large Animal Sciences, Faculty of Life Sciences, University of Copenhagen, Department of Building Technology and Structural Engineering, Aalborg University	Denmark	2012	An assessment of a partial pit ventilation system to reduce emission under slatted floor – Part 2: Feasibility of CFD prediction using RANS turbulence models	Wentao Wu, Guoqiang Zhang, Bjarne Bjerg , Peter V. Nielsen	(Wu et al., 2012b)
7	Royal Veterinary and Agricultural University, Gronnegaardsvej 8, DK-1870 Frederiksberg C, Denmark	Denmark	2002	Modeling of air inlets in CFD prediction of airflow in ventilated animal houses	B. Bjerg , K. Svidt, G. Zhang, S. Morsing J.O. Johnsen c	(Bjerg et al., 2002)
8	Department of Engineering, Faculty of Sciences and Technology, University of Aarhus, Blichers Allé 20, 8830 Tjele, Denmark.	Denmark	2012	Comparison of different methods for estimating ventilation rates through wind driven ventilated buildings	Shen, X., Zhang, G. , Bjerg, B.	(Shen et al., 2012)
9	School of Agricultural Sciences, Department of Agriculture Crop Production and Rural Environment, University of Thessaly	Greece	2007	Analysis of airflow through experimental rural buildings: Sensitivity to turbulence models	Bartzanas, T., Kittas, C., Sapounas, A.A., Nikita-Martzopoulou, Ch.	(Bartzanas et al., 2007)
10	Food Refrigeration and Computerised Food Technology (FRCFT), University College Dublin, National University of Ireland	Ireland	2010	A computational fluid dynamics study of air mixing in a naturally ventilated livestock building with different porous eave opening conditions	Toma´s Norton, Jim Grant, Richard Fallon, Da-Wen Sun	(Norton et al., 2010b)

11	Food Refrigeration and Computerised Food Technology (FRCFT), University College Dublin, National University of Ireland	Ireland	2010	Improving the representation of thermal boundary conditions of livestock during CFD modelling of the indoor environment	Tomás Nortona, Jim Grant, Richard Fallon, Da-Wen Suna	(Norton et al., 2010c)
12	Food Refrigeration and Computerised Food Technology (FRCFT), University College Dublin, National University of Ireland	Ireland	2010	Optimising the ventilation configuration of naturally ventilated livestock buildings for improved indoor environmental homogeneity	Toma´ s Norton, Jim Grant, Richard Fallon, Da-Wen Sun	(Norton et al., 2010d)
13	Food Refrigeration and Computerised Food Technology (FRCFT), University College Dublin, National University of Ireland	Ireland	2010	Assessing the ventilation performance of a naturally ventilated livestock building with different eave opening conditions	Norton, T,Grant, J.,Fallon, R.,Sun, D.-W.	(Norton et al., 2010a)
14	Food Refrigeration and Computerised Food Technology (FRCFT), University College Dublin, National University of Ireland	Ireland	2009	Assessing the ventilation effectiveness of naturally ventilated livestock buildings under wind dominated conditions using computational fluid dynamics	Norton, T.,Grant, J., Fallon, R., Sun, D.-W.	(Norton et al., 2009)
15	Food Refrigeration and Computerised Food Technology (FRCFT), University College Dublin, National University of Ireland	Ireland	2007	Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: A review	Norton, T., Sun, D.-W. ,Grant, J., Fallon, R., Dodd, V.	(Norton et al., 2007)
16	Institute of Agricultural Engineering, Faculty of Agriculture, Serbia	Serbia	2010	Energy efficiency optimization of combined ventilation systems in livestock buildings	Ecim-Djuric, O. and Topisirovic, G.	(Ecim-Djuric and Topisirovic, 2010)

17	Department of Rural Systems Engineering and Research Institute for Agriculture and Life Sciences	South Korea	2011	CFD modelling of livestock odour dispersion over complex terrain, part I: Topographical modelling	S. Hong , I. Lee, H. Hwang, I. Seo, J. Bitog, K. Kwon, J. Song, O. Moon, K. Kim, H. Koe	(Hong et al., 2011a)
18	Department of Rural Systems Engineering and Research Institute for Agriculture and Life Sciences	South Korea	2011	CFD modelling of livestock odour dispersion over complex terrain, part II: Dispersion modelling	S. Hong , I. Lee, H. Hwang, I. Seo, J. Bitog, K. Kwon, J. Song, O. Moon, K. Kim, H. Koe, S. Chung	(Hong et al., 2011b)
19	Cornell University, Ithaca, New York	USA	2001	CFD DEVELOPMENT AND SIMULATION OF FLOW FIELDS IN VENTILATED SPACES WITH MULTIPLE OCCUPANTS	Binxin Wu, K. G. Gebremedhin	(Wu and Gebremedhin, 2001)
20	Department of Agricultural and Biosystems Engineering, University of Arizona, Tucson	USA	2010	Optofluidic device monitoring and fluid dynamics simulation for the spread of viral pathogens in a livestock environment	Kwon, H. J. Lee, C. H. Choi, E. J. Song, J. Y. Heinze, B. C. Yoon, J. Y.	(Kwon et al., 2010)
21	WUR, Greenhouse Horticulture	The Netherlands	2009	Simulating the effect of forced pit ventilation on ammonia emission from a naturally ventilated cow house with CFD	Sapounas, A.A., J.B. Campen, M.C.J. Smits and H.J.C. Van Dooren	(Sapounas et al., 2009)
22	WUR, Greenhouse Horticulture	The Netherlands	2012	Natural ventilation of commercial dairy cow houses: simulating the effect of roof shape using CFD	Sapounas, A.A., H.J.C. Van Dooren and M.C.J. Smits	(Sapounas et al., 2012)

Appendix 2. Most important research groups worldwide regarding CFD application in livestock buildings

	Research or Educational organization	Country	contact details	url	email of reseracher
1	Technology and Food Science Unit Agricultural Engineering, Institute for Agricultural and Fisheries Research (ILVO)	Belgium	Burg. Van Gansberghelaan 115, bus 1 Merelbeke, 9820 Belgium	http://www.ilvo.vlaanderen.be/	merlijn.depaepe@ilvo.vlaanderen.be
2	Department of Agricultural and Bioresource Engineering, University of Saskatchewan, Saskatoon, SK, Canada	Canada	College of Engineering University of Saskatchewan, 57 Campus Drive Saskatoon, SK S7N 5A9	http://www.engr.usask.ca	huiqing.guo@usask.ca
3	Key Lab. in Bioenvironmental Engineering of Ministry of Agriculture, China Agricultural University, Beijing, China	China	Key Lab on Bioenvironmental Engineering of the Ministry of Agriculture, P. R. China China Agricultural University Address: 17 QingHua Dong Lu Road, Haidian District, Beijing 100083		
4	Department of Civil Engineering, Aalborg University, Denmark	Denmark	Department of Civil Engineering · Sohngaardsholmsvej 57 · DK-9000 Aalborg East	http://www.civil.aau.dk/	

5	Department of Engineering, Aarhus University, Denmark	Denmark	Department of Engineering, Aarhus University, Blichers Alle 20, P.O. Box 50, 8830 Tjele, Denmark	http://eng.au.dk/en/	Guoqiang.Zhang@agrsci.dk
6	Department of Large Animal Sciences, Faculty of Life Sciences, University of Copenhagen	Denmark	University of Copenhagen Faculty of Science Bülowsvej 17 1870 Frederiksberg C	http://healthsciences.ku.dk/home/	Bjarne Berg: bsb@sund.ku.dk
7	School of Agricultural Sciences, Department of Agriculture Crop Production and Rural Environment, University of Thessaly	Greece	Technology Park of Thessaly 1st Industrial Area, GR 385 00, Volos	http://www.cereteth.gr/	bartzanas@cereteth.gr
8	Food Refrigeration and Computerised Food Technology (FRCFT), University College Dublin, National University of Ireland	Ireland	UCD School of Biosystems Engineering Room 3.03a Agriculture & Food Science Centre, Belfield, Dublin 4.	http://www.harper-adams.ac.uk/ http://www.ucd.ie/refrig/	tnorton@harper-adams.ac.uk
9	Institute of Agricultural Engineering, Faculty of Agriculture, Serbia	Serbia	Institute of Agricultural Engineering, Faculty of Agriculture, Nemanjina 6, Belgrade - Zemun 11080, Serbia	http://www.agrif.bg.ac.rs/	gogi@agrif.bg.ac.rs

10	Department of Rural Systems Engineering and Research Institute for Agriculture and Life Sciences	South Korea	Laboratory of Aero-Environmental Engineering Department of Rural System Engineering College of Agriculture and Life Science Seoul National University San56-1, Silim-dong, Gwanak-gu, Seoul Republic of Korea	http://aeel.snu.ac.kr/	iblee@snu.ac.kr
11	Biological and Environmental Engineering Department, Cornell University, Ithaca, New York.	USA		http://bee.cornell.edu/	Kifle Gebremedhin, kggl@cornell.edu
12	Department of Agricultural and Biosystems Engineering, University of Arizona, Tucson	USA	Department of Agricultural & Biosystems Engineering, The University of Arizona, Tucson, Arizona 85721-0038, United States.	http://www.cals.arizona.edu/abe/	Jeong-Yeol Yoon, jyyoon@email.arizona.edu
13	WUR, Greenhouse Horticulture	The Netherlands	Wageningen UR – Glastuinbouw, Building 107, Droevendaalsesteeg 1, 6708 PB, Wageningen	www.glastuinbouw.wur.nl	athanasios.sapounas@wur.nl

Wageningen UR Livestock Research

Edelhertweg 15, 8219 PH Lelystad T 0320 238238 F 0320 238050

E info.livestockresearch@wur.nl I www.livestockresearch.wur.nl