ECCOMAS Congress 2016
VII European Congress on Computational Methods in Applied Sciences and Engineering
M. Papadrakakis, V. Papadopoulos, G. Stefanou, V. Plevris (eds.)
Crete Island, Greece, 5–10 June 2016

HEAT RECOVERY BY CROSS FLOW

K. Adámek[1], J. Kolář[1], and P.Peukert[1]

[1] VÚTS Liberec
Svárovská 619, CZ-46001 Liberec
{karel.adamek, jan.kolar, pavel.peukert}@vuts.cz

Keywords: Heat recovery, cross flow, numerical flow simulation.

Abstract. *Important problem of residential buildings, social centers etc. is the increased concentration of pollutants during use of the building / room.*
The air-tight outer housing is typical for new or reconstructed buildings, due to tight windows, outer thermal insulation, etc. It is good for the reduced heat consumption in winter period but there are some disadvantages, in general, too, known in general as the “sick building syndrome”. For instance, the higher air humidity could result in water vapor condensation in the colder outer corners of the observed room and consequently in mould growth, or in assembling rooms (classrooms, etc.) with poor ventilation, the concentration of CO_2 is increasing quickly over the hygienic limits and is the reason of fatigue, headache etc., or due to internal heat sources, the room temperature is increasing and it is necessary to keep it on a suitable level. Many other pollutants can be present in observed buildings, as for instance formaldehyde, radon, bacteria etc., but it is over the extent of this short paper.
In standard rooms without ventilation or air conditioning, the natural or forced ventilation is used, only. Results of numerical simulations describe, how to simply remedy the so-called sick building syndrome and without any expenses to get and keep healthy living environment.
The design of air conditioning units, taking into account all such pollutant sources, is well-known. Many producers offer many different units for various kind of use. In general, the costs are significant – not only investment, but operational, too. In reconstructed buildings is not space enough for such equipment.
In each system of ventilation / air exchange is a part of polluted inner air replaced by fresh outer one. The exhausted inner warm air contains some heat, which must be in winter period added again into the fresh and cold outer air. To reduce the extent of such thermal energy, various heat recovery systems are used.
The paper deals with numerical simulations of flow and heat exchange between warm and cold air, using heat exchanger with crosswise flows. It contains two systems of crosswise oriented channels, in the first one the outlet warm air is cooled by inlet cold air, situated in the second channel. The aim is to get the maximum of recovery heat to reduce costs for reheat of outer air, at minimum pressure losses during air flow.

1 INTRODUCTION

Important problem of residential buildings, social centers etc. is the increased concentration of pollutants during use of the building / room.

The air-tight outer housing is typical for new or reconstructed buildings, due to tight windows, outer thermal insulation, etc. It is good for the reduced heat consumption in winter period but there are some disadvantages, in general, too, known in general as the “sick building syndrome” [1]. For instance, the higher air humidity could result in water vapor condensation in the colder outer corners of the observed room and consequently in mould growth, or in assembling rooms (classrooms, etc.) without ventilation, the concentration of CO_2 is increasing quickly over the hygienic limits and is the reason of fatigue, headache etc., or due to internal heat sources, the room temperature is increasing and it is necessary to keep it on a suitable level [2]. Many other pollutants can be present in observed buildings, as for instance formaldehyde, radon, bacteria etc. [3], [4], but it is over the extent of this short paper.

In standard rooms without ventilation or air conditioning, the natural or forced ventilation is used, only. Results of numerical simulations [5] describe, how to simply remedy the so-called sick building syndrome and without any expenses to get and keep healthy living environment. Similar data about problems of ventilation are presented for instance in [6] to [11].

The design of air conditioning units, taking into account all such pollutant sources, is well-known. Many producers offer many different units for various kind of use. In general, the costs are significant – not only investment, but operational, too. In reconstructed buildings is not space enough for such equipment.

In each system of ventilation / air exchange is a part of polluted inner air replaced by fresh outer one. The exhausted inner warm air contains some heat, which must be in winter period added again into the fresh and cold outer air. To reduce the extent of such thermal energy, various heat recovery systems are used.

The paper deals with numerical simulations of flow and heat exchange between warm and cold air, using heat exchanger with crosswise flows. It contains two systems of crosswise oriented channels, in the first one the outlet warm air is cooled by inlet cold air, situated in the second channel. The aim is to get the maximum of recovery heat to reduce costs for reheat of outer air, at minimum pressure losses during air flow.

2 MODEL OF ONE CHANNEL FLOW

The simple introductory model simulates the influence of various kinds of inner ribs. It is clear that ribs increase the heat transfer surface in general, but in the same time the pressure losses are increasing, too. For comparison of various designs of ribs the simple model of one channel, only, is created, flowed by warm air (as an example). The warm air is cooled by outside cold walls, defined as constant cold temperature. Really, on outside surfaces there is flowing the second cold air, warming along the heat transfer surface, and then its temperature is not constant.

This model is idealization, only, for simple and quick evaluation of various designs.

Some results are presented here for several kinds of triangular ribs of various pitches in the same channel height. The Figure 1 shows the pressure field in inlets cross-sections for three simultaneously solved cases (signed as 7-8-9, where 3-6-9 ribs are used on the same distance).

Figure 1: Pressure field in inlet cross sections (cases signed 7-8-9)

On the Figure 2 to Figure 4 there are presented the fields of pressure, temperature and velocity, all in the middle height of observed channels. Used example is for warm air at the inlet. In narrow channels there is faster temperature change, comparing with large ones. Probably, for specified channel length and temperatures difference in the recuperation, in the narrows channel the maximum possible temperature gradient was reached. It should be to check this case for higher specified temperature difference.

In narrow channels the pressure decreasing in the flow direction is slower – the flow resistance in narrow channels is higher in general, the pressure decreasing is not so fast.

The velocity in narrow channels is smaller in general, due to higher flow resistance.

From realized numerical simulations all necessary flow parameters were evaluated and the relation between thermal outputs received by recovery is determined as

$$Q = m \,.\, cp \,.\, \Delta T \; [W]$$

and mechanical input, necessary for overcoming of flow resistance, is

$$P = V \,.\, \Delta p \; [W].$$

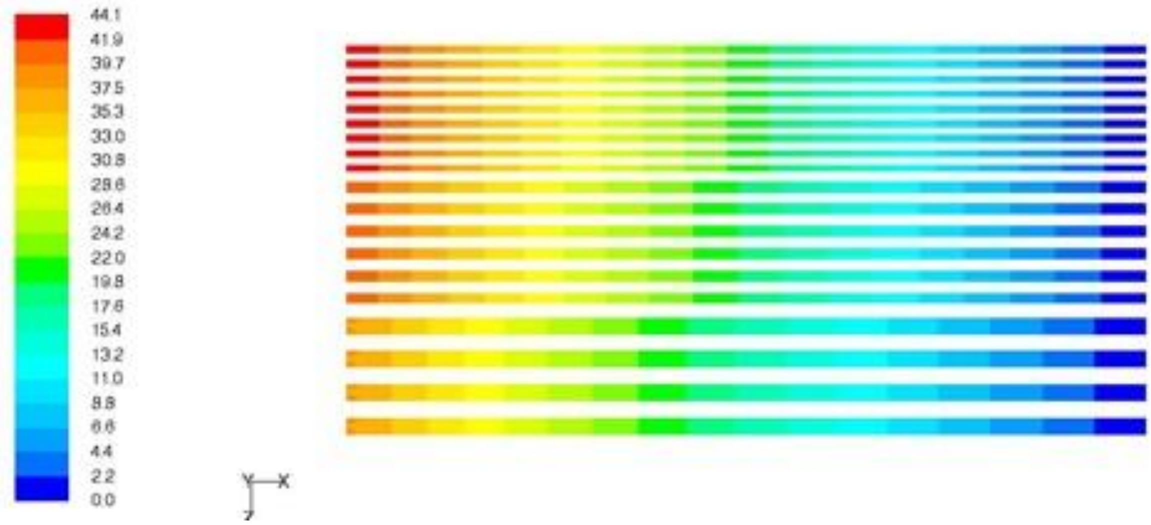

Figure 2: Pressure field

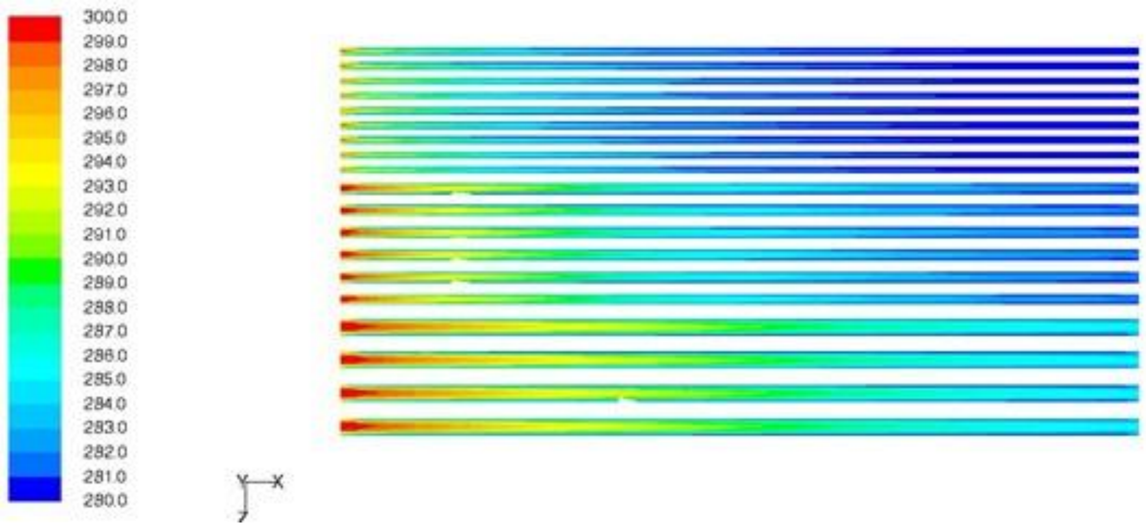

Figure 3: Temperature field

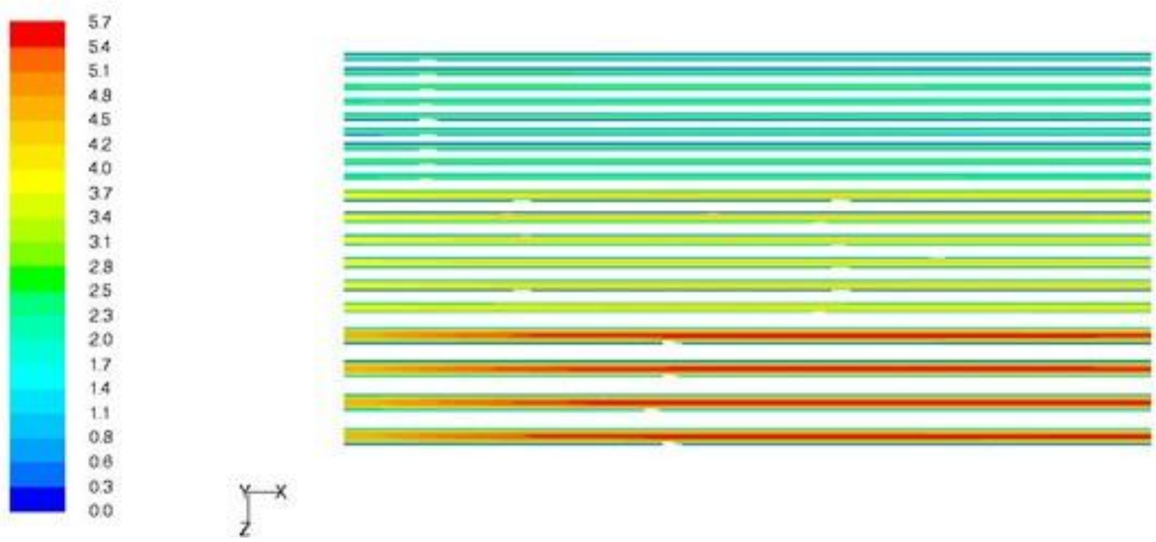

Figure 4: Velocity field

Received values for simulated cases of different ribs arrangements are in the Table 1, as the first information, only, because channel dimensions and their arrangement are not identical with reality.

case	Q/P	
1	**435.6**	**100%**
2	456.3	105%
3	477.6	110%
4	481.9	111%
5	468.6	108%
6	471.1	108%
7	490.7	113%
8	496.4	114%
9	528.5	121%

Table 1: Effectivity of various shapes of ribs – summary

The cases 1-2-3 contain 1-2-3 channels. The effectivity is increasing with channel density (i.e. with decreased channel pitch), cases (1-2-3-7-8-9). The influence of various detailed modifications of the case 3 (cases 4-5-6) on the value of effectivity is practically none. The greater heat transfer surface influences markedly the greater thermal output. The influence of greater flow resistance due to the higher flow velocity in smaller cross-section of channel is not so important.

Probably, with decreased ribs pitch (increased ribs density) the recuperation effectivity is increasing theoretically. Practical restriction is given by production feasibility – which slimness of ribs (their density and height) is possible to produce using the actual technology.

3 MODEL OF TWO CROSS FLOWS

The channel modifications above, tested on single channel, only, are verified on the more complicated model of cross flow of both warm and cold flows. The model contains two flat channels of the crosswise orientation of flows, equipped by ribs, arranged after the previous paragraph. The heat transfer is realized on the common surface between them, while the outer surfaces are defined as thermaly insulated. Really, on outside surfaces there are connected next crosswise channels, the modeled situation is really repeated in each pair of neighboring layers.

Real heat exchanger contains many pairs of such channels of warm and cold air, to get an adequate total air flow. Simulation of such real design is not possible due to large extent – large outer dimensions, many fine details of ribs etc.

Temperatures are specified after [12], warm air (25°C, dew point 5°C) and cold air (5°C, dew point 1°C), i.e. without humidity condensation.

Features of the model:

1).The simulated temperature distribution is the same as in the reality. The heat exchange is realized between two layers of warm and cold air. On the outer surfaces there is defined perfect thermal insulation – really there is next heat transfer into next air layer, but for the evaluation of designed geometry of heat transfer surface here is used such simplification.

2) The simulated flow or velocity distribution is real as in the reality. The complete exchanger consists from many pairs of such channels, in each of them some partial air volume is flowing and the flow resistance is the same in each layer.

From results of simulations we can state that the transferred heat output (m.cp.ΔT) is much higher than the mechanical input consumed by flow resistance (V.Δp). Using more complicated shape or higher number of ribs, the gain of thermal output from heat recovery is probably more important than the increased consumption of mechanical energy due to higher flow resistance in narrow channels among ribs.

As an information here is presented one of possible solution procedure and received results. As mentioned above, due to the task extent the heat transfer between two adjacent layers of warm and cold air is solved, only, the outer surfaces are thermally insulated (really there are next heat transfer surfaces here). This modeled situation is repeated in each pair of adjacent layers.

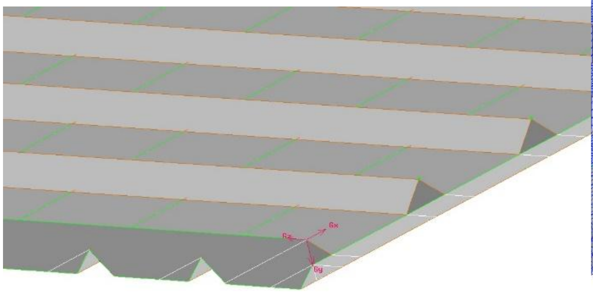

Figure 5: Geometry of 1 layer

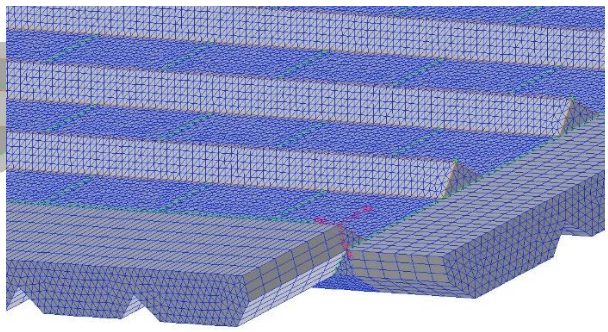

Figure 6: Mesh of 2 layers

Geometry

Created after the Figure 5, the model contains two layers situated across. Along outlines some inlet and outlet channels are added to suppress an influence of possible backflows on the total mass balance. In the model should be removed all details, necessary for production (rounding, foil thickness etc.), because such details extremely complicate the model creation and solution, too. The foil thickness is equal zero in the geometry; the real value is defined later, together with other material specifications.

By laying of individual surfaces in crosswise manner, inside arise complicated shapes of individual channels, with high turbulence of the flow, which increases the heat transfer coefficient of observed case.

Mesh

Created from tetrahedrons of 1.5 mm, see the Figure 6 - together 5.6 millions of elements, over 0.5 GB of data respectively. Regarding fine geometric details and heat exchange, the mesh should be even finer, but restrictions are given by the extent of actual operational memory of standard PC and the time of solution.

Boundary and initial conditions

After [12] the cold air temperature is defined +5°C (278 K) for lower body and warm air temperature +25°C (298 K) for upper body.

Defined pressure gradient of 100 Pa is identical for both bodies. Thus in the model of certain geometry arises flow of certain velocity, here in the specified geometry 1.4 – 1.5 m/s approx. For several specified pressures several operational points are simulated for prescribed velocity range 1 to 4 m/s approx. and so-called operational characteristics V = f(Δp) is evaluated. From the simulation the characteristics of thermal input, mechanical output, thermal efficiency etc. are evaluated, too.

Results

As an illustration, only, there are presented some basic figures of the flow field for one solved case, only, see the Figure 7 to Figure 12.

Values in individual points of observed surface are quite different, so it should be to remind here, that the standard deviation of evaluated data is quite high. Nevertheless, the errors of energy balances for heat recovery and flow resistance, using averaged values, are not important.

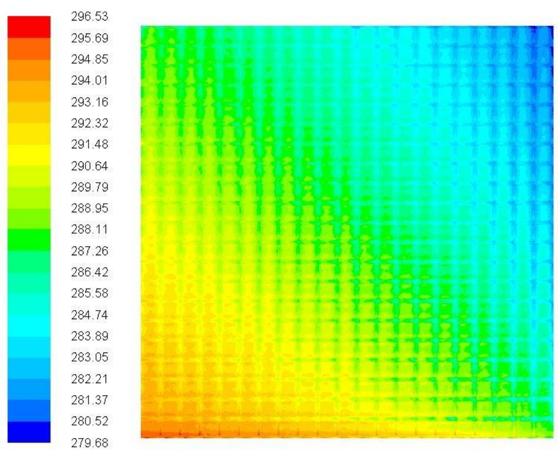

Figure 7: Temperature on the middle (heat transfer) surface – warm air from down, cold air from right

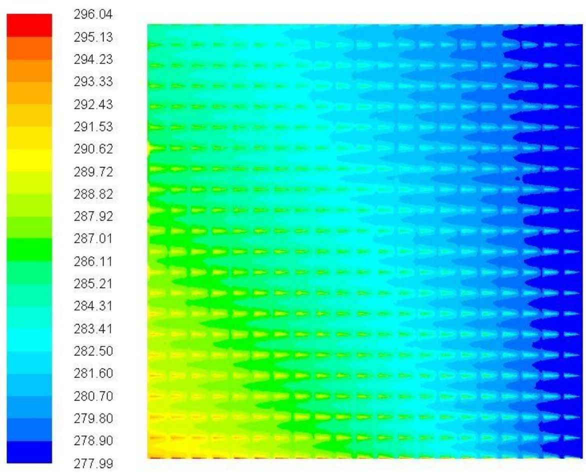

Figure 8: Temperature on the lower (insulated) surface – cold air inlet from right

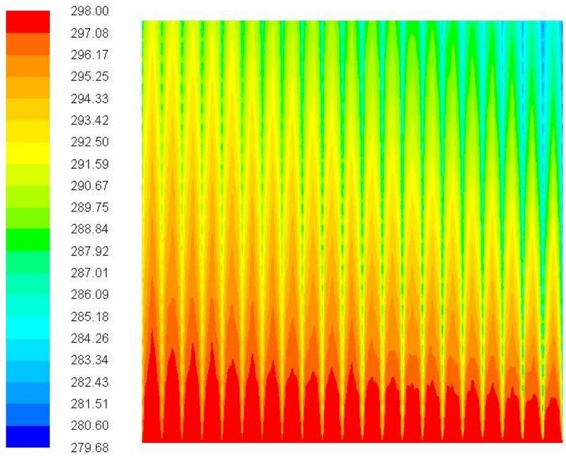

Figure 9: Temperature on upper (insulated) surface – warm air inlet from down

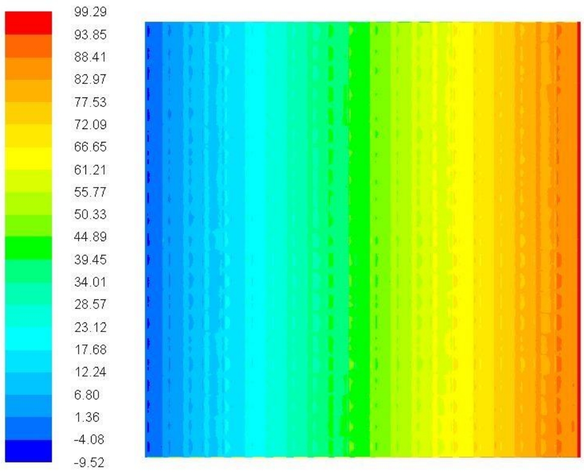

Figure 10: Pressure field in the lower layer (inlet of 100 Pa from right)

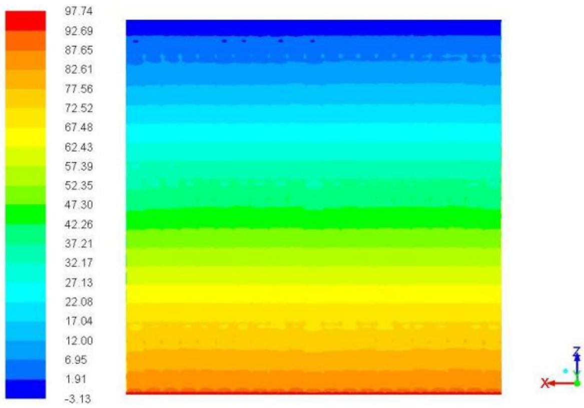

Figure 11: Pressure field in the upper layer (inlet of 100 Pa from down)

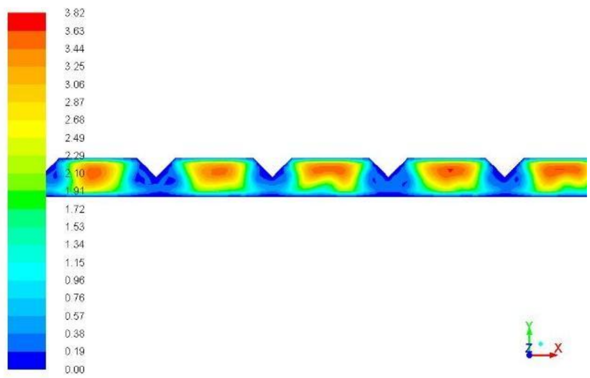

Figure 12: Velocity in the outlet cross section

3.1 Rectangular channels

Above mentioned model contains ribs of complicated shape, which means the complicated modeling and simulation, too. The advantage of the model with rectangular elementary channels is the smaller extent.

To get the operational characteristics V = f(p) of the model, it is necessary to solve 4-5 different pressure gradients for each geometry. For channel cross sections of 6x6, 6x9, 6x12, 6x18 and 6x36 mm the sum of 20-25 solved cases is necessary. For instance the model of channels with cross section 6x6 mm contains 0.9 millions of elements sized 2 mm, and then the total time of solution for all solved cases will be shortened significantly.

For more detailed temperature and velocity fields it would be better to use finer mesh, but it needs longer time of solution. But higher number of elements in cross section of channels is here on the limit of applicability. Boundary conditions are identical with previous model.

Using the same pressure gradient as in the previous paragraph, the thermal output is decreasing here and the negligible mechanical input is increasing a little. As an illustration the distribution of some flow field parameters is presented on Figure 13 to Figure 15.

Eliminating some partitions (i.e. model 6x36) the pressure losses are decreasing and the flow rate is increasing.

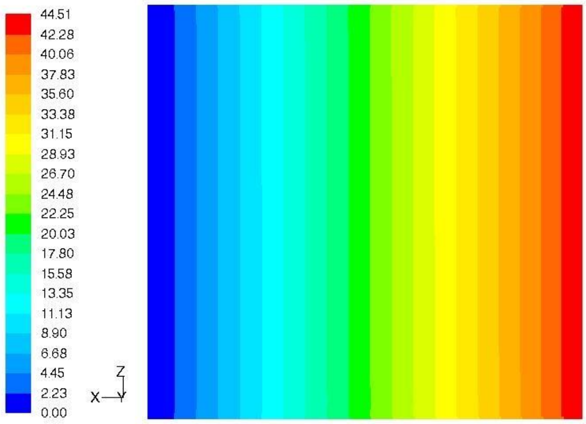

Figure 13: Pressure field of air

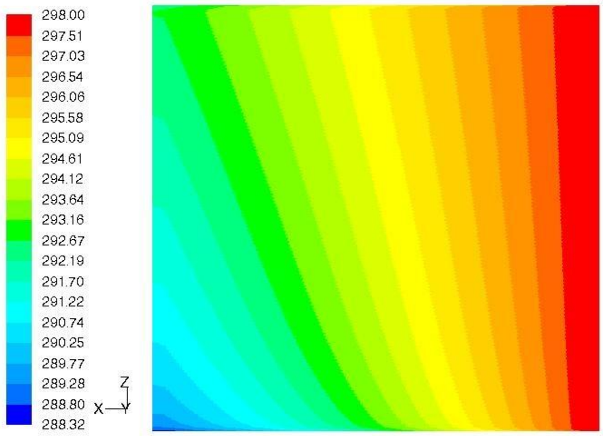

Figure 14: Temperature field of air (flow direction from the right to the left)

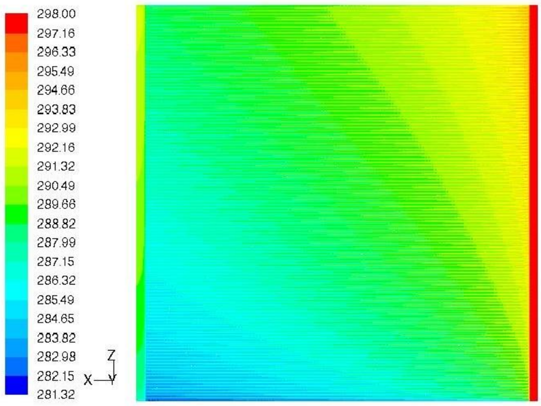

Figure 15: Temperature field on the middle surface (warm air from right, cold air from down)

3.2 Influence of wall thickness

Rectangular channels have simple geometry and maybe here should be used cellular boards known as Makrolon. By assembling of such board there arises double thickness of heat transfer wall between adjacent channels. The influence of wall thickness on the heat transfer coefficient is summarized in the Table 2, using well-known formula for heat passage through simple flat wall

$$1/k = 1/\alpha_1 + s/\lambda + 1/\alpha_2.$$

The basic case 1 contains plastic ribs of thermal conductivity λ = 0.2 W/(m.K) and thickness 0.2 mm. The used heat transfer coefficients between air and wall is preliminary specified as α = 20 or 40 W/(m^2.K), depending on the flow velocity. For expected velocity range up to 5 m/s the empirical formula [13] can be used

$$\alpha\ [W/(m^2.K)] = 5.8 + 4.1\ .\ w\ [m/s].$$

Using double wall thickness (case 2), the heat passage coefficient is decreasing of 0.6%, only.

Using double heat transfer coefficient (case 3) the heat passage coefficient is increasing twice (198.8%) and the double wall thickness (case 4) means its relative decreasing of 1.9%.

The next wall thickness increasing on the real value of cellular board (case 5) means its relative decreasing of 2.9%, compared with case 3 or for double wall thickness (case 6) decreasing of 2.2%, compared with case 4. The difference between high and low conduction (case 7 - aluminum) is negligible.

Result: Use of standard cellular boards could be possible.

No.	λ	α	s	k
	W/(m.K)	W/(m^2.K)	mm	W/(m^2.K)
1	0.2	20	0.2	9.901
2			0.4	9.804
3		40	0.2	19.608
4			0.4	19.230
5		40	0.5	19.048
6			1.0	18.182
7	200	20	0.2	9.9000
			0.4	9.9800

Table 2: Influence of main parameters on resulting heat passage

4 COMPARING OF RESULTS

Actual PC enables to simulate the flow in 1+1 layers of cross flow heat exchanger for outlines of 0.8x0.8 m approx. For comparison here is used the heat exchanger with triangular ribs (6 mm height, 12 mm base, marked as Δ6x12t), see the Par. 2 and the exchanger like the cellular board ("Makrolon") with cross sections of rectangular channels 6x6 mm and 6x36 mm (marked as 6x6t, 6x36t), see Par. 3. From above mentioned results here are presented some basic ones.

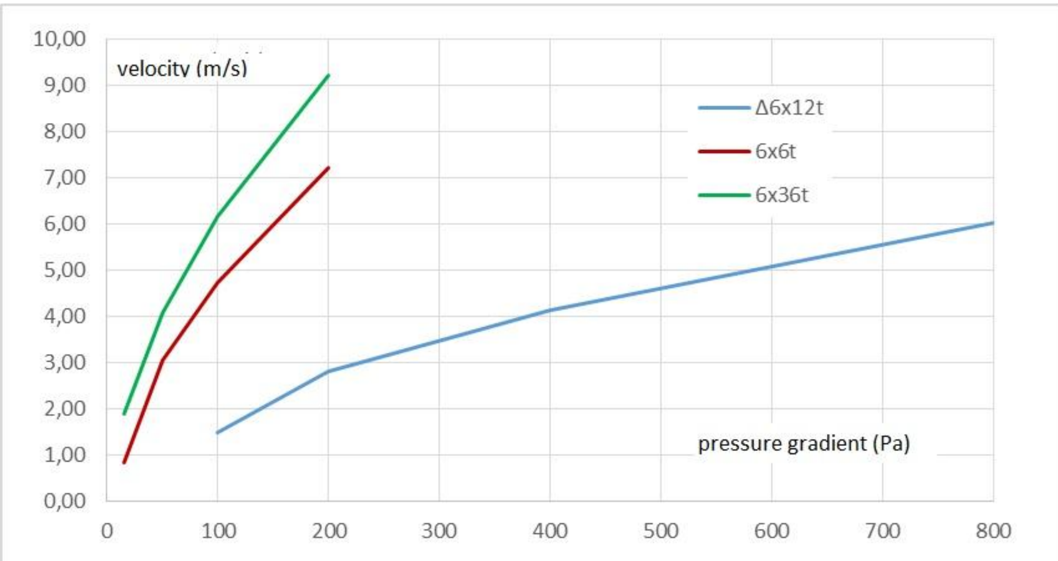

Figure 16: Conversion of pressure gradient on velocity in different geometries (recommended range 1 to 4 m/s)

To create specified range of velocities 1 to 4 m/s approx. it is necessary to specify the pressure gradient. It is different depending on used geometry. The Figure 16 presents the conversion between specified pressure gradient and velocity. It is visible that in smooth channel of small flow resistance the velocity is much higher than in shaped channel, using the same pressure gradient.

Velocity differences between inlet and outlet of the channel or between channels of warm or cold air can be neglected – it is given by different air density at different temperatures. Those differences here are not displayed.

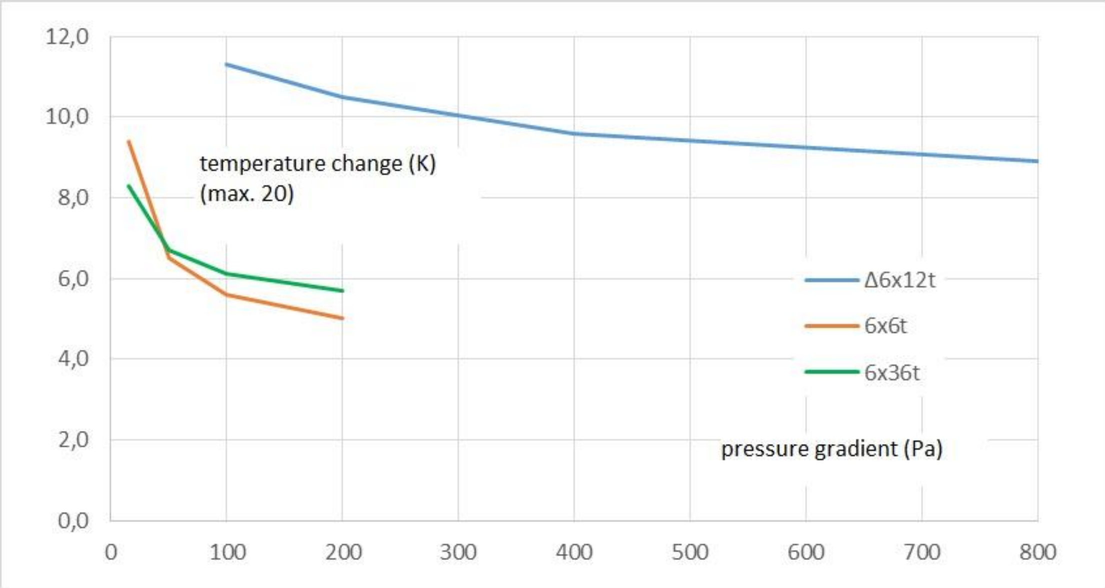

Figure 17: Change of air temperature during passage through heat exchanger

The change of air temperature (i.e. the cooling of warm air or warming of the cold one) after the Figure 17 is typically higher at small flow. With increasing flow the value is going to any value in asymptotic way.

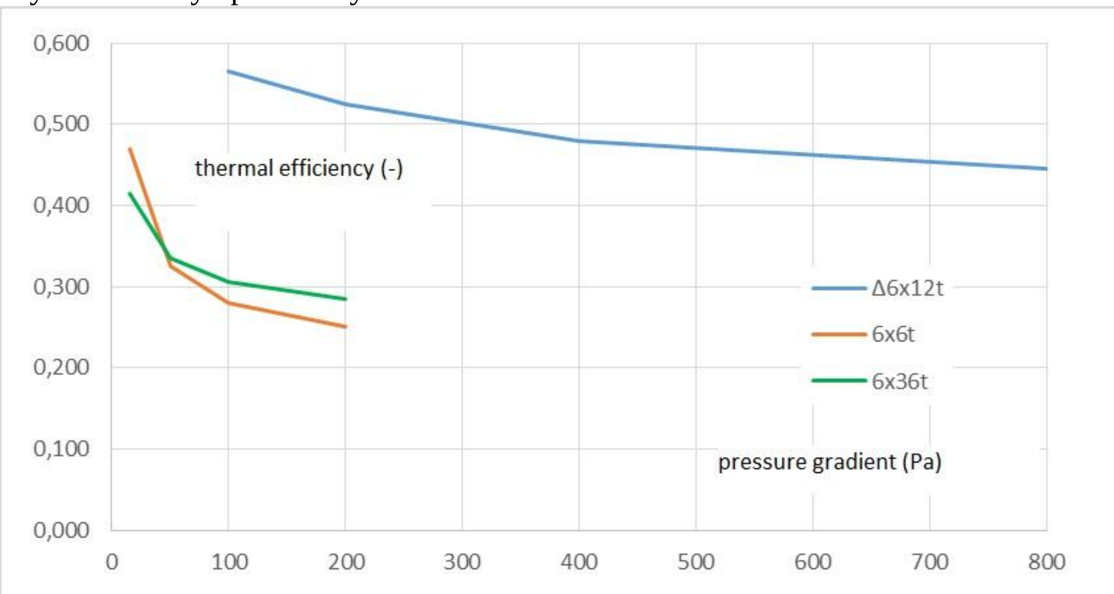

Figure 18: Thermal efficiency of observed heat exchangers

Definition of thermal efficiency: After specified values here is the theoretical maximum temperature change of 20 K (the cold air +5°C would warmed on the temperature of warm air +25°C and vice versa), but practically it is not possible. The rate of real temperature change to the maximum theoretical one is here defined as **thermal efficiency** of the heat exchanger, see the Figure 18.

Remark: Both characteristics (Figure 17 and Figure 18) are identical; the distinction is given by used proportionality constant, i.e. by specified difference between inlet temperatures of cold and warm air.

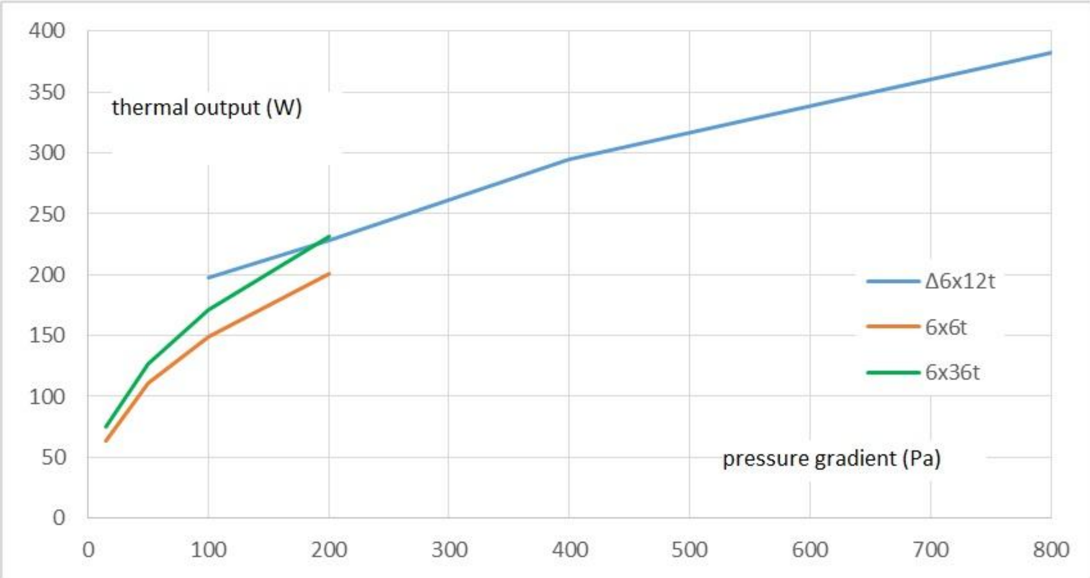

Figure 19: Thermal output of heat exchangers

Thermal output Q = m.cp.ΔT of both kinds of heat exchangers is presented on the Figure 19. The spatially more complicated design (Δ6x12) gives higher thermal output. Its change in dependence on change of flow velocity is slow, compared with smooth channels (6x6t, 6x36t). Their thermal output is smaller, because they have smaller heat transfer surface and higher flow velocity. Thermal output of smooth channels is going near to the output of shaped design until for very high flow velocities (comparing Figure 19 with Figure 16).

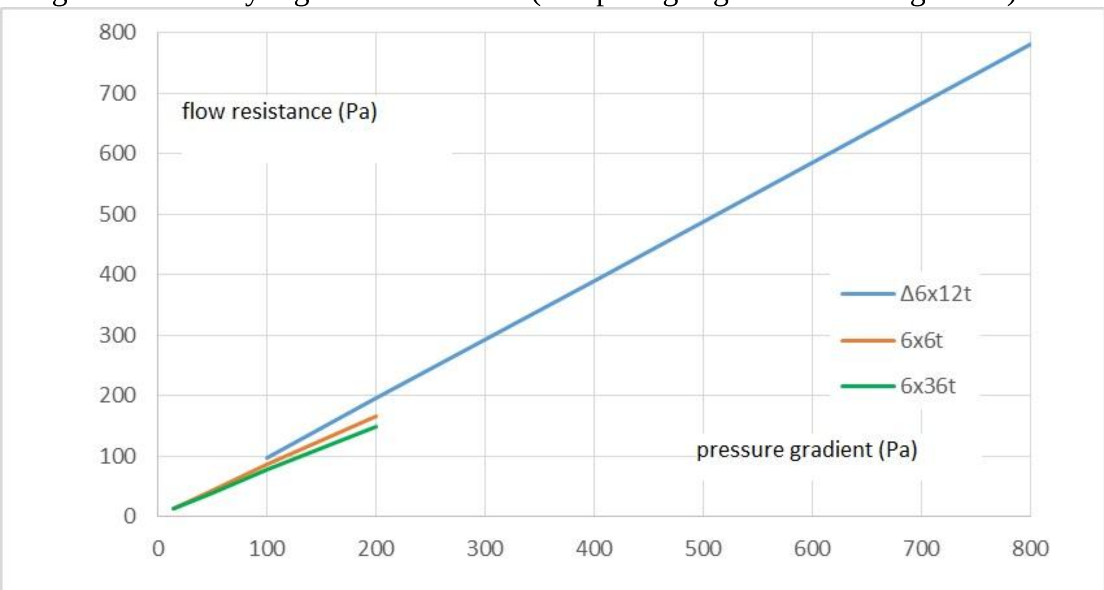

Figure 20: Flow loss of heat exchangers

Flow resistance Δpz = f(w^2) on the Figure 20 is proportional to the specified pressure gradient Δp = f(w^2). Characteristics of both types of exchangers are quite consecutive, but for smooth channels it is visible absolutely smaller loss.

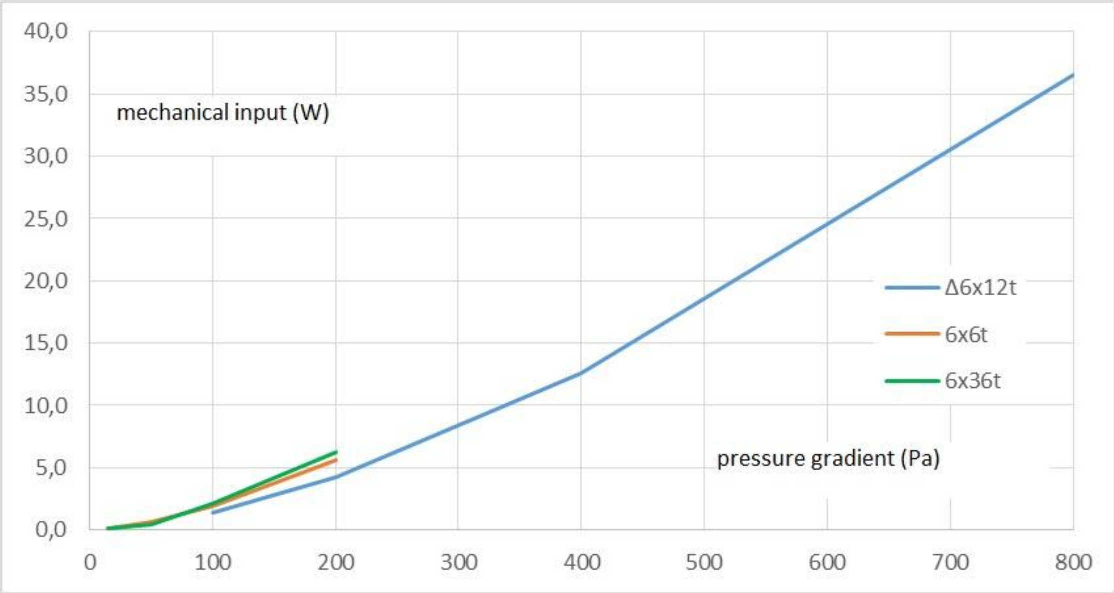

Figure 21: Mechanical input for overcoming of flow resistance

The Figure 21 shows the theoretical mechanical input $P = V. \Delta pz / \eta$, necessary for covering of flow resistance, at the fan efficiency of 100%. Comparing with thermal output, this value can be neglected, even though any real value of fan efficiency would be used, for instance 20%, only.

5 CONCLUSION

Models for numerical simulations of flow and heat exchange in cross flow heat exchangers are simplified on 1+1 layer of cold + warm air, only. For simulation of a complete model of any real arrangement is not actually enough of operational memory and number of parallel processes, too. But the system of two cross flows is repeating and so the total air volume is multiplied.

After some testing simulations the simplest model was used as sole channel of different shape of outlines and of ribs, too. The cross flow is simulated by alternative way as inlet temperature at the inlet in channel (for instance warm air) and constant temperature of channel surfaces (for instance cold air) or vice versa.

The simple model can be used for simple and quick testing of several design arrangements of the geometry on two substantial results: recovered heat output between warm and cold air and consumed mechanical input due to the flow resistance. Only the design, which seems to be hopeful, i.e. of relative small flow resistance and of relative high heat recovery, is suitable to verify on enlarged model and finally by experiment, too.

The more complicated model contains 1+1 layers of cross flows with different inlet temperatures (warm + cold) and with heat transfer surface between them. The outside surfaces are defined as thermally insulated; really here are next layers of cross flows, where the heat transfer is realized, too. Such model observes the heat transfer between two adjoining layers of cold and warm air. It is possible to suppose that the really transferred heat could be similar to the reality and the flow resistance is proportional to the lengths of modeled channels.

Actual standard PC enables to simulate the flow and heat exchange in 1+1 layer of cross flow heat exchanger of outlines 0.8x0.8 m approx. Several results of such testing cases are presented as fields of important flow field parameters and operational characteristics as graphs.

Probably the best solution gives the heat transport surface created as narrow corrugated channels. The optimizing of ribs shape and their arrangement by numerical simulation will determine the amplitude and period of the channel axis, the rib pitch and height, where the minimum rib thickness is given by used production technology.

REFERENCES

[1] "Sick Building Syndrome - Fact Sheet". US EPA

[2] EN 15251 Indoor environmental input parameters for design and assessment of energy performance of buildings-addressing indoor air quality, thermal environment, lighting and acoustics.

[3] Sick Building Syndrome: Modern Day Dilemma - http://www.home-air-purifier-expert.com/sick-building-syndrome.html.

[4] Sick Building Syndrome. *The Environmental Illness Resource Team* ei-resource.org/illness-information/related-conditions/sick-building-syndrome/

K. Adámek, J. Kolář, P. Peukert, Sick Building Syndrome Suppressing by Room Ventilation, *WSEAS Transactions on Fluid Mechanics*, Vol. 11, **2015**

[6] Adámek, K.: Problems of industrial air exhausting, *5th ECCOMAS*, 2014, p. 7454-7465

[7] Ventilating slot „Regel-air“, www.aluplast.net

[8] Pavlů, M., Ventilation for windows suppliers, *Int. Conf. ISOZ*, Liberec, 2015

[9] Adámek, K., Kolář, J., Peukert, P.: Room ventilation, *Proc. of the 13th International Conference on Heat Transfer, Thermal Engineering and Environment (HTE 15*), p. 49-56, Salerno, June, 27-29, 2015

[10] Adámek, K., Kolář, J., Půlpán, P., Pustka, M., Noise Reduction at the Fan Outlet, *Proc. of the EEED 2013, International Conference on Energy, Environment, Ecosystems and Development,* Rhodes, July 16-19, 2013

[11] Adámek, K., Pavlů, M., Bandouch, M., Solar ventilation and tempering, *Proc. of Int. Conf. - XIXth Applic. of Exp. and Num. Meth. in Fluid Mech. and Energ., 2014,* AIP Conf. Proc. 1608, 1 (2014)

[12] ČSN EN 308: Method of laboratory testing for devices of heat recovery air-air

[13] Černoch, S.: Strojně technická příručka, SNTL Praha, 1967