Mathematical modeling of airflow velocity and temperature fields for experimental test houses

Jānis Ratnieks, B.Sc.[1]
Andris Jakovičs, Associate professor[1]
Staņislavs Gendelis Ph.D.[1]

[1] University of Latvia, Laboratory for Mathematical Modelling of Technological and Environmental Processes, Latvia

KEYWORDS: *CFD, energy efficiency*

SUMMARY:
Five equal size test houses of different constructional solutions have been built in Riga for building energy performance monitoring. Monitoring is done both during the heating and cooling seasons and necessary temperature is provided by an air-air type heat pump. An important issue regarding the heat pump is the coefficient of performance that changes for different outside temperatures. Therefore, even knowing the total power used by the heat pump, it is impossible to find out how much heat is taken into the room exactly. In this paper previously tested mathematical model is used to calculate coefficient of performance of the heat pump by exploiting the physical measurements of the inflow speed, temperature and air exchange in the room. A transient simulation is done results are compared to those of a stationary simulation and experiment. The integral model is also used to calculate coefficient of performance and compare it with numerical simulation.

1. Introduction

Energy efficiency of buildings and building materials is highly topical subject nowadays. To compare different building materials and constructions energy performance, five test houses with different constructional solutions have been built in Riga. The heating and cooling is done by air – air type heat pump (HP) that has velocity of 2 $m \cdot s^{-1}$. To try out different constructional options, a mathematical model is necessary as it is impossible to experimentally build a test house for every possible option. This experiment gives excellent verification options for the mathematical model.

The problem itself is transient both because the HP's cycle is time-dependent and because of fluctuating outside temperature. Period of time has been found form experimental data where temperature is constant and thermal radiation is negligibly small. These problems are studied in chapter 2 – experimental setup. Both - transient and stationary simulations have been made. The assumptions made, turbulence model used, meshing and other considerations regarding experimental implication in mathematical model are discussed in chapter 3 –modelling approach. The results acquired with mathematical model are compared with experimental data in chapter 4 – numerical results and discussion. Precision of numerical results are also discussed there.

2. Experimental setup

2.1 Test houses

As the detailed experimental setup is given in previous publication, the reader is referred to see the work of (Dimdina 2013) as only basic ideas are given here. The test houses (fig.1) are built of equal inner dimensions and cover the main locally produced building materials such as wood logs, ceramic bricks, aerated concrete and rock wool. The inner dimensions are 3x3x3 m that gives the total volume of 27 m^3. The calculated U-value is 0.16 $W \cdot m^{-2} \cdot K^{-1}$ for each construction therefore the expected heat

consumption is equal for every building type. The façade of all the houses are made ventilated to ensure equal conditions for every house. The data are gathered every minute for both the meteo station and sensors inside the building (fig.1).

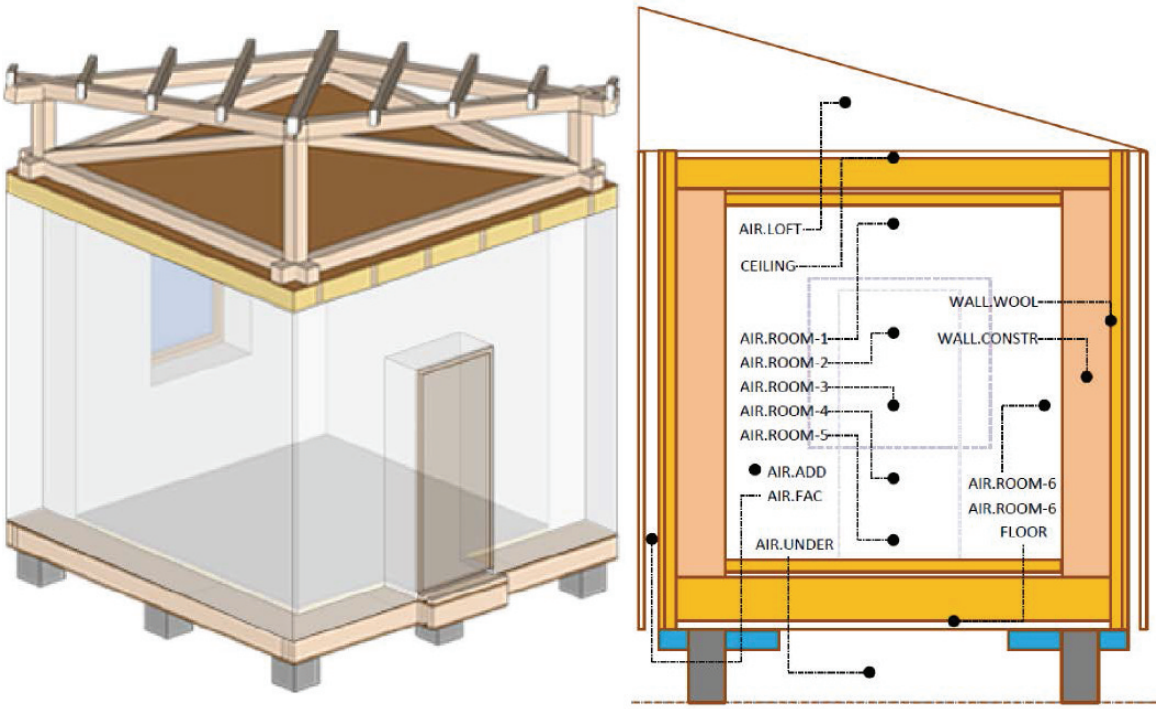

FIG 1. A) Test house geometry and B) monitoring sensor positions.

2.2 Heating cycles

Heating and cooling in test hoses are done with air – air type HPs that are located above doors (fig. 1) and ventilation opening is above the window. The air exchange (Gendelis 2013) in the test buildings have been measured to be n = 0.45 h^{-1}. This means that part of the air that is taken into the room by HP is forwarded to ventilation and some are taken back by the HP. The air – air type heat pump make a rectangular cycle every 51 second by changing the inflow angles form -45° to 45° on the horizontal plane and from -30° to -70°on vertical plane with respect to line that is made by cross section of both planes according to (Daikin Ltd. 2012). This makes the problem time dependent, as the transient simulations are computer resource demanding, it was decided to see if the stationary solution is a good approximation. For the latter the inflow angle was set constant for horizontal plane – 50° and for vertical plane 0°.

As the mathematical model is developed also for stationary case it is necessary to find a period of time when temperature is constant or at least changes a little around some fixed value. Such situation does not often appear in nature however, a period of time was selected from monitored data for outside temperature 5.1 °C that lasted for 2 h 52 min. The corresponding inside temperature should go asymptotically to a stationary temperature if the heating is done continuously with equal power. In experiment however temperature experiences a cyclic behaviour (fig. 2). This means that the HP is making some heating cycles with higher power then the rest. From temperature values it can be seen that last two cycles are approximately the same. Unfortunately no minute by minute power data are present during this particular cycle, only the heat consumption that cannot resolve the time when power was increased.

For the simplicity of the model and the lack of specific experimental data regarding inflow temperature and velocity the average data from experiment were taken into account for model verification. As it is not possible to find out how much mass is taken into the room, additional airflow velocity and temperature measurements were carried out.

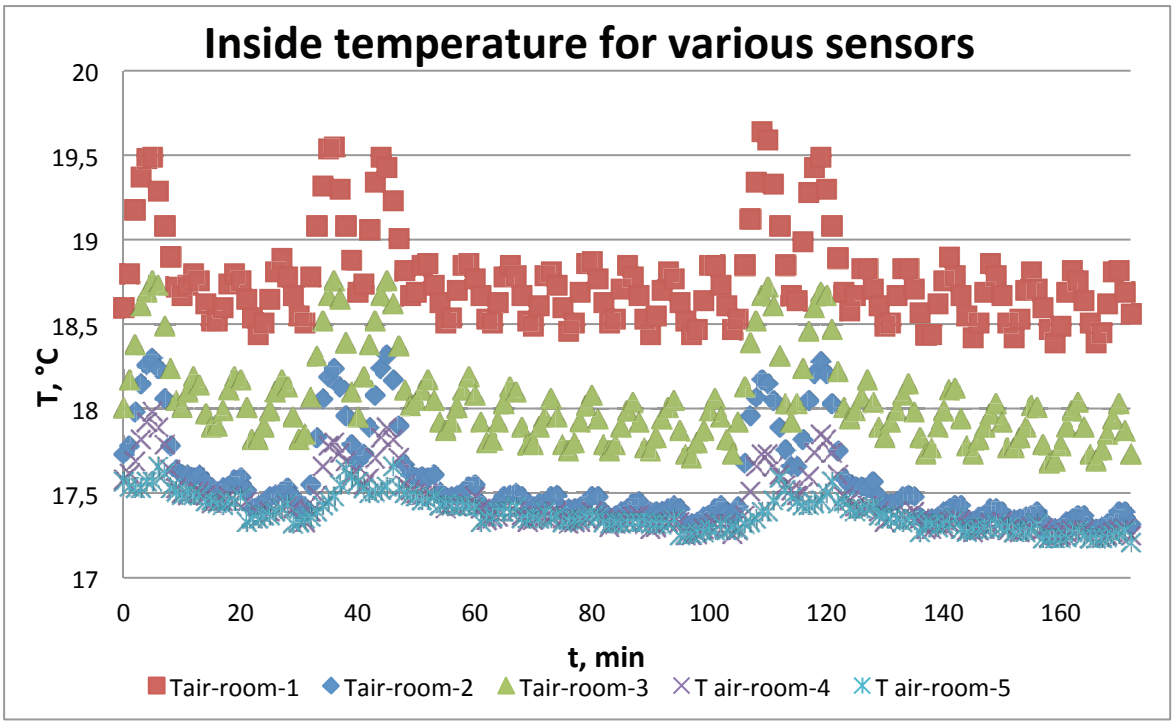

FIG 2. Inside temperature at monitoring points during the period of stationary temperature outside.

3. Modelling approach

3.1 Mathematical model

3.1.1 Governing equations and boundary conditions

For numerical realization the ANSYS/CFX program packet were used. The fluid flow is governed by Navier – Stokes (NS) equations that cannot be solved for this problem so the Reynolds averaged NS equations were used. These equations are derived multiple times in various standard texts on fluid dynamics like (Batchelor 1967) and (Versteeg 1995). The equation implementation in Ansys/CFX environment is described in user guide (Ansys Inc. 2011). Wall functions were used on solid – fluid interfaces that correspond to no slip conditions. Air inlet was defined as a constant mass flux and temperature with time-dependent or stationary direction depending on case. For air feedback to HP a constant mass flux boundary conditions were used because the free total flux would produce a negative outflow at some part of the boundary. For ventilation boundary there was an “opening” type boundary conditions where the relative pressure and outside temperature are defined. This was done to ensure that mass is conserved.

3.1.2 Model assumptions and numerical realisation

The thermal radiation was not taken into account because the radiation from wall to wall is negligibly small and there were negligibly small amount of thermal radiation in the night from the windows. As the air velocities are high – 2 $m \cdot s^{-1}$ the flow is turbulent and therefore a turbulence model was needed. The k-omega shear stress transport model was chosen, because it is robust and performs well both in the volume and near wall region. For buoyancy the *Boussinesq* approximation was used.

The initial conditions were taken as average temperature in test house that was T=17.84°C and the final result from stationary simulation were given as an initial condition for transient simulation. The transient simulation was run for 6 full cycles with time step of 0.05 s. With the given mesh the time required for simulation was 8 days on 3.2 GHz processor with 7 cores.

3.1.3 Geometry and meshing

As the test houses are built with from many layers that are slim or inhomogeneous, the model was simplified and the effective values of heat transfer coefficient, density and heat capacity were used. The meshes were made different for stationary and transient simulations to reduce computational time (Ozolinsh et.al. 2013). For stationary simulations results for coarser and finer mesh were compared.

3.2 Balance calculation and HP efficiency

The numerical simulation is only an approximation and convergence usually doesn't ensure that results are physically consistent. Heat balance for the model must show that heat gains are the same as heat loses. The heat gains by inflow and loses by both – the ventilation and feedback are calculated (1) as integral over surface (Ozolinsh et.al. 2013).

$$P_j = c_p \cdot \rho \oiint T \cdot \vec{v} \cdot d\vec{S} \qquad (1)$$

Where P_j – power by mass flux through boundary (W)
c_p – heat capacity at constant pressure (J/(kg·K))
ρ – density (kg/m^3)
T – temperature (K)
$\vec{v}$ – airflow velocity (m/s)

Heat loses through walls are calculated (2) as heat flux through outer surfaces.

$$P_\Phi = \oiint \Phi \cdot d\vec{S} \qquad (2)$$

Where P_Φ – power due to heat flux through solid material surface (W)
$\vec{\Phi}$ – heat flux (W/m^2)

For the various inflow temperatures the average temperature from the experimental points can be determined and compared to the experimental average. By doing two stationary calculations with the different temperatures interpolation can be done to find the inflow temperature that makes model average equal to experiment average. From these calculations the HP coefficient of performance can be determined by mathematical modelling. This result can be compared to integral model by taking into account experimental values.

4. Numerical results and discussion

To compare numerical experiments mutually a line perpendicular to the floor in the middle of room is considered. This is done because there are experimentally measured temperatures (fig. 1b) along it. The temperatures for stationary case are taken directly from final solution, but for transient case averaged over last HP cycle. For the verification that after six HP cycles the flow is stationary, the results were compared between the last two cycles and the difference was negligibly small.

The approximation that in the given time, when outside temperature is almost constant, heat flux through wall is quasi stationary will be better for constructions with lower heat capacity. Therefore for calculation the lightweight construction (made of plywood and rock wool) were taken and experimental data from this particular house was used for verification.

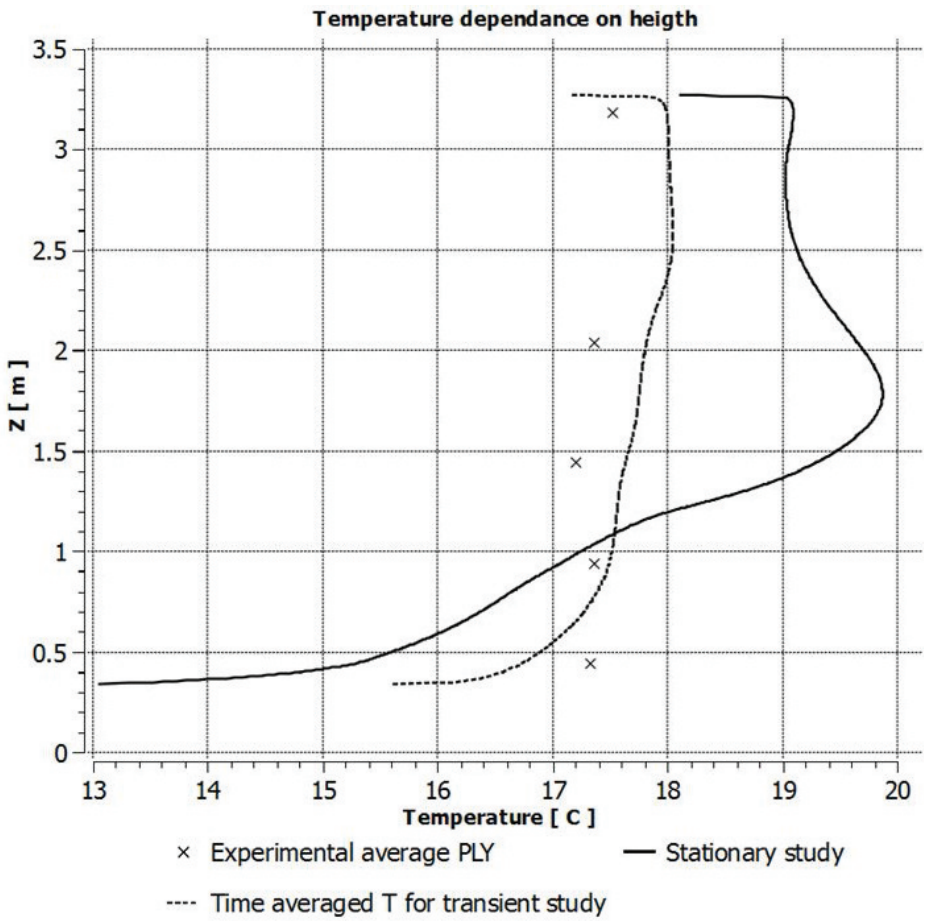

FIG 3. Temperatures at the middle line for mathematical models and experimental data.

The heat balance were computed (eqs. 1, 2) and the results (table 1) show that the heat balance is off by approximately 3%. This means that although the convergence was set to be 10^{-4} for maximum residual, the total error is considerably higher. This is due to third type boundary conditions. The temperature is not fixed at any point and therefore an error can occur.

TABLE 1. Heat balance for numerical simulation.

Heat gain	
Inflow	697.0
Heat loses due to ventilation	
Ventilation	-55.6
Feedback	-537.0
Heat loses due conduction	
Window	-15.0
Wall	-71.1
Floor	-15.8
Ceiling	-25.0
Doors	-13.1
Total loses	**-732.5**
Error, %	5

Temperature and velocity field plots (figs 4 and 5) for stationary simulation show that results are physically consistent. The overall velocity profiles are as expected and at the near wall region the flow are downward as expected for cold wall.

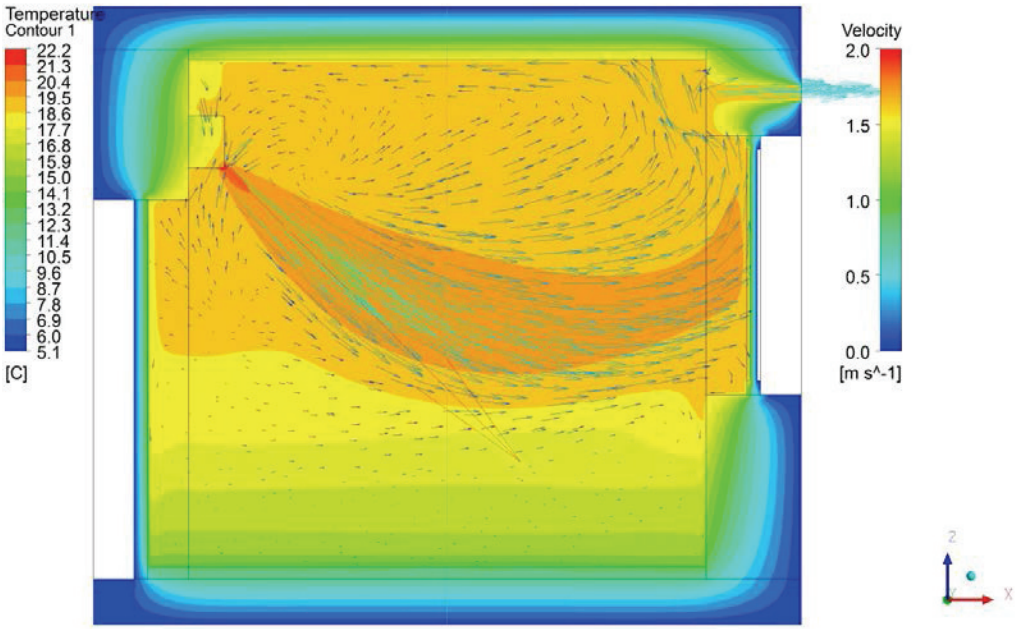

FIG 4. Temperature and velocity field for stationary study.

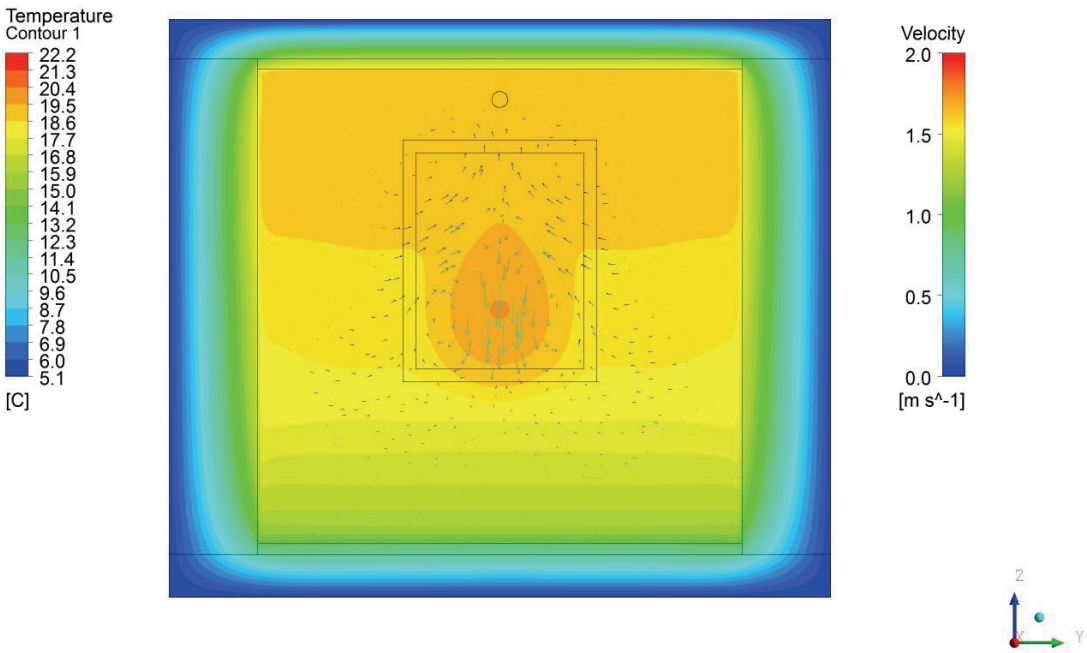

FIG 5. Temperature and velocity field for stationary study.

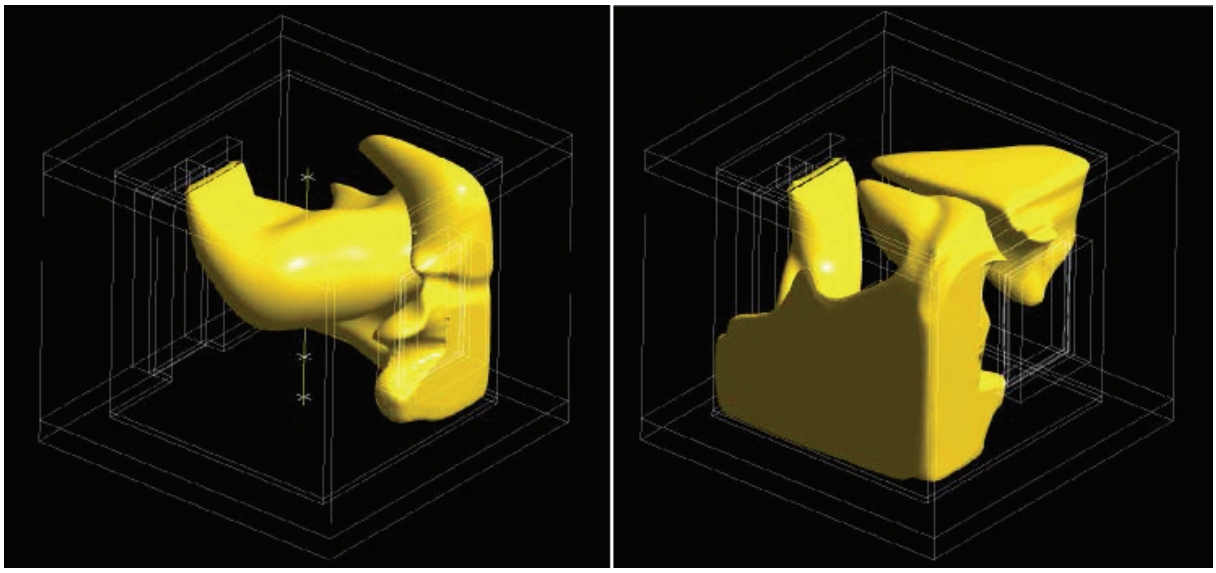

FIG 6. 18 °C isosurfaces at various time for transient study.

5. Conclusions

Mathematical model for airflow in the test houses have been set up and two types of calculations done. The numerical results seem to be qualitatively correct as the airflow directions at near wall regions tend to be physically consistent. Also the temperature gradients in the solid are as expected.

As it can be seen, stationary and transient model give similar volume average temperatures for the room, but the experimental point average is higher than model predictions. This is due to inflow angle that is directed toward the experimental point location. The values for each individual experimental point vary significantly in stationary model for points closer to ceiling and floor. This is due to bad mixing that stationary model offer. The transient model however gives much better results as the changing inflow ensures better mixing. Therefore we can conclude that for precise temperature fields in the room the stationary model is insufficient and transient model should be used. If, for example, only the average temperature of the room is necessary, the stationary model could be sufficient, but more tests for different temperatures must be made to verify this claim.

An important drawback for this model is that the power data weren't available at the time these measurements were carried out. The data is available now and therefore time averaging will be avoided in future studies and the full heating cycle will be included.

The first experimental results show that there are houses that perform in similar manner and therefore it has been decided to test the COP value for heat pump by letting the heating be done by inefficient heater that have COP value of unity.

6. Acknowledgements

This research was done at the University of Latvia with the financial support of European Regional Development fund support. Project nr. 2011/0003/2DP/2.1.1.1.0/10/APIA/VIAA/041

References

Ansys Inc. Ansys CFX user guide

Batchelor G.K. 1967 An introduction to fluid dynamics, Cambridge university press, ISBN 0 521 66396 2

Daikin Industries Ltd. 2012 Daikin room air conditioner user manual

Dimdina I. Jakovics A., Gendelis S., Klavins J. 2013. A Testing Ground for Measuring Influence of Building Envelope Materials on Energy Efficiency and Indoor Environment, Proceedings of the 11th REHVA World Congress & 8th International Conference on IAQVEC – CLIMA 2013

Gendelis S., Jakovics A., Nitijevskis A., Ratnieks J. 2013 Comparison of Different Air Tightness and Air Exchange Rate Measurements in Very Small Test Buildings, 34th AIVC conference: "Energy conservation technologies for mitigation and adaptation in the built environment: the role of ventilation strategies and smart materials". Athens, Greece, 2013

Ozolinsh A., Jakovics A., Ratnieks J., Gendelis S. 2013 Numerical Modelling of Humidity and Temperature Conditions in Buildings with Different Boundary Structures, Proceedings of the 11th REHVA World Congress & 8th International Conference on IAQVEC – CLIMA 2013

Ratnieks J. Jakovics A., Gendelis S. 2013. Stationary Heat Transfer and Airflow Simulation for Test Polygon Houses, CYSENI 2013, 10th international conference of young scientists on energy issues, Kaunas, Lithuania May 29-31, 2013 p. VI – 382-389

Versteeg H.K. Malalasekera W. 1995 An Introduction to Computational Fluid Dynamics. The Finite Volume Method, Longman group Ltd., ISBN 0 582 21884 5