

CFD study on the effects of boundary conditions on air flow through an air-cooled condenser

Zdeněk Sumara^{1*}, Michal Šochman¹

¹CTU in Prague, Faculty of Mechanical Engineering, Department of Fluid Mechanics and Thermodynamics, Czech Republic

Abstract. This study focuses on the effects of boundary conditions on effectiveness of an air-cooled condenser (ACC). Heat duty of ACC is very often calculated for ideal uniform velocity field which does not correspond to reality. Therefore, this study studies the effect of wind and different landscapes on air flow through ACC. For this study software OpenFOAM was used and the flow was simulated with the use of RANS equations. For verification of numerical setup a model of one ACC cell with dimensions of platform 1.5x1.5 [m] was used. In this experiment static pressures behind fan and air flows through a model of surface of condenser for different rpm of fan were measured. In OpenFOAM software a virtual clone of this experiment was built and different meshes, turbulent models and numerical schemes were tested. After tuning up numerical setup virtual model of real ACC system was built. Influence of wind, landscape and height of ACC on air flow through ACC has been investigated.

1 Introduction

In standard design of air-cooled condenser (ACC) the influence of wind is not considered, as well as the surrounding terrain or other aspects that certainly affects the effectiveness of ACC [1]. This paper presents results based on CFD model of real smaller 2 cells of ACC. In CFD model the influence of wind and surrounding buildings is considered. From resulting data, the change of flow rate with the change of boundary conditions is evaluated as well as the influence of uniformity of velocity field on condenser on the effectiveness of ACC.

The study was provided with the software OpenFOAM and CFD model was validated on a smaller model of one cell ACC [2-3].

This study shows that the change of air volume flow with the change of boundary conditions can be very significant. The change of air volume flow through condenser can reach 25 [%] with the change of wind velocity. The influence of surrounding building can change the volume flow by about 10 [%]

2 Validation of CFD model

Validation of used CFD model is the step that cannot be omitted in any CFD study. For this study a model of one ACC cell was built with dimensions 1.5 x 1.5 [m]. The scheme of this model can be seen in Fig. 1.

Single row condenser that is used in modern ACC was replaced by screens and honeycombs in the model. Pressure drop of screen and honeycombs was measured in a wind tunnel.

In CFD a virtual image of this model was made. For simulation of pressure drop of screens and honeycombs a

porous medium approach was used. Fan was simulated as a pressure jump. In Figure 2 below there is a computational domain.

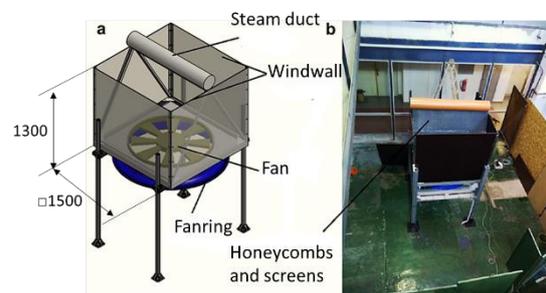


Fig. 1. Model of one cell ACC

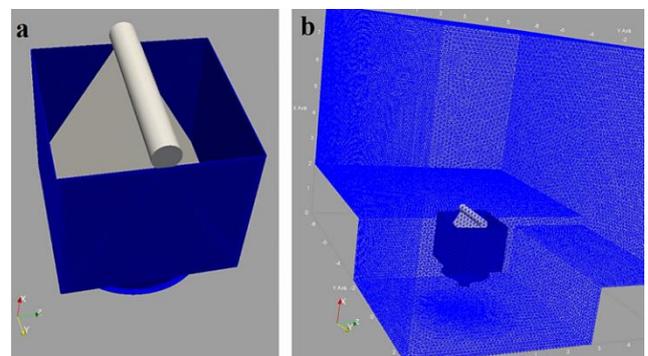


Fig. 2. Computational domain of CFD model

Static pressure behind the fan and air volume flow through screen for several fan rpm were measured on the experimental model. For measurement of air flow a multi constant anemometer (MCCA) was used. In the following figure there is a picture from measurement of

* Corresponding author: zdenek.sumara@fs.cvut.cz

air volume flow for one position. Air flow and static pressure behind the fan were measured for 237, 444, 591 and 884 [r/min] of the fan.

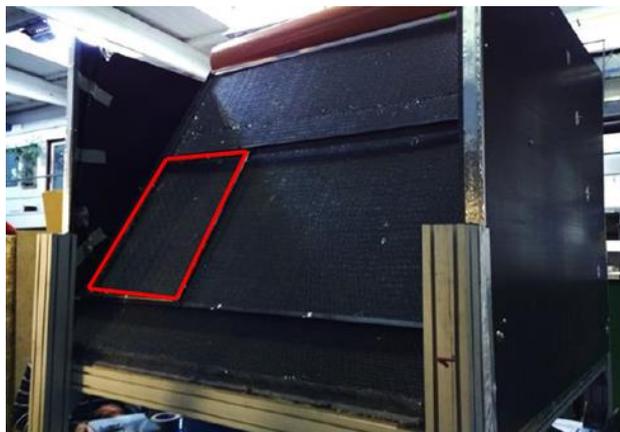


Fig. 3. One position of MCCA

Every position in which air flow rate was measured is in the scheme in Figure 4.

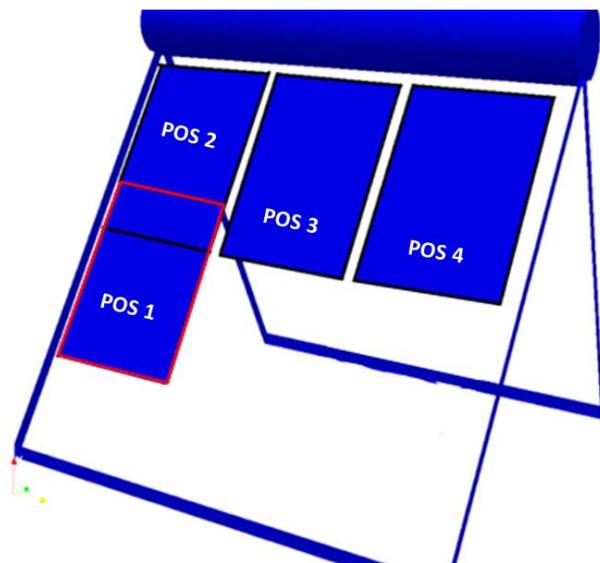


Fig. 4. Positions of MCCA on condenser surface

Experimental data were used for the validation of computational mesh as well as numerical schemes and model of turbulence. Sensitivity of simulation on mesh was tested in the first place. Meshes with 4, 6, and 13 millions of cells were tested. Results of this first testing can be found in Figure 5.

In this figure the comparison of experiment with the simulation for air flow through position 1 and 3 can be found. The biggest difference between CFD and experiment is for *Mesh 1*. The difference between *Mesh 2* and *Mesh 3* is only 4 % on average. Therefore, structure of *Mesh 2* was chosen for CFD study.

The results of simulation depend to a great degree on chosen turbulent model, that is why 3 models were tested – *standard k-ε*, *realizable k-ε* and *k-ω SST*. Deviation of simulation with *standard k-ε* model is about 20% on average. For model *realizable k-ε* and *k-ω SST* is deviation about 10% on average. The simulation with

model *k-ω SST* ends very often with divergence. Model *realizable k-ε* was very stable during testing. For reasons given above the turbulence model *realizable k-ε* was chosen for CFD study [4].

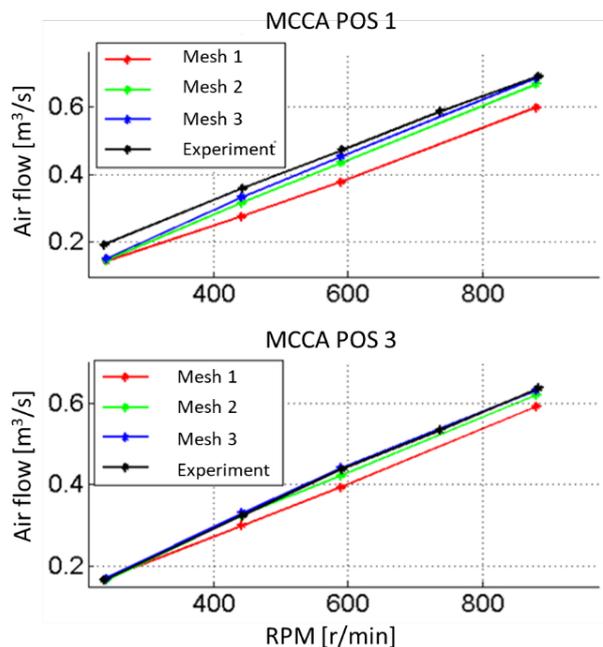


Fig. 5. CFD simulation sensitivity on mesh

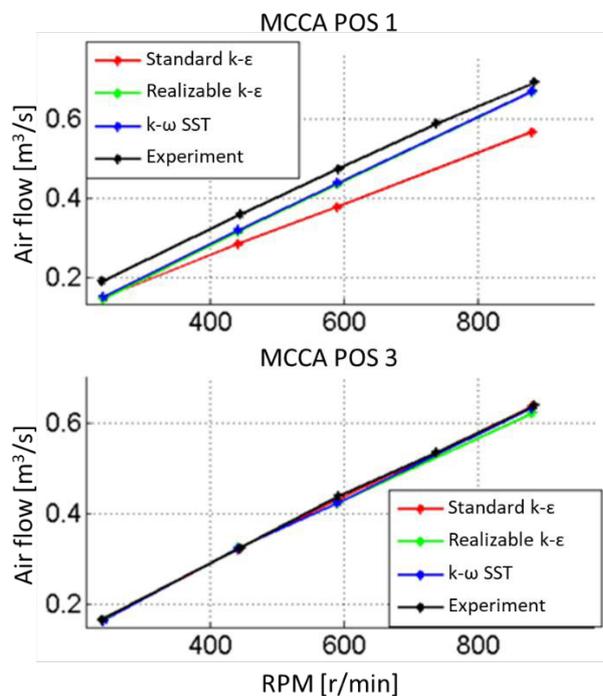


Fig. 6. Simulation results for different models of turbulence

3 CFD study

Geometry and surrounding buildings for study was inspired by real ACC system located in Slovakia.

Dimensions of two ACC cells in CFD model are equal to real ACC. In CFD model buildings with similar positions and dimensions were also placed as in surrounding of real ACC system. It is fact, that the building placed directly under ACC certainly influences an air flow inlet towards a condenser. The same can be said about the building on side of ACC that blocks an air inlet to ACC.

In figure 8 there is a scheme of model of ACC system. In red building under ACC there is a steam turbine, a generator, a control room etc. A warehouse (green building) is also the building that can influence significantly the air flow through ACC. Last building in model is a 48 [m] high chimney.



Fig. 7. Picture of real ACC system

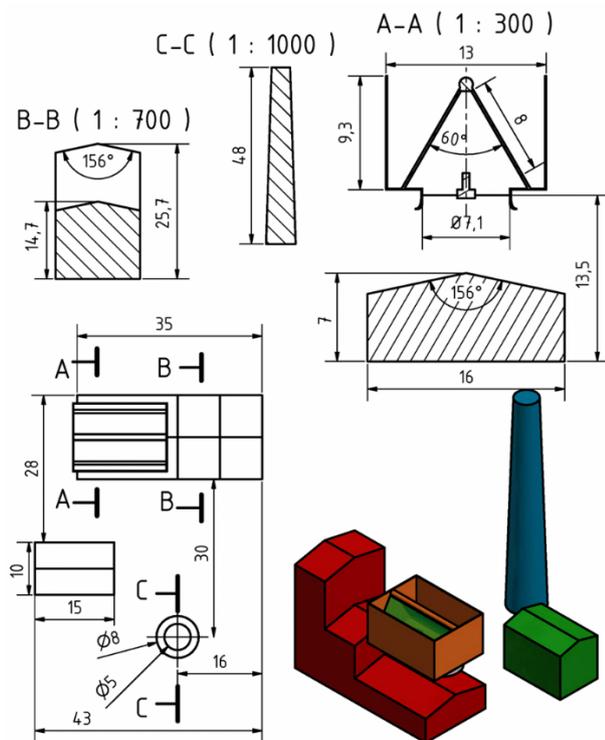


Fig. 8. Scheme of ACC model with dimensions in meters

3.1 Computational domain

Dimensions of computational domain is 600 x 600 [m] and the height is 180 [m]. For every simulation one of domain patch is the inlet and the other one is the outlet. Boundary conditions on ground are set as slip for course mesh region and as noslip for fine and finer mesh region. Boundary conditions for other walls of domain are set to slip condition.

For every calculation mesh has around 21 M cells. The scheme of domain with one of 4 inlets position can be found in Figure 9.

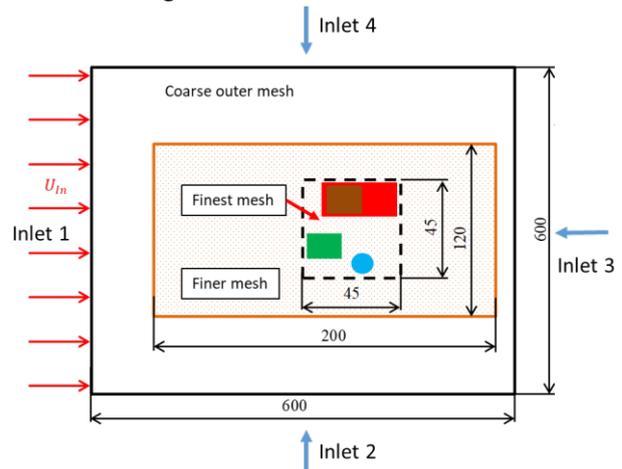


Fig. 9. Computational domain

3.2 Definition of simulation

Simulation is designed as a steady and incompressible flow without heat transfer. This is a reasonable simplification because of low velocities and quite low temperature of the stem in duct – heat exchange doesn't change velocity field much.

Simulation was provided for 3 velocities of wind – 0, 2 and 4 [m/s] and for 4 inlets. Mesh was especially adjusted for every inlet. In Figure 9 mesh adjust for inlet 1 can be found. Simulation was also made with and without buildings for every wind velocity and inlet.

4 Results

There is almost no difference between volume air flow through ACC from simulation with and without buildings for wind velocity 0 [m/s]. The building under ACC was probably considered in the design. Therefore, suction height is sufficient even with building under it and higher suction height cannot increase air flow through a condenser. The difference between these two main settings is found for wind velocity higher than 0 [m/s]. In Figure 10 there are graphs with the change of air volume flow through a condenser with the change of wind velocity. Base volume flow is the volume flow for wind velocity 0 [m/s]. For both settings - with/without buildings the base flow is almost the same. For wind velocity 2 [m/s] volume flow is higher for both cases. Highest increasing of volume flow is caused by wind velocity 2 [m/s].

Simulation without building does not depend on wind inlet direction which was expected and the increase of

volume flow is about 24 [%]. In case with buildings there is the highest increase of volume flow for about 29 [%] for wind inlet 4 which makes sense.

The lowest effectiveness is shown in simulation with the wind inlet 2. That is because of chimney and green warehouse in Figure 8. Inlet air must flow around or above these buildings and the wake forms behind these buildings. Wake causes the decrease of pressure in inlet of ACC. The whole system has then bigger pressure drop.

Wind velocity 4 [m/s] causes the decrease of volume flow in the case without building. Volume flow is about 5 [%] lower than for conditions without wind.

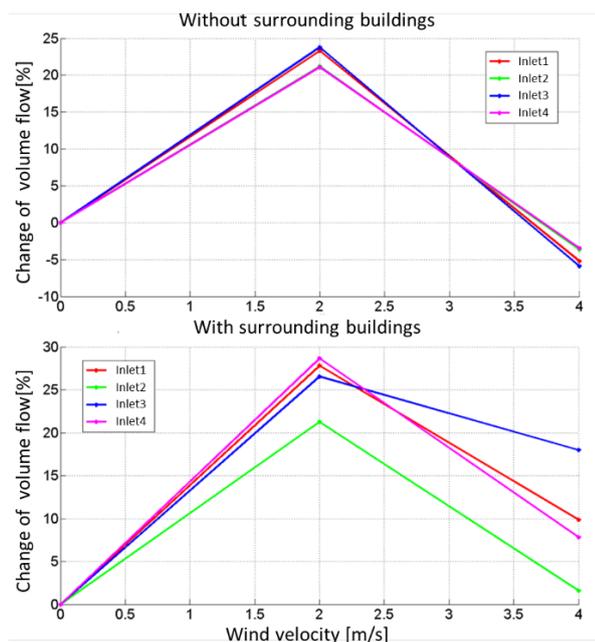


Fig. 10. Change of air volume flow with wind velocity

Reason for decreasing the volume is the decrease of a pressure at ACC inlet.

On other hand, volume flow in case with buildings is still bigger than for conditions with zero wind velocity. Buildings have positive impact on effectiveness of ACC. Significant effect is seen for wind inlet 3 – between inlet and ACC there is a red building as can be seen in Figures 9 and 8. The wake behind this building decreases the pressure at ACC outlet and at the same time ACC inlet is shielded so that decrease of inlet pressure is not so huge as without shielding.

Influence of non-uniform velocity field on condenser has been investigated which is not considered in standard design of ACC system. Therefore, with known equation for calculation of heat transfer coefficient heat duty of ACC was calculated for uniform velocity field and for the same volume flow but non-uniform velocity field known from CFD simulation. Fraction of these two heat duties is plot in graphs in Figure 11 for different wind velocities and wind inlets. Sensitivity of this fraction on wind velocity, wind inlet or surrounding buildings is small. On average heat duty for non-uniform velocity field is about 5 [%] smaller which is for sure not negligible.

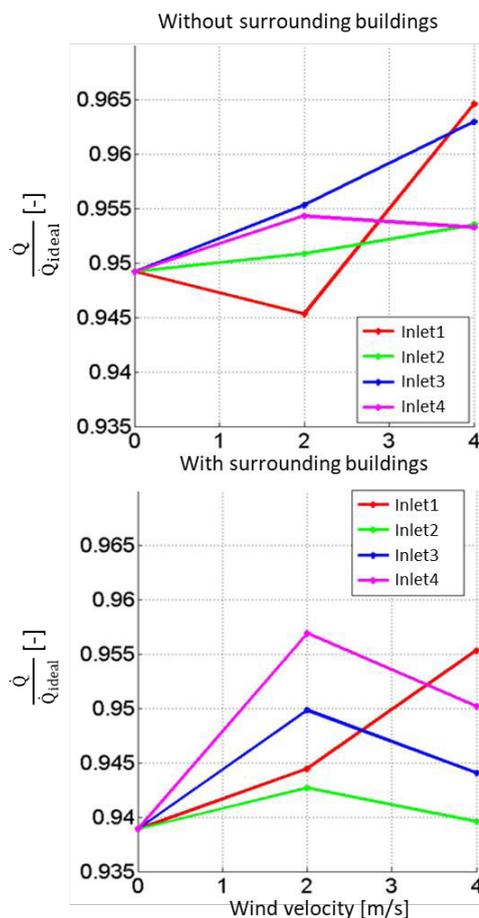


Fig. 11. Heat duty for uniform and real velocity field

5 Conclusion

In this study the impact of boundary conditions on effectiveness of ACC has been investigated. The results show significant influence of wind and surrounding buildings on the effectiveness of ACC. Wind can increase volume of air flow through condenser about 28 [%] or decrease volume of air flow about 5 [%]. Whether wind has negative or positive impact, it depends on wind velocity and on surrounding buildings. Next, the influence of non-uniformity of velocity field on surface condenser cannot be negligible. It causes about 5 [%] decrease of heat duty.

This study shows benefit of CFD simulation in design of ACC system. It would take into account an influence of surrounding conditions in design of ACC. Cooling systems based on ACC could be then smaller or their effectiveness could be higher and with that efficiency of steam turbine would be also higher.

In next studies a real ACC system with all important details included terrain and exact wind characteristics could be simulated. Results of this simulation should be compared with the design data of ACC and with measurement of air volume flow and heat duty direct on real ACC.

References

1. Kroeger, Detlev G. Air-cooled heat exchangers and cooling towers. Tulsa, Okl.: Penwell, c2004. ISBN 08-781-4896-5
2. Greenshields, Christopher J. OpenFOAM - Programmer's Guide. London, UK, 2017. OpenFOAM Foundation Ltd.
3. Greenshields, Christopher J. OpenFOAM - User Guide. London, UK, 2017.
4. Ferziger, Joel H. a M. Peric. Computational methods for fluid dynamics. 3rd, rev. ed. New York: Springer, c2002. ISBN 978-3-540-42074-3.