SAT-9.2-2-HT-07

CFD simulation of the influence of air conditioner on the turbulence intensity in residential room

Ivan Denev, Petko Tsankov, Ivan Antonov

***Abstract**: It is made a simulation of the flow in air condition room. The aim of the investigation is a the discovery of emerging circulation (stagnant) areas and levels of turbulence in the room*

***Key words:** CFD, turbulence intensity, air conditioning*

INTRODUCTION

With the development of systems and automatic adjustment in the processes of heating, ventilation and air condition are imposed more demanding for maintaining a stable, comfortable environmental conditions, both at work and at home. Huge progress has been made to manufacturers of HVAC systems in terms of maintaining a constant temperature in living rooms, it is working hard on regulating of velocities and humidity at air conditioning. Factor when choosing a system for HVAC is also the noise level during operation of the system. It is working hard on all these parameters to improve occupant comfort

It is poorly studied the impact of the turbulence intensity in the air-conditioned room on people. Turbulence that occurs in air conditioning and the presence of "dead" zones may have a negative impact on residents stay.

SUMMARY

The purpose of this work is to implement different variants of modelling heating residential premises and comparing the values of the turbulence intensity in each of them.

The turbulence intensity according to [1] is expressed as the ratio of the mean-squared velocity pulsation and the average velocity:

$$T.I. = \frac{\sqrt{\frac{1}{3}(\overline{u}'^2+\overline{v}'^2+\overline{w}'^2)}}{\overline{V}} = \frac{\sqrt{\frac{2}{3}k}}{\overline{V}} \qquad (1)$$

If all three directions are with same pulsations u'=v'=w' then we have isotropic turbulence and for turbulence intensity can be written:

$$T.I. = \frac{\sqrt{\overline{u}'^2}}{\overline{V}} \qquad 2)$$

In current work is made a visualization of turbulent intensity and arise of circulation (stagnant) areas with the help of software for numerical modeling Ansys Fluent.

For this purpose are developed three options for heating the room:

- Clean convective heating with good sealing (free of infiltrated air);
- Convective heating in the presence of infiltration;
- Heating by using the air conditioner.

It is considering a household room which is schematically shown on Figure 1.

In the first version of the review is set convective heating with panel radiator, it is accepted that the room is well sealed, ie it is not available infiltration / exfiltration. To solve the problem is using a standard k - ε turbulence model. This is a semi-empirical model based on transport equations for the turbulent kinetic energy (k) and dissipation rate (ε) [2]. They are defined with the following transport equations:

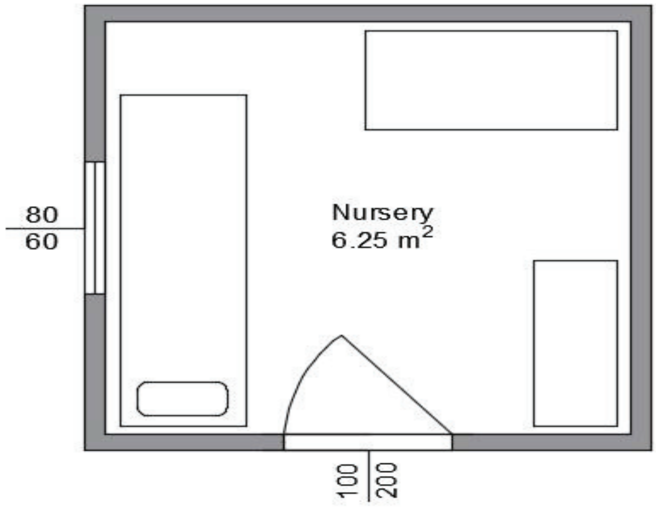

Fig. 1. Sketch of the heating room

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+ \\ +G_k+G_b-\rho\varepsilon-Y_M+S_k \qquad (3);$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+ \\ +G_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-G_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon, \qquad (4)$$

where:

G_k – generation of turbulent kinetic energy due to the presence of a velocity gradient.

G_b - generation of turbulent kinetic energy due to lift force.

Y_M –It shows the degree of increase in the turbulent kinetic energy to compensate for attenuation in the rate of dissipation. This ratio reflects the effect of the compressibility of the fluid at high Mach numbers

To solve the problem is set and calculate the energy equation which the Fluent software product has the form:

$$\frac{\partial}{\partial t}(\rho E)+\nabla\big(\vec{v}(\rho E+p)\big)=\nabla\big(k_{eff}\nabla T-\textstyle\sum_j h_j\vec{J}_j+(\bar{\bar{\tau}}_{eff.}\vec{v})\big)+S_h \qquad (5)$$

where:

k_{eff} – effective thermal conductivity (k + k_t, where k_t is turbulent thermal conductivity);

$\vec{J}_j$ – diffusion flow for j-th component;

S_h – includes heat energy obtained when a chemical reaction or some other form of energy released from the source volume.

For all types of flows FLUENT solves the equation of continuity and impuls movement. For flows in which there are processes of heat transfer or they are compressible it is necessary to solve the equation for energy conservation.

On fig 2 and fig. 3 are shown the results for the maximum velocity and the section with a high turbulence intensity.

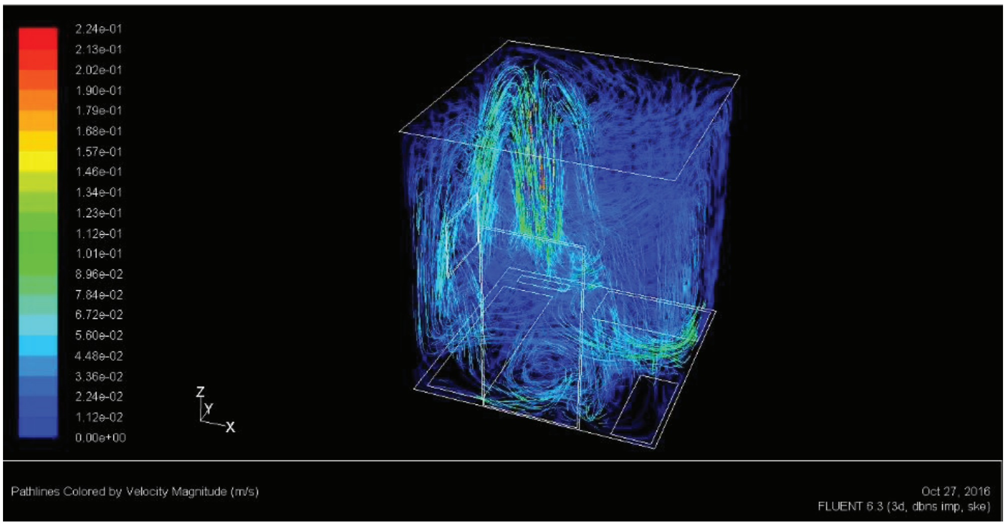

Fig. 2. Streamlines of velocity field at pure convection

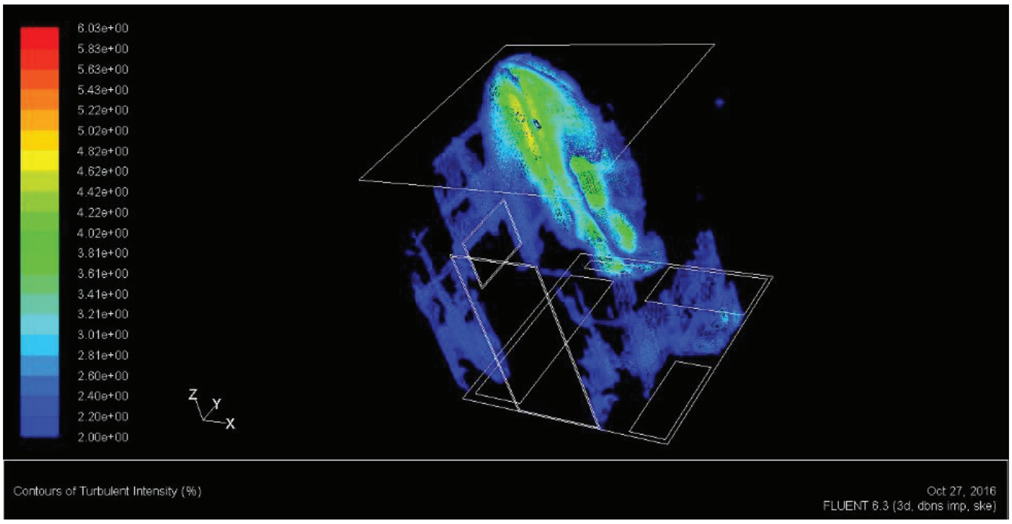

Fig. 3. Local increasing of turbulence intensity in pure convection

In the second option is considered a room heated by a panel radiator, it is available infiltration / exfiltration in the joints of window and door of the room again is using k - ε turbulence model with include energy equation. Streamlines of the velocity and the local increase of the turbulence intensity are visualized in fig. 4 and fig.5.

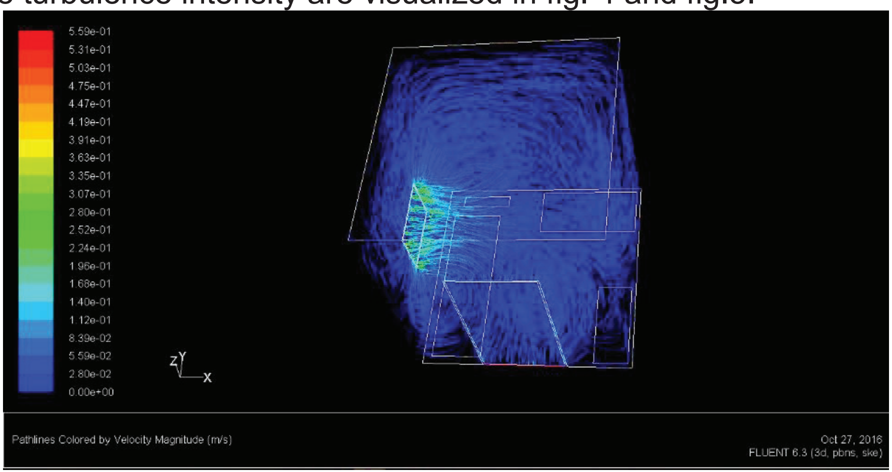

Fig. 4. Streamlines of velocity field at presence of infiltrated air

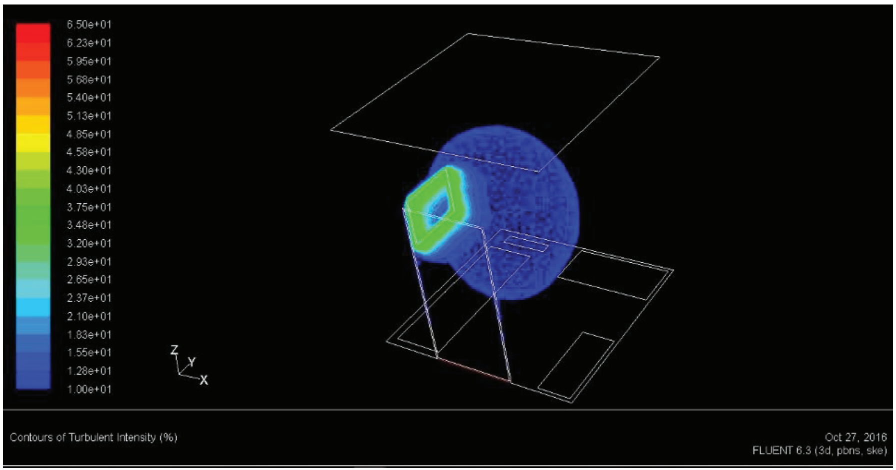

Fig. 5. Locally increasing of turbulence intensity in the presence of infiltrated air

The third option involves heating in room through an air conditioner again is using using k - ε turbulence model with include energy equation. The results of numerical simulation are shown in fig. 6 and fig. 7.

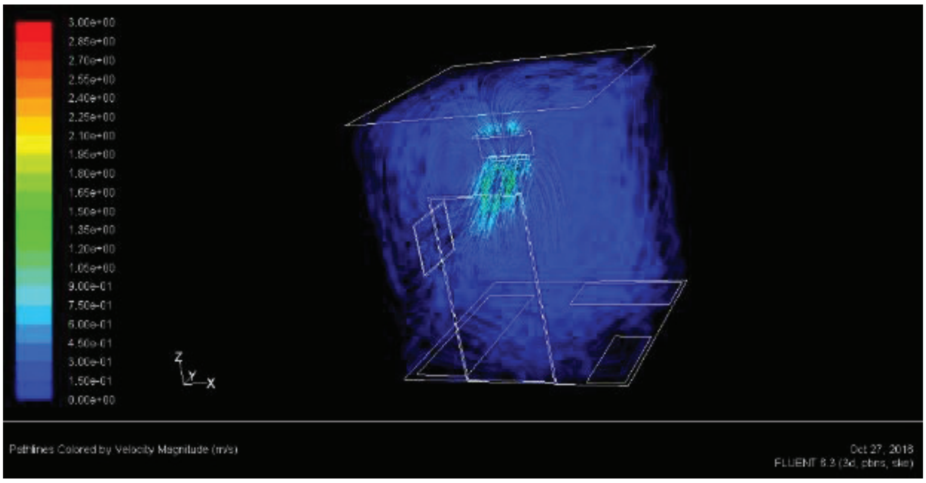

Fig. 6.Streamlines of the velocity field in heating air conditioner

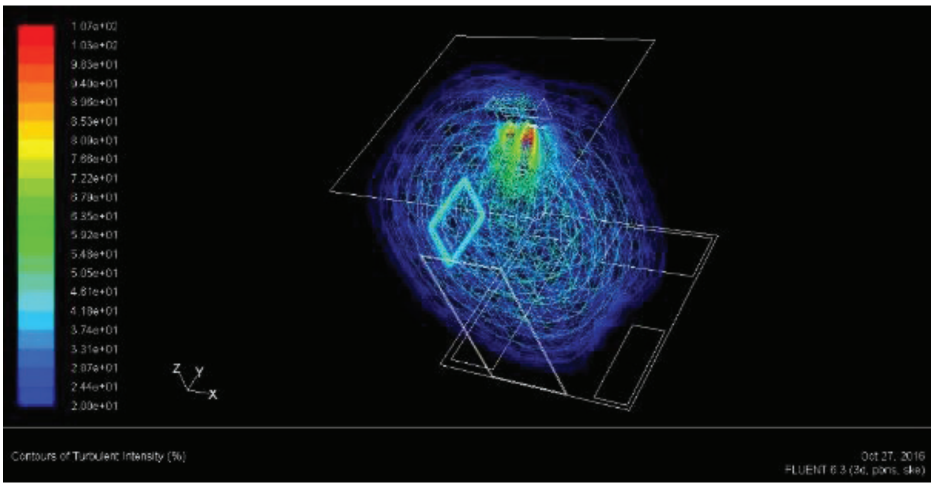

Fig.7. – Increasing of the turbulence intensity in heating air conditioner

CONCLUSION

An analysis of three versions which is made for heating the room shows how forced circulation of air-conditioned air increases the turbulence intensity stepper. The availability of pure convection leads to values for turbulence intensity in the range of from 0 to 6%, and only in 10% of the space are monitored values above 2%. Locally over the heated body was observed an increase in the range of from 2 to 6%. In infiltration values increased to 60%, but it is again local and it does not take more than 1/3 of the volume of heated room.

In terms of the increased level of turbulence the worst is the third option – heating with air conditioner. In 90% of the volume are observed values of the turbulence intensity over 20%, in local area under the air conditioner reach 107%.

The analysis which is made may be useful for modelling and design of air conditioning, which prior "calm" the flow of treated air before its supply to the heated room.

Literature

[1] Antonov I., Applied fluid mechanics, S. 2009
[2] Antonov I., Terziev A., Manual tool for applied fluid mechanics, S. 2012
[3] Fluent user guide
[4] Gambit manual

The paper is financed under the contract N162ПД012-02 “Simulation of processes and systems for the evacuation of harmful substances in the environment

Contacts:

PhD student Ivan Denev, Department of Hydroaerodynamics nand hudraulic machines, Technical university of Sofia, e-mail: ivan_denev.eng@abv.bg ;

Senior assist. prof. Petko Tsankov, PhD, Faculty of engineering and pedagogy of Sliven, Technical unibersity of Sofia e-mail: ptsankov@abv.bg

Prof. DSc Ivan Antonov, department of Hydroaerodynamics and Hydraulic machines” Technical university of Sofia, phone:02-965- 33-67,mfantonov@abv.bg

This paper has been reviewed.