

NEW METHODS IN EDUCATION OF HVAC FOR ERABEE

Pavel Kic, Milan Zajicek

Czech University of Life Sciences Prague; Academy of Science of the Czech Republic

kic@tf.czu.cz, zajicek@utia.cas.cz

Abstract. The paper describes the new approach to the education of the course focused on heating, ventilation and air-conditioning in the frame of MSc. study programs specialized on Agricultural and Biosystems Engineering. Special attention is paid to the new methods of individual study with e-learning, modern methods of designing with use of specialized and also commercial software and programs (e.g. CFD Fluent, OpenFOAM etc.). Seminars and laboratory work complete the education with practical or experimental measurements of all main microclimatic parameters (thermal state, purity of air, noise, illumination etc.) and individual designs prepared by the students during the semester and presented by the students in seminars of the group at the end of semester, before the examination period. Lectures and seminars cover all principal problems related to the buildings and equipment in agriculture and Biosystems engineering.

Keywords: agriculture, study program, course, microclimate, ventilation, design.

Introduction

Education specialized to agricultural engineering has a long tradition in the Czech Republic. During the last decades great changes of the economy and policy in the country resulted also in changes of the educational system. Many universities decided to prepare new study programmes according to the priorities and interests of young generation.

Currently, after the many changes provided during the nineties and the first years of this century, the Faculty of Engineering educates graduates for the whole of the agri-food sector including designing of technological equipment of buildings in food industry and agriculture, for road automobile transport and for trade and business involving machinery.

Three year Thematic Network entitled Education and Research in Biosystems or Agricultural and Biological Engineering in Europe; a Thematic Network (ERABEE TN) has connected Universities active in these problems from all European countries including several non EC countries. Participation helped to exchange the information and prepare some new ideas for development and better education in all active countries [1]. New curricula and methods for education and research were proclaimed.

The paper is focused on software equipment, which is used during the course focused on heating, ventilation and air-conditioning.

Materials and methods

Two kinds of software are discussed here. The first one is the Moodle. The Moodle is an Open Source Course Management System (CMS), also known as a Learning Management System (LMS) or a Virtual Learning Environment (VLE). It has become very popular among educators around the world as a tool for creating online dynamic web sites for their students [2]. To work, it needs to be installed on a web server somewhere, either on one of your own computers or one at a web hosting company. There are two ways the Moodle is used. The first is the place, where students can find the appropriate materials as, for example, texts and presentations with individual themes prepared by the teacher. The second way allows students to manage their own projects considering the education schema and individual projects.

The second kind of software used during the course is the Computer Fluid Dynamics CFD software. Students are familiarized with an Open source CFD package OpenFoam [3] and commercial CFD product Fluent [4]. Both of them are able to solve the wide range of problems from the HVAC field.

The plan of the exercises is prepared to lead the students through the whole cycle of the numerical analysis. It starts with the measurement in real buildings, where it is possible to see the measurement techniques and practical limits of the used devices, e.g., thermovision camera IR Flexcam Pro, mobile weather station, Ahlborn measuring instruments and sensors with ALMEMO connectors for measurement of all main parameters needed for this area of research and education, e.g., temperature sensors with NiCr-Ni thermocouples, Pt100 and NTC, thermowires and compensation lines, capacitive

humidity sensors, psychrometer, thermoanemometers, air flow, comfort index measurement in room conditions, WBGT measurement, measuring thermal transmittance and heart flow for building physics, sensors for measuring moisture in materials, surface temperature, soil humidity measurement, global radiation, light, luminous intensity, gas concentration in air (oxygen, carbon dioxide, carbon monoxide, ammonia, hydrogen sulphide, etc.

The student projects are based on real dimensions of the existing buildings or machines and the needed simplifications are made to the corresponding geometries and models. The students are led to use the OpenFoam software for solving them. Some projects are also solved using the Fluent and the comparison of the results for such cases is done and discussed.

The geometry which corresponds to the transversal shape of the broiler house [5] can be seen at Figure 1. It is a rectangle with two inlets (windows on the right side) and one outlet (the ventilator on the left side), it corresponds to a more generalized case of transversal ventilation which is often used as a ventilation schema at animal houses.

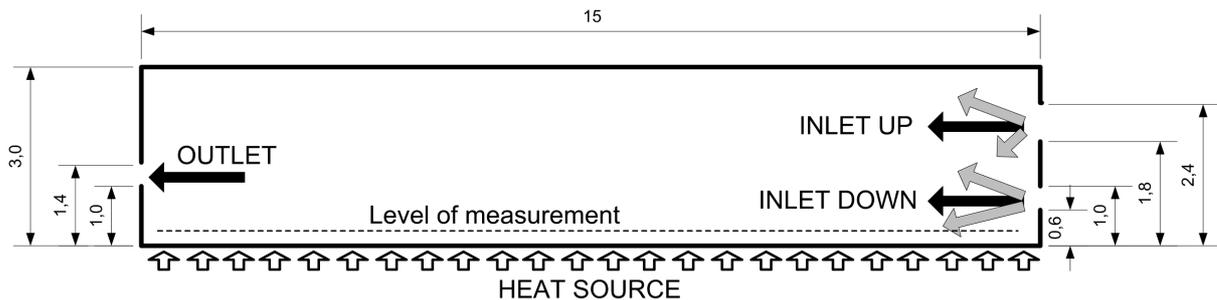


Fig. 1. **Typical geometry used as a student project case:** All distances are given in meters. The level of measurement is the height above the floor, where the measurement at real objects was made. The heat source represents the heat generated by poultry

It has to be mentioned, that there are completely different needs for the ventilation system during the summer and winter period. During the winter, the internal space of the broiler house is heated by additional heat sources and the amount of incoming fresh air has to be preheated before the fresh air reaches the zone of animals to prevent their undercooling. It means that the amount of the incoming air is relatively small and is limited by the critical values of humidity and pollutant (CO_2 , NH_3) concentration. During the summer period the amount of air depends mainly on the maximum temperature, which is reached at the zone of animals to prevent overheating of them.

It is clear, that the shape of the flow field is influenced by many parameters, but one of the most important is the direction of the incoming air stream. Therefore, the analysis of the temperature profile at the level of measurement can be very important and can be done relatively easily with CFD and it is a typical exercise for students' work.

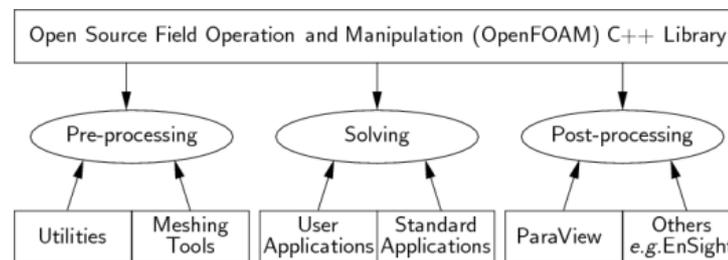


Fig. 2. **OpenFoam toolbox structure:** All data processing and manipulations can be done using a part of this toolbox either with benefits of external programs (e.g., mesh generators, visualisation products, user defined inputs of data ...)

The structure of the OpenFoam software can be seen in Figure 2. The OpenFoam geometry definition file is shown in Figure 3. It is clear, that the definition is a poor text file with the definition of points, edges and boundary parts. The physical properties, solver setup and other parameters of the model have to be defined in a similar way. This attempt is not so user friendly, but is very useful, for understanding the problem and thinking about physical background of the given case. For example,

the definition of any physical unit is given as a value and vector which represents the SI expression in basic units. For example, the pressure is given in Pascals $p=72$ (Pa) written as: $p [1 -1 -2 0 0 0] 72$; where the vector after p corresponds to the power of the basic units at the transcription of the derivative unit. For Pascals the basic unit transcription is $\text{kg}\cdot\text{m}^{-1}\cdot\text{s}^{-2}\cdot\text{K}^0\cdot\text{kmol}^0\cdot\text{A}^0\cdot\text{cd}^0$. This attempt is clear from mathematical point of view, but is not so user friendly.

When you start to solve the same case using the Fluent software, there is no need for editing any text files, but you have to be familiar with a CAD-like pre-processor Gambit, which is a tool that can import the complex geometry from almost all CAD systems and prepare it for the CFD analysis. Gambit also can stay as an independent geometry and mesh generator, which is the way it can be used for preparing the case shown here. The main benefit of Fluent is only one graphical interface, which is used for all activities you like to do with the software, from defining the boundary conditions up to visualisation.

Once the model is meshed with fully defined boundary conditions, it is possible to run the solver and wait for the results. The shape of solution is obviously dependent on the solver algorithm and therefore it is needed to compare the calculated values with the measurement. This part of work is usually very time consuming and takes a lot of energy. When the correct model is chosen and is converged, here is the time for starting with analysis of the flow variants. The case from Figure 1 can be an example analyzed according to different inlet stream directions.

```

FoamFile
{
  version      2.0;
  format       ascii;
  class        dictionary;
  object       blockMeshDict;
}

convertToMeters 1;

vertices
(
  (0 0 0)
  (0 0.6 0)
  (0 1 0)
  (0 1.4 0)
  -
  -
  -
  (15 1 .1)
  (15 1.4 .1)
  (15 1.8 .1)
  (15 2.4 .1)
  (15 3 .1)
);

blocks
(
  hex (0 7 8 1 14 21 22 15) (100 10 1) simpleGrading (1 1 1)
  hex (1 8 9 2 15 22 23 16) (100 10 1) simpleGrading (1 1 1)
  hex (2 9 10 3 16 23 24 17) (100 10 1) simpleGrading (1 1 1)
  hex (3 10 11 4 17 24 25 18) (100 10 1) simpleGrading (1 1 1)
  hex (4 11 12 5 18 25 26 19) (100 10 1) simpleGrading (1 1 1)
  hex (5 12 13 6 19 26 27 20) (100 10 1) simpleGrading (1 1 1)
);

edges
(
);

patches
(
  wall fixedWalls
  (
    (0 7 21 14)
    (7 21 22 8)
    -
    -
    -
    (4 18 19 5)
    (5 19 20 6)
    (6 13 27 20)
  )
  patch
  outlet
  (
    (2 16 17 3)
  )
  patch
  inlet-down
  (
    (8 22 23 9)
  )
  patch
  inlet-up
  (
    (11 25 26 12)
  )
  empty frontAndBack
  (
    (0 7 8 1)
    (1 8 9 2)
    -
    -
    -
    (18 25 26 19)
    (19 26 27 20)
  )
);
mergePatchPairs
(
);

```

Fig. 3. OpenFoam geometry source code corresponding to Fig. 1: The three vertical dots mean the lines excluded from the file to decrease the size of the picture. There are 28 vertices in this geometry in the matter of fact

Results and discussion

There are a lot of parameters, which can be changed even on relatively simple geometry and boundary conditions presented here. The input stream direction is the only parameter which is changed during the simulation. All other parameters of the model are the same at all calculations which are discussed below. Figure 4 shows the velocity contours for two inlet configurations. The upper simulation corresponds with the situation, when only the up inlet is open and the direction of the flow

is 30° . The bottom result shows the velocity field when both inlets are used, the flow from the down inlet has a direction 30° and the flow from the up inlet has a direction 60° . The angle is measured from the right wall, e.g., 0° means the direction down and 180° means the direction UP.

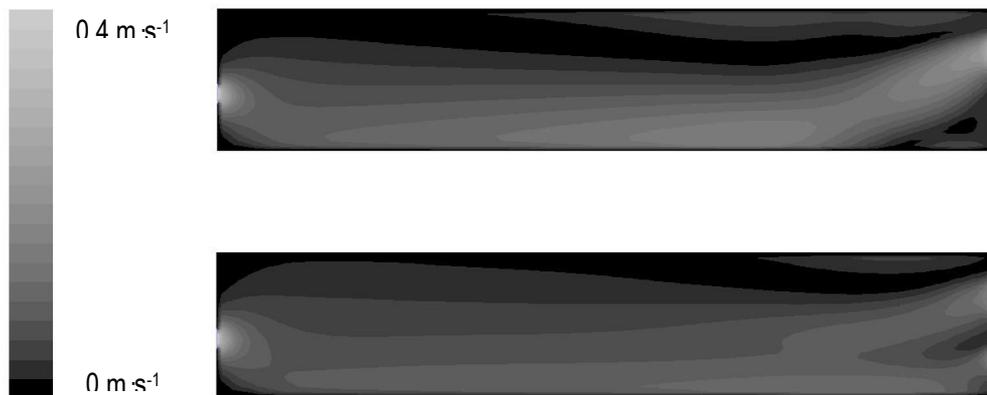


Fig. 4. **Comparison of two velocity profiles for the same flowrate.** The upper corresponds to only the upper window open and the direction of the input stream 60° . The lower case shows the change when both inlets are open and the input stream of the down inlet has the direction 30° .

It is seen, that there are uncountable amounts of variants that can arise, which can be solved and therefore it is very important to find some criteria, which can give a brighter idea about comparison of them. Fig. 5 illustrates the comparison of the velocity and temperature profiles for 10 cases with changeable inlet position and directions of the inlet flow. All profiles are obtained from the same line (level 0.3 m above the floor).

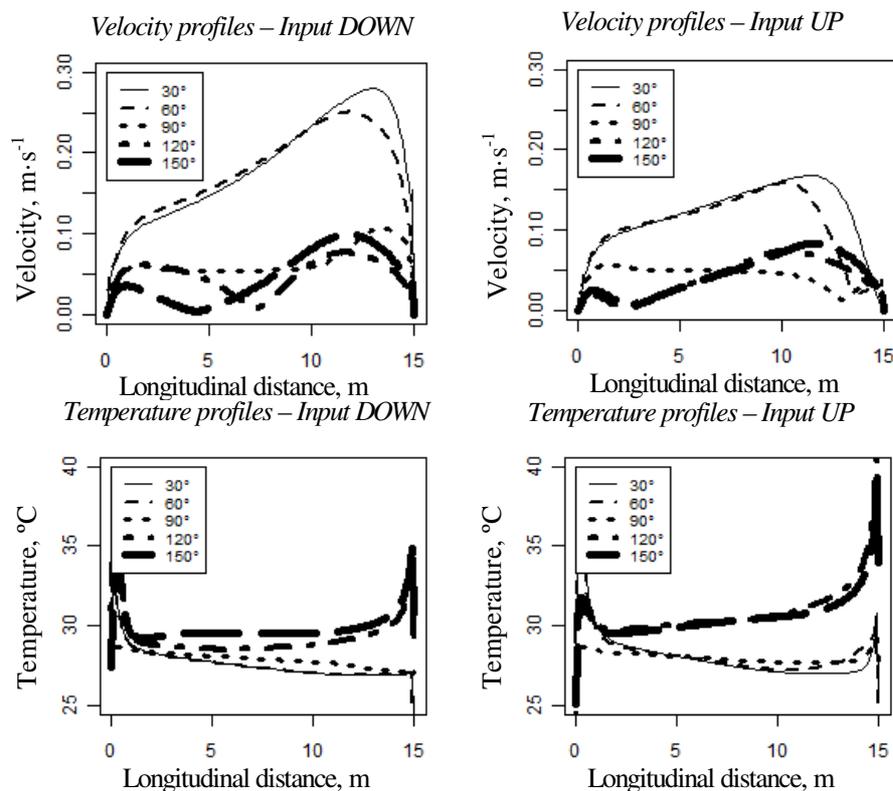


Fig. 5. **Typical results of CFD analysis of variants.** The comparison of the velocity and temperature profiles at the level of measurement. The angle of the input stream is the parameter of these graphs. 90° degrees mean that the input flow is horizontal. Lower angles mean that the intake air goes down; higher values mean that it goes up

There can be observed some basic properties of the system behavior, which has to correspond with the physical principles. For example, it is clean, that the input stream at the same direction causes higher velocities at the animal zone when it is coming from the down inlet, than that, which is coming from the upper one. It is clear, that there is a greater rise of temperatures at the flow with smaller velocity or with recirculation.

This set of pictures is a typical kind of a result, which can be expected as an output from the student projects. It should be also mentioned, that the amount of the possible applications of CFD simulations in Biosystems or Agricultural and Biological Engineering is practically inexhaustible. Every process which has a deal with fluid flow and energy transport can be simulated (with appropriate simplifications) and the results can be used for the process improving.

A short comparison of OpenFoam and Fluent is made in Table 1. It is not easy to compare the detailed level of models and solvers implemented inside each of these software products. But there can be one general advice given. If the user is a scientist or student, who has time to prepare his model in every detail and aspect, and likes to learn about the principles of CFD, the OpenFoam is ideal for him. It is free and open.

If the user needs to solve complex and large problems with precise pre- and post-processing or if he needs to spare his time, the Fluent is only choice. Not free, but a very user friendly arranged system with a support from ANSYS brings well done user guides and lot of functions, which are accessible through one graphical user interface, with included a large material properties database and interfaces for using this software together with many common CAD systems.

Table 1

Comparison of OpenFoam and Fluent

Feature	Open Foam	Fluent
Main characteristics	Toolbox which can be used for making and solving the problem (set of libraries)	Complex GUI driven software with robust pre- and post-processing
Licensing	Free download, install and use	Commercial license, pricing per process, lower prices for academic and research institutions.
Pre-processing	Similar to the programming code	User friendly geometrical CAD like pre-processor Gambit with integrated mesh generator.
Mesh generation	As a part of the definition code	Automatic widely configurable mesh generator
Solvers	A number of independent solvers	Solvers easily configurable from GUI
Postprocessing	ParaFoam – the graphical frontend which can visualize the calculated data.	Very robust and professionally equipped, included directly into the Fluent
Library of physical properties of materials	no	yes
Time to make a numerical model and solve the comparable problem	days	hours

Conclusions

The education process of the Agricultural and Biosystems Engineering has a lot of aspects, which can be supplementary supported by CFD simulations. For students it is very useful to recognise the possibilities of numerical simulation of the thermal analysis and fluid flow during the course. The knowledge of the basic principles of software such as OpenFOAM can be very useful for the diploma thesis of the students and also for their professional lives.

References

1. Kic P., Jurča V. Restructuring agricultural/biosystems engineering studies and employability in the Czech Republic. In: Restructuring agricultural/biosystems engineering studies and employability in Europe. Athens, 2006, pp. 30-35, ISBN 960-89313-0-4
2. Vanova T., Vanova A. Moodle in the Net, Pragodata Consulting, Prague, 2009, ISBN 978-80-7399-447-1 (in Czech)
3. Fluent brochure. [online] [10.03.2011]. Available at: <http://www.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/brochure/ansys-fluent-brochure.pdf>
4. The open source CFD toolbox OpenFoam – User guide. [online] [10.03.2011]. Available at: <http://www.openfoam.com/docs/user/>
5. Kic P., Zajicek M., Broiler house ventilation, CFD analysis of variants. In: Proceedings of the 3rd International Congress on Information and Communication Technologies in Agriculture, Food, Forestry & Environment (ITAFFE'10), edit by Zeynel Cebeci, Ondokuz Mayıs University, Samsun, Turkey, 2010, pp. 126-131.