International Journal of Engineering Technology and Scientific Innovation

ISSN: 2456-1851

Volume:02, Issue:06

ROD GAS BURNER IN A REAL OPERATION

Pavel Peukert, Jan Kolář and Karel Adámek

Dept. of numerical simulations, VÚTS Liberec, a.s., CZ-46119 Liberec

ABSTRACT

The aim of this study is flow numerical simulation in gas burner body, equipped by outlet sieve. Regarding the complicated system of burner, simulations of flow fields are without combustion, well describing the actions inside the system. For better flows imaging, the inlet flow of fuel mixture is "colored" by higher temperature. Partial results document images of important parameters of the flow field, as velocity, temperature, turbulent kinetic energy, streamlines. The conclusion generalizes the results of the study, important for next development.

Keywords: component; flow numerical simulation; gas burner; sieve;

I. INTRODUCTION

The aim of this study is flow numerical simulation in gas burner body, equipped by outlet sieve. Regarding the complicated system of burner, the preliminary simulations of flow fields are without combustion, but well describing the actions inside the system. Partial results present the images of important parameters of the flow field, as velocity, temperature, turbulent kinetic energy, streamlines. The conclusion summarizes main generalized results of the study. Next parts of the solution describe the flow in complicated surroundings, quality of the combustion etc.

II. PRELIMINARY MODEL OF BURNER

<u>Geometry</u> - In upper wall of prismatic burner body is created system of various holes, rows of holes continue as common rectangular channel [1]. The common inlet is situated transversely in the middle, see the Fig. 1. The aim of such bar burner is to get uniform temperature field at the outlet.

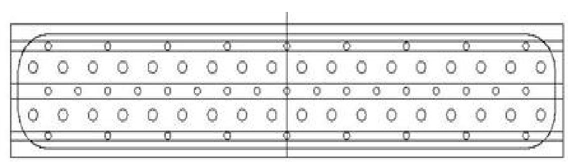


Fig. 1: ***Ground plan of burner***

<u>Mesh</u> – Relative complicated and fine, due to small dimensions in general.

<u>Boundary and initial conditions</u> – The common inlet is through the transverse tube,

ISSN: 2456-1851

Volume:02, Issue:06

outlet is through walls of the added outlet block, connected on the burner body and presenting the surroundings. Cross plane of symmetry is used.

Across of rectangular channels, connected on individual holes, are defined planes of porous jump, replacing complicated shapes of inserted sieves. Permeability parameters were evaluated from both measuring and simulation.

The primary model is solved without combustion, the spreading of the fuel and/or burnt gases in surrounding area are modeled as higher temperature of the inlet air (of 100 K), relative to the surroundings, where proceeds the mixing of both environments.

<u>Solver</u> – Spatial stationary model of viscous compressible flow, turbulence model k-ε.

Model with burning uses the mixture of methane and air in stoichiometric ratio of 10:1.

<u>Results</u> are received from standard commercial code [2] – Due to large number of images of the flow field there are eliminated the relevant scales. In general, the red is the maximum value and blue is the minimum value – according to wave lengths in the spectrum of visible light. Here are not substantial the absolute values of solved cases, but their qualitative character or distribution in the volume of model.

A. Test of flow uniformity in the burner volume

The first checking simulation, if the flow is uniform in the whole volume of burner. The common cross section is twice larger as the sum of all small orifices. So that the inlet can be regarded as pressure chamber with similar outlet velocity at all orifices.

Good filling of the inner volume by the inlet flow on the velocity field (0…13 m/s) in Fig. 2 and uniform streamlines from inlet through burner chamber and orifices in the surroundings in the Fig. 3 verify good design of burner body.

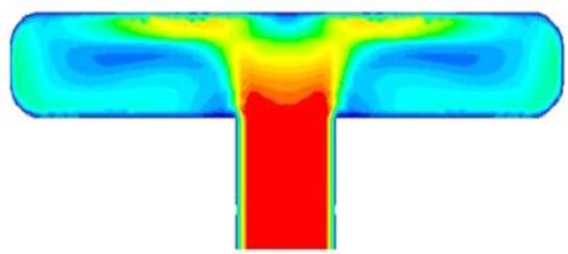
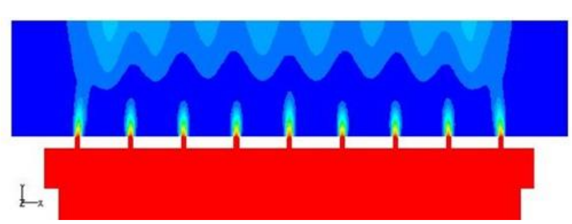


Fig. 2: *Inlet and burner chamber*

The Fig. 4 presents set of temperature fields in vertical lengthwise sections in the rows of orifices 1-2-3 (from the main inlet). In the rows 4-5 the situation is similar to rows 2-1, therefore not presented here. To get better contrast, the inlet flow is “colored” by higher temperature – it is of 100 K higher than in the surroundings, pressure gradient of 100 Pa.



ISSN: 2456-1851

Volume:02, Issue:06

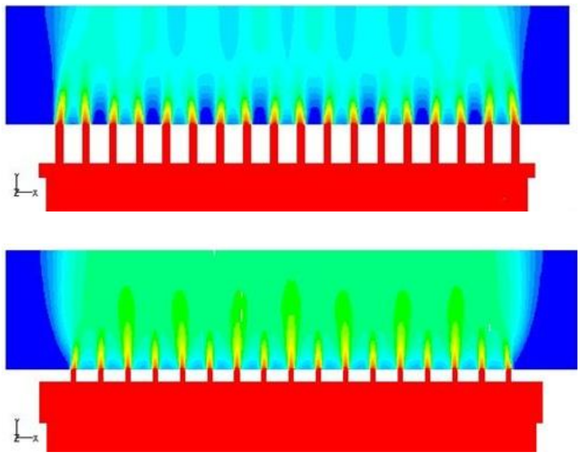


Fig. 4: ***Lengthwise sections (rows 1-2-3) of the temperature field (300...400 K)***

The next set in the Fig. 5 presents the temperature fields in typical cross sections through various combinations of 1-2-3 orifices. It is visible slightly different shape of mixed outlet flows, their mutual good equalizing, independent on the section position from the burner center and a good mixing with surrounding air – really it is the mixing of the secondary air, necessary for good combustion. Only three typical sections are presented here, repeated along the whole burner length – to compare with the Fig. 1. Boundary conditions are the same as in the Fig. 4.

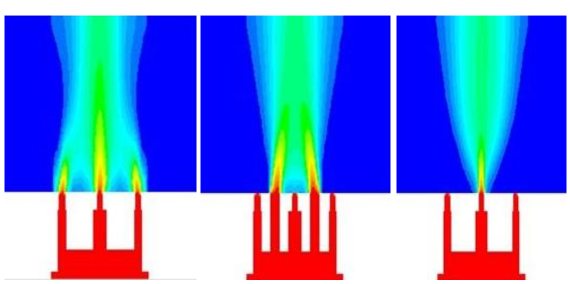


Fig. 5: ***Set of cross section of the temperature field (300...400 K) in typical rows of inlets***

The Fig. 6 presents the temperature fields in ground plans 5 and 20 mm from the plane of orifices. For defined inlet temperature of 100 K higher than in the surroundings the corresponding maxima are +65 a +43 K towards surroundings. Temperatures are well balanced; local small differences correspond to the number and dimension of holes. If necessary, it is possible to correct it by adding small holes or by dispersion effect of inserted sieve. It is well visible the air suction from the surrounding by the ejection effect of flows, outgoing from individual holes.

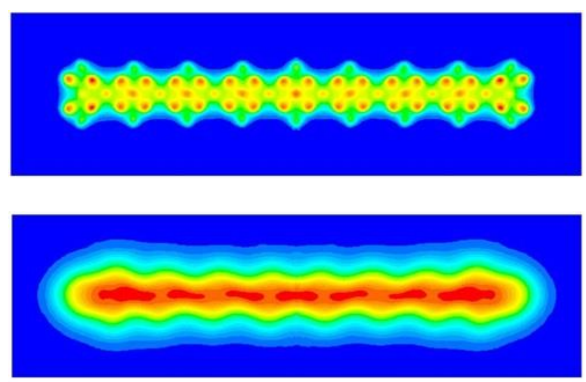


Fig. 6: ***Temperature field (300...400 K) - ground plans, at distances 5 and 20 mm from outlets***

The Fig. 7 presents velocity fields related to above mentioned temperature fields of the Fig. 6 (maxima of 20.0 and 9.5 m/s – to get good contrast of isolines there are used different ranges). The character is corresponding to the number and positions

ISSN: 2456-1851

Volume:02, Issue:06

of orifices. At short distance of 5 mm are visible contours of individual flows, at longer distance of 20 mm the details are not so sharp.

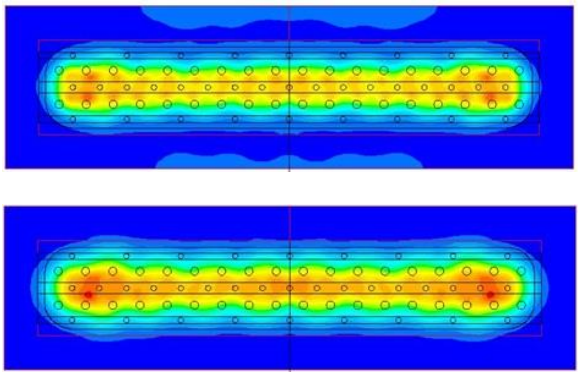


Fig. 7: Velocity fields (m/s) in ground plans

III. ELEMENTARY CHANNEL WITH SIMPLE SIEVE

The uniformity of global field is confirmed above, so the next detailed considerations and sets of solved variants were restricted on elementary channel with one orifice, only, equipped by sieve - reputedly installed for dispersion of flame and burnt gases. It is supposed that such sieve equalizes the punctual effect of flows from individual holes. According to the results of the previous section, the flow equalizing is good here without sieve. Here is solved the influence of such sieve on global character of the flow field.

Geometry - The real shape of sieve element after the documentation is modelled on the Fig. 8, repeating pitch is limited by walls and by periodic boundary condition. Just before and after the sieve volume are situated two auxiliary planes for tracing the influence of inserted sieve on changes of pressure, velocity and turbulent kinetic energy. This volume is followed by standard prolongation of the outer area in both directions (before and after).

The real shape of the sieve must be simplified; deformations of individual wires during manufacturing and assembly are neglected. The Fig. 8 shows one module of the observed sieve. The volume of this element of 0.82x0.82x0.32 mm is comparable with holes diameter of 0.8-1.0 mm, therefore the resulting flow field depends also on the mutual position of the wire and orifice. In larger channels of 1-1.6 mm in width are 3-5 wires of the sieve, only.

Mesh – The singular points, it means mutual contacts of wire-wire and wire-wall are eliminated by small approach of both bodies or by inserting of small volumes (sphere, cylinder) around each singular point, see the Fig. 9. Theoretical singular points make problems at model meshing and at convergence of solution.

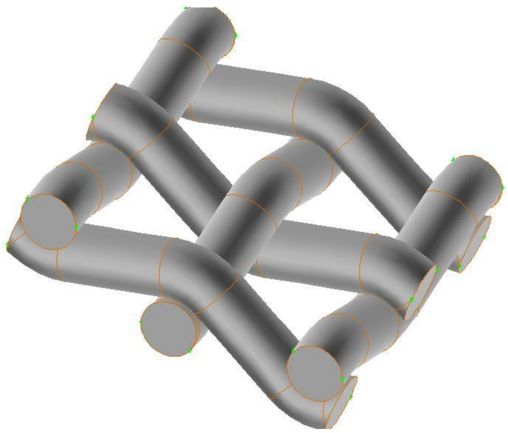


Fig. 8: ***Sieve element***

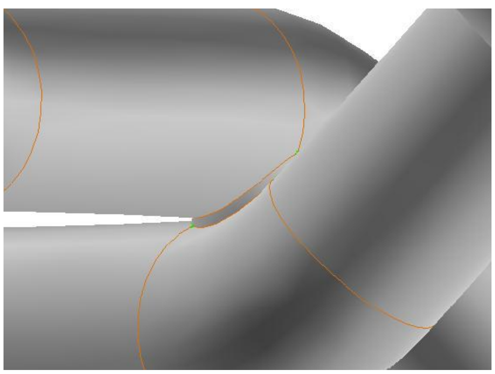


Fig. 9: ***Chamfer instead singular point***

Boundary and initial conditions and solver are the same as in previous section.

Results – To evaluate the flow turbulization by inserted sieve the values of air permeability for designed sieve shape are used.

Two auxiliary planes, situated ±0.25 mm before and after the middle plane of the sieve, present next results (Fig. 10).

Pressure field – local pressure increasing in the stagnation area, where the flow hits the sieve, is projected in the area before the sieve, too.

In the velocity field is visible an expressive wake after the sieve and an expressive velocity increasing between neighboring wires.

In the field of turbulence kinetic energy is visible small increasing in the wake after the sieve, and higher increasing in the flow between wires.

The character of fields is similar for different pressures, therefore without scales.

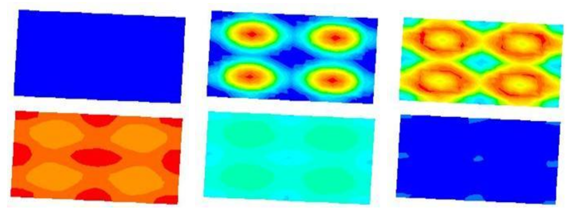


Fig. 10: ***From the left: pressure, velocity and turbulence field– cross sections before and after the sieve***

On next Fig. 11 are the same parameters in the lengthwise axial cross section through the model. Pressure is decreasing due to pressure resistance of the sieve, in the velocity fielod are local wakes after each wire and local maxima in eyes between wires, the character of the turbulent kinetic energy is similar.

ISSN: 2456-1851

Volume:02, Issue:06

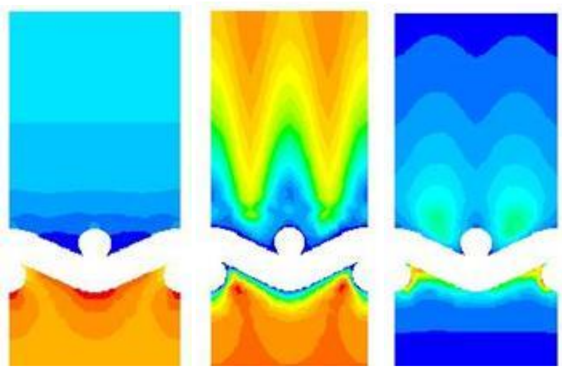


Fig. 11: ***From the left: pressure, velocity and turbulence field in lengthwise axial cross section***

A. Sieve permeability

Due to the complicated shape of the sieve it is not possible to model it in whole burner. Therefore in the numerical model is inserted the boundary condition „porous jump“. Its necessary parameters are evaluated by simulation or by measuring. Detailed procedure was described in [3] etc., here are presented real results, only. Graphs in the Fig. 12 present the evaluation of sieve permeability by simulation (up) and by measuring (down); values of both permeability parameters are in good coincidence (+/-5%), the absolute error less than 4%, even if the measuring of very permeable sieve was possible up to 1 Pa, only.

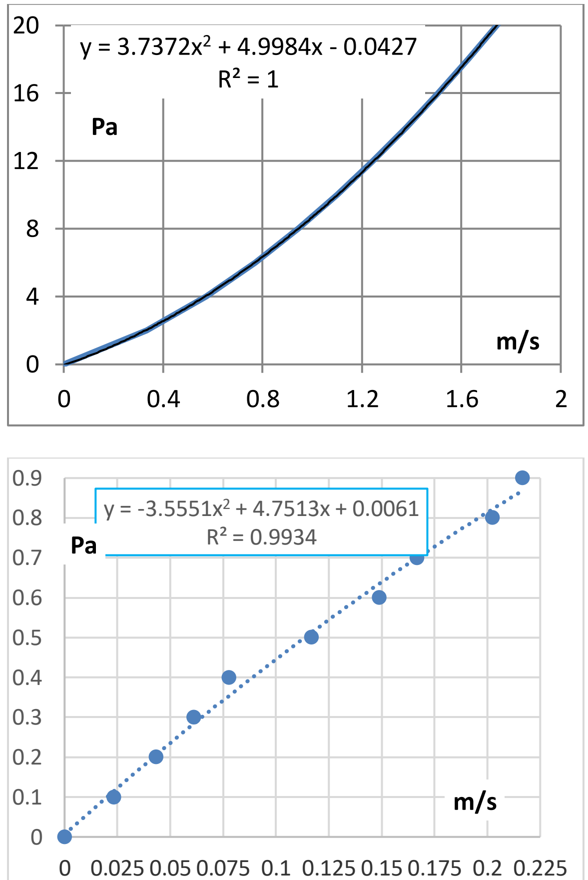


Fig. 12: ***Sieve permeability***

- simulated C2 = 1,246e+4 (1/m), α = 7,223e-3 (m^2)

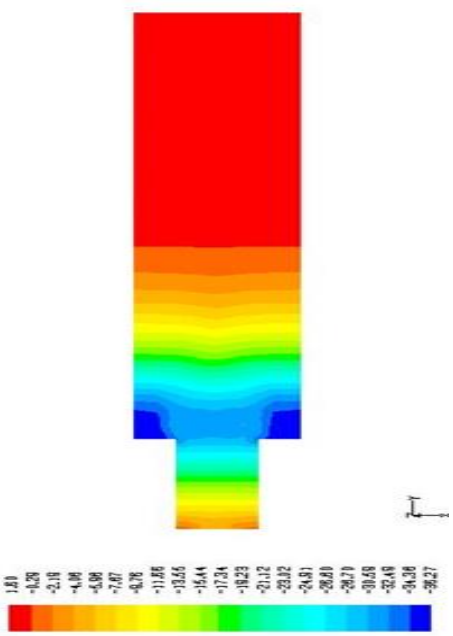
- measured C2 = 1,185e+4 (1/m), α = 7,598e-3 (m^2)

B. One elementary orifice of burner

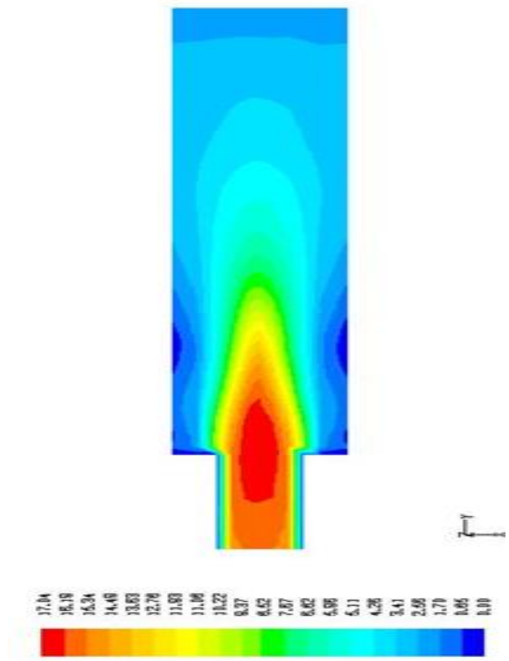
This is a checking model of the flow through one orifice of 0.8 mm in the middle row (pitch of 10 mm), leading into the channel of 1.0 mm width, using crosswise oriented sieve at the channel outlet. The aim is to determine the influence of the sieve, its flow resistance and of turbulence on global outlet flow field. Preliminary – the mutual affecting of neighboring flows is low (regarding the Fig. 4).

The creation of the model is similar as above, so without repeated details here, permeability parameters after the previous section, pressure gradient of 100 Pa.

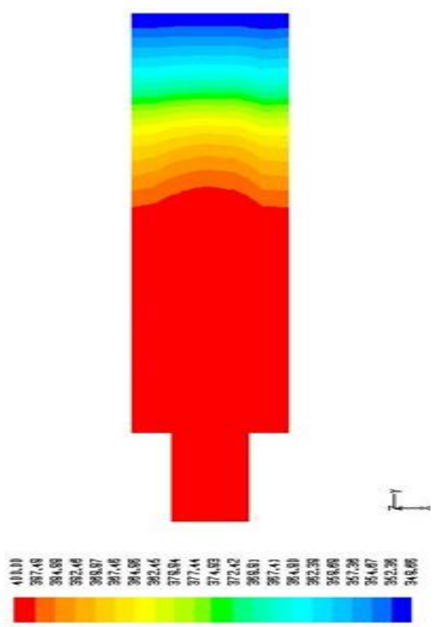
The Fig. 13 presents the basic model of the transition from cylindrical inlet orifice into rectangular outlet channel, without sieve. All observed values (pressure, velocity, temperature, turbulent kinetic energy) correspond to the well known flow through the stepped channel.



(-35...0 Pa)



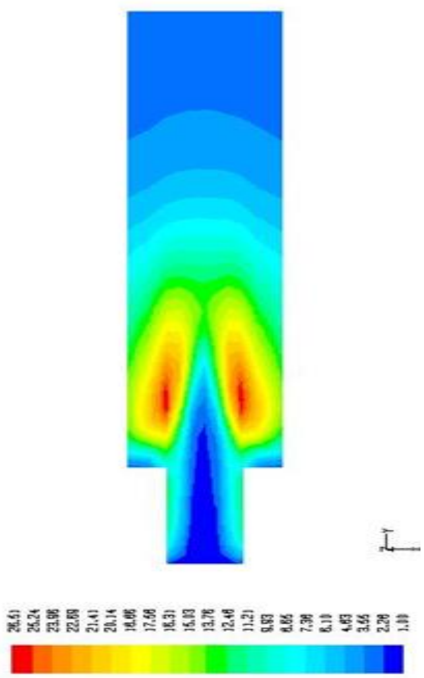
(0..17 m/s)



(350...400 K)

ISSN: 2456-1851

Volume:02, Issue:06



(0..26 J/kg)

Fig. 13: ***Flow field parameters in hole + channel (pressure, velocity, temperature, turbul. kinet. energy)***

Pressure field – it is visible fully expanded flow just after inlet, not any sieve resistance. Just after the change of cross section there is wake with under pressure, the leveling on the surrounding pressure is coming along the length of about 5 inlet diameters.

Velocity field - constant inlet, typical fading out of the flow with wakes at walls, due to the sudden enlargement of the channel.

Temperature – inlet flow (hot) successively mixing with the cold surroundings.

Turbulent kinetic energy - maximum at the transition between cylindrical and rectangular channel.

As results of next solved cases are presented velocity fields, only, the other parameters of the flow field are logically connected as above, see the Fig. 14.

Using one distant sieve (left, 0…13 m/s) – the velocity is dissipated by distant sieve.

Using two sieves (middle, 0…10 m/s) - on the first sieve, placed just after orifices, the velocity is decreasing, on the second one is hardly visible any next change.

In the diffuser with sieve at the end (right, 0…13 m/s) there is fluent deceleration and uniform profile after the sieve situated at the diffuser end.

Summarizing Table I. shows, that the sieve operates like inserted resistance, which decreases the flow, then the heating input, too. The sieve is modelled as homogenous porous jump, really the sieve wires diameter is very similar to the inlet diameter, and therefore it should be to take into account some influence of actual mutual position of wire toward the inlet.

ISSN: 2456-1851
Volume:02, Issue:06

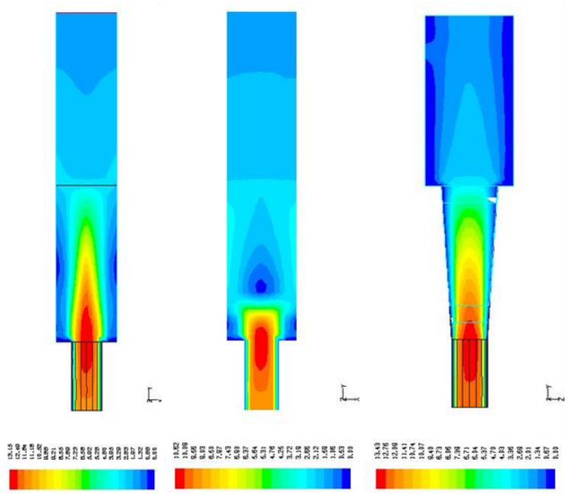


Fig. 14: ***Velocity fields – various sieve positions***

Table 1: Flow in one inlet – summary of results

No.	flow	velocity	inlet press.	remark
	g/s	m/s	Pa	
1	10,25e-3	15,19	-3,2	no sieve
2	7,72e-3	11,44	41,3	1 sieve
3	6,10e-3	9,04	63,2	2 sieves
4	8,09e-3	11,98	35,6	diffuser + sieve (inlet)
4	8,09e-3	3,05	41,4	diffuser + sieve (outlet)
5	11,02e-3	16,33	-18,9	diffuser only (inlet)
5	11,02e-3	4,28	-0,66	diffuser only (outlet)

IV. INFLUENCE OF SIEVE SHAPE

The procedure of model creation is similar as above, therefore no details about it. Sieve volume is arranged between the inlet and outer surroundings. Across the narrow channel only two sieve eyes are placed, see the Figures below.

A. Sieve across the orifice mouth

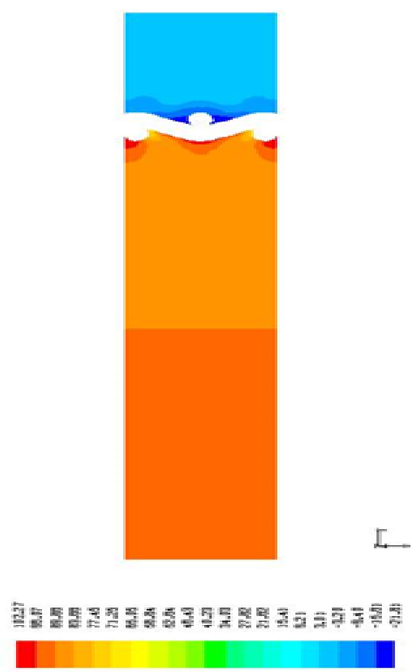
Main parameters of the flow field (pressure, velocity, temperature, turbulence) are on the Fig. 15, following from the left:

Pressure (-20…+100 Pa) – sieve resistance means not any important pressure decreasing before the sieve.

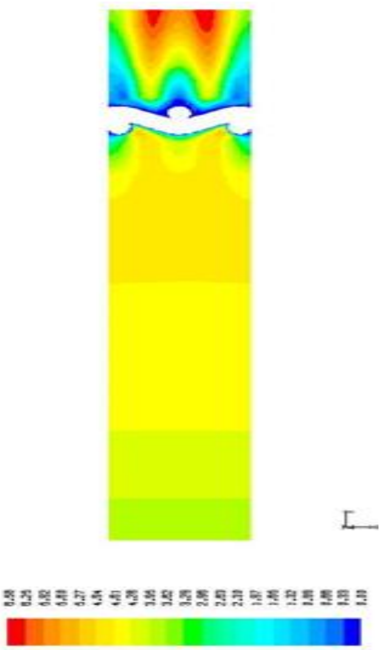
Velocity (0…0.6 m/s) – wake after the wire, maximum value in the eye between wires.

Temperature (300…320 K) – mixing of inlet and surroundings after the sieve.

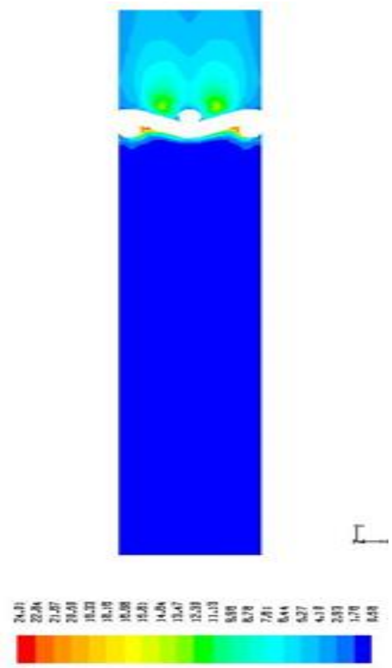
Turbulent kinetic energy (shortened as tke, 0…0.24 J/kg) – maxima around the sieve, helps for mixing.



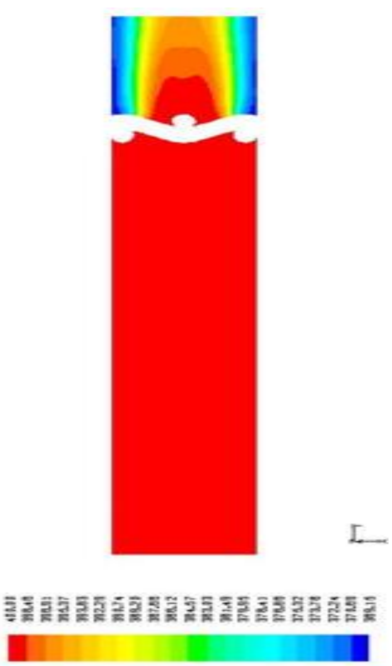
(-20…+100 Pa)



(…0.6 m/s)



(…0.24 J/kg)



(300…320 K)

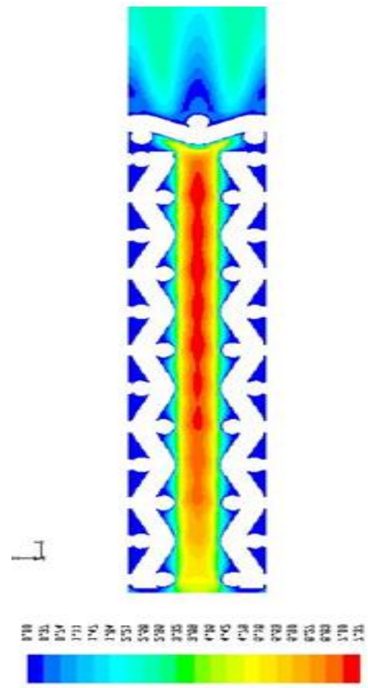
Fig. 15: ***Flow field parameters (pressure, velocity, temperature, tke) for the sieve situated across the flow***

B. Sieve "U"

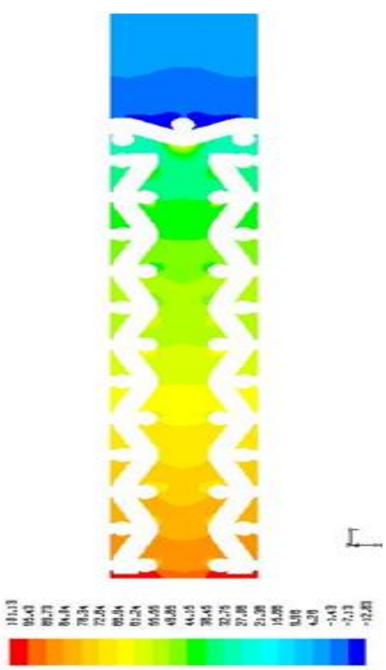
The sieve is inserted in the channel as U-shape, fixed by its elasticity and friction. The sieve is modelled as repeating of basic module from the Fig. 8, see the Fig. 16. Sieve deformations due to the manufacturing and forming are neglected, but it is clear that influence of such deformation in given dimensions can be substantial.

ISSN: 2456-1851

Volume:02, Issue:06



(0…7.2 m/s)



(-20.+100 Pa)

Fig. 16: ***U-shaped sieve and its pressure and velocity field***

It is evident that the flow through the sieve layer along channel walls is practically none. The sieve volume decreases the free cross section at 50% approx.; therefore the velocity in the middle is increasing, but the flow resistance too.

Results of several similar solved cases are summarized in the Tab. 2. The inserted sieve markedly cloggs the flow cross section. Comparing the most clogged case of the sieve „O“ with free channel „N“ the cross section is 28%, only, the flow of 7% and the velocity of 3%, so the pressure before the sieve remains the same. Maximum turbulence is on the contours of individual

ISSN: 2456-1851

Volume:02, Issue:06

wires, where the mixing effect is the maximum.

Table 2: Main parameters for various shapes of the sieve

sieve	flow	velocity	press.	tke	cross section
	mg/s	m/s	Pa	m^2/s^2	%
None	4,472	10,770	45,1	4,034	100
I-cross	1,401	3,364	95,0	7,265	100
U-shape	0,912	2,043	97,2	4,730	52
O-double	0,305	0,688	99,7	2,296	28

C. Conclusion

The sieve inserted into or after gas burner helps to create more uniform velocity and temperature field, at the same time the flow resistance is increasing and the heat input is decreasing. But the best way for an optimum design is suitable number, dimensions and positions of individual holes, to create uniform flow field in advance. In such case an additional correction by added sieve is not necessary. Presented method of flow numerical simulation can help to find such optimum arrangement.

For presented study of the flow field in and after rod gas burner is not necessary to use a combustion model. It was used for final test of combustion in an optimum design of burner.

V. INFLUENCE OF SURROUNDINGS

The burner studied above is arranged in a real device, where some next influences on the flame area arise.

A. Basic model

Geometry - Due to the symmetry plane of the burner, the one-half model is solved, only, to shorten the time of solution, see the Fig. 17. From the left side up there is cross-inlet, in the middle part there is the inlet chamber of burner with outlet holes and following rectangular channels, finished by dispersive sieves. As proved above, the sieve decreases the flow cross section and to the total resistance is higher.

The model area is limited below by supporting construction for treated material, which is here preliminary neglected – really, it increases the total flow resistance in the lower outlet.

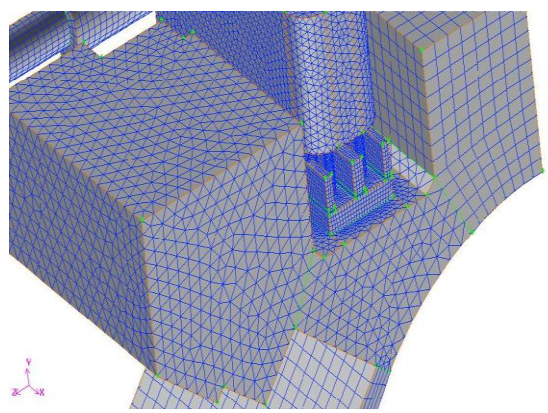


Fig. 17: ***Model geometry and mesh***

Mesh – The mesh in small diameters must be fine enough. Due to relative complicated shape, in some volumes must be used tetrahedral mesh, total 0.5 mill. of elements approx.

Boundary and initial conditions – Inlet of fuel mixture of diameter of 6 mm, flue gas outlet in gaps between burner, and other solids.

At the outlets from burner is defined so-called „porous jump“, as substitution of the flow resistance of inserted sieve – its parameters see above.

Similarly, in the lower outlet is defined the resistance of treated material.

In volumes of 0.5 mm before porous jump is defined source of turbulence, instead the inserted sieve, too.

The condition of symmetry is defined in the middle vertical plane of inlet channel, to decrease the extent of model.

In all inlets in holes etc. are defined conditions „interior“, for possible recording and evaluation of flows in those sections – to test the flow uniformity.

Preliminary model is without combustion, therefore for better contrast the inlet flow is "colored" by higher temperature.

Because the gas density changes with both temperature and pressure, it is necessary to use as inlet/outlet conditions the pressure difference between them. Used iterative procedure determines this difference, which creates the given flow in the solved model, to fulfill the needed heat output of the device.

Note: For small pressure gradients it is possible to define mass flows, too – used combustion model it expects. Necessary pressure difference for those mass flows are result of simulation.

<u>Solver</u> – 3D flow of viscous compressible gas, turbulence model k-ε. It assumes the disturbed flow due to various shape irregularities, although the calculated static Reynolds‘ number corresponds to the laminar flow.

<u>Results</u> – images below present typical parameters of flow fields. Despite some inaccuracy in geometry several new knowledge is received – how to increase the flow through burner and its output and efficiency, too.

The Fig. 18 to Fig. 20 present the flow field (streamlines, velocity, temperature) in the symmetry plane. In the other cross sections the results are similar. At the first view, the flow field in the working area of lower wedge seems to be uniform, but from the temperature field is clear that significant part of the flow escapes without effect in the surroundings.

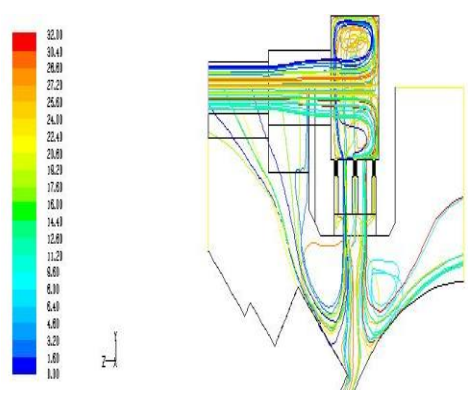


Fig. 18: *Streamlines in cross section*

ISSN: 2456-1851

Volume:02, Issue:06

Inlet from upper left channel into burner volume, turned down through burner holes into lower working area.

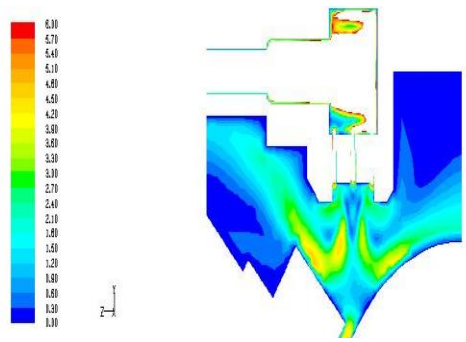


Fig. 19: ***Velocity field (suppressed scale)***

In the working area (lower wedge) exists the stagnation area with low velocity, therefore with low heat transfer, too.

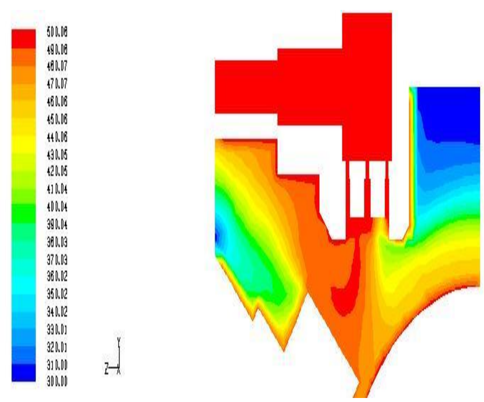


Fig. 20: ***Temperature field***

Significant escape of hot inlet flow in side surroundings. The heating of fuel inlet (left up) is not suitable, when the mass flow of the fuel mixture is decreasing. Additionally, from the right side is coming the cold air from the surroundings, the working area is cooled.

B. Model with labyrinth

Simple labyrinth on the left side, composed from two flat partitions, suppresses the excessive escape of hot flue gases in the left side. Although the shape of the partition is random, it confirms the rightness of the hypothesis. On the Fig. 21 to Fig. 23 there is suppressed the flow of hot flue gas in both sides (left and right), the flow along (across the plane of paper) makes no problem, because it warms the treated material, coming in this direction. After the Fig. 24, the temperature in the lower working area is higher.

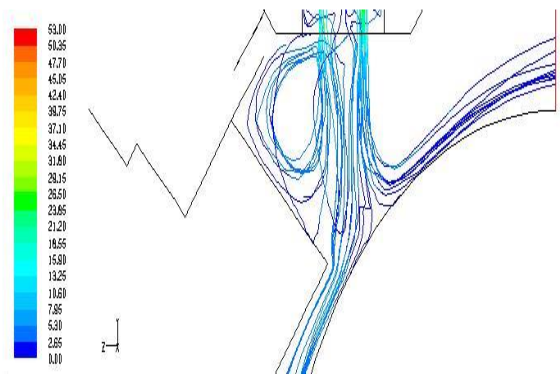


Fig. 21: ***Streamlines in the working area***

Simple labyrinth on the left side suppresses the escape of hot flue gases in the left side.

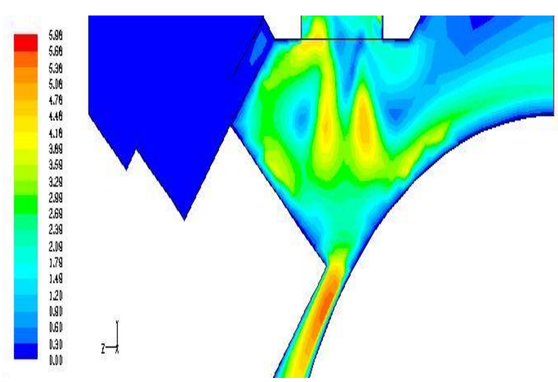


Fig. 22: ***Velocity field in the working area***

In this area the treated material is heated.

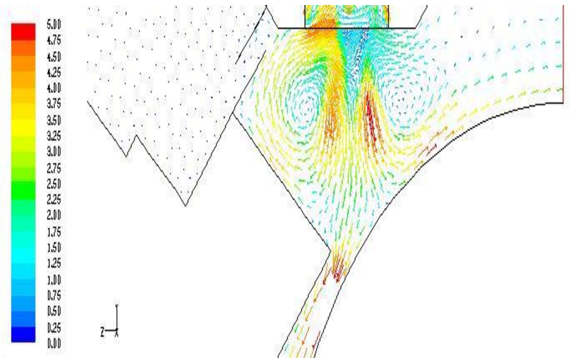


Fig. 23: ***Directional field in the working area***

Flue gases, initially escaping in the left side, remain in working area, due to the installed labyrinth.

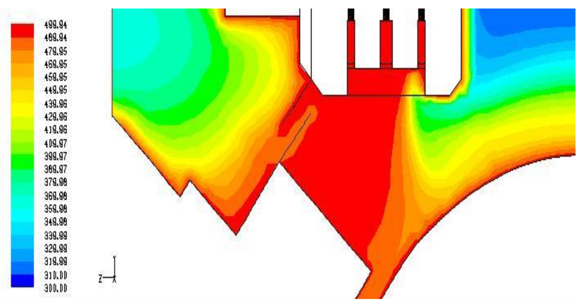


Fig. 24: ***Temperature field***

In the working area the temperature is substantially higher than without the labyrinth (see the Fig. 20 above).

Using next modification, it is possible to increase the effect. For instance to install next partition on the right side, to change gaps between partitions etc. It is necessary to ensure the inlet of secondary combustion air into the working area, for instance through perforated partitions. Given ratio of premixed gas with air of 10:1, given by the equation of burning, corresponds here to the air excess of $\lambda = 1.05$, therefore it is possible that such secondary air inlet will not necessary.

To be sure that the combustion is OK, it is always good, if some inlet of the secondary air exists. The primary mixture, with small lack of air, is ignited well and the secondary air, necessary for good combustion in the entire volume, will added in the burning mixture. The air excess in primary mixture can lead to the flame separation, lack of the secondary air can lead to the imperfect combustion, soot formation etc.

C. Model with combustion

The basic model above is completed by combustion model "Species – transport and reactions" for used mixture of methane and air, their volumetric reactions, with inlet diffusion, with interaction between turbulence and chemical reactions of type EDC etc. The calculation uses lowered under relaxation parameters and is finished in the second order of precision. Therefore, much longer time of each solved case. The procedure of the model creation is the same, therefore not mentioned here.

The temperature field arises here by simulated chemical reactions; next principal parameters of the flow field as above are completed by fields of concentration of individual components, participating in chemical reactions during burning.

ISSN: 2456-1851

Volume:02, Issue:06

Next Fig. 25 to Fig. 27 present the main parameters of the flow field and the Fig. 28 to Fig. 32 complete the fractions of components during the combustion – both fuel mixture and flue gases.

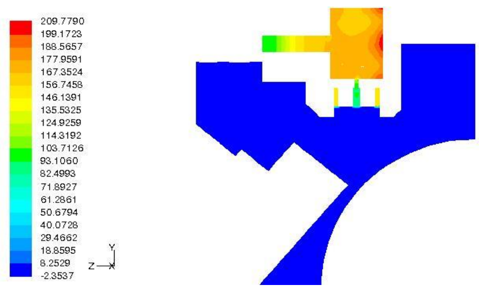


Fig. 25: *Pressure*

Higher pressure in the inlet and in the burner expands on the zero pressure in surroundings.

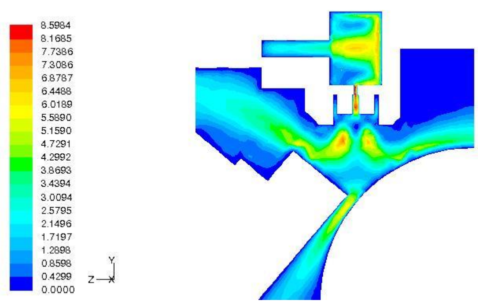


Fig. 26: *Velocity*

Velocity arises from the change of pressure energy in the kinetic one; the influence has the higher volume of hot flue gases, too. Considerable volume is escaping in both sides. At the bottom, where moves the treated material, is the stagnation area of lower velocity of hot flue gases.

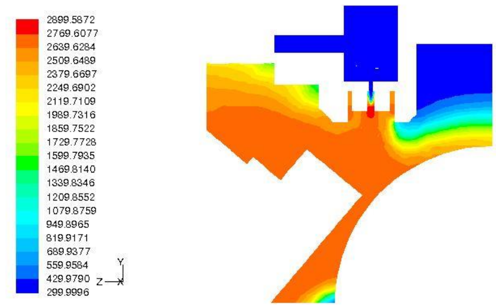


Fig. 27: *Temperature*

The temperature increases by fuel burning. It is visible, that more heat is flowing to the left side and on contrary from the right side is coming some volume of cold air from surroundings.

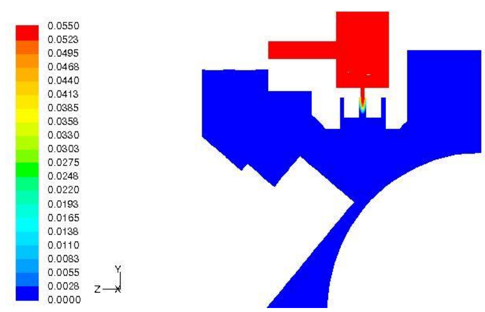
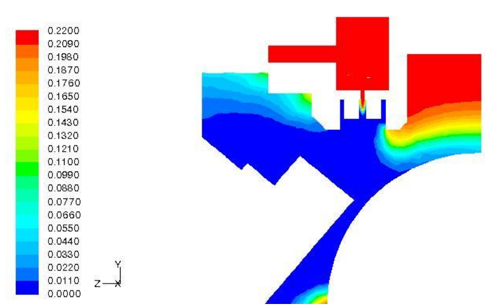


Fig. 28: *Mass fraction of methane*

Check of burning – it takes place correctly in mouths of individual holes in wall of burner body, the volume under burner is full by hot flue gases (methane fraction is zero here).



ISSN: 2456-1851

Volume:02, Issue:06

Fig. 29: ***Mass fraction of oxygen***

Check of burning – in the working area under the burner is not any oxygen (consumed for methane burning). Is visible the cross flow from the right side to the left one.

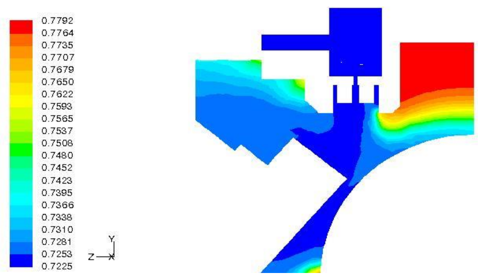


Fig. 30: ***Mass fraction of nitrogen***

Some light change of the concentration is visible, but differences are absolutely low, some numerical error of used simulation. Nitrogen does not participate in burning process.

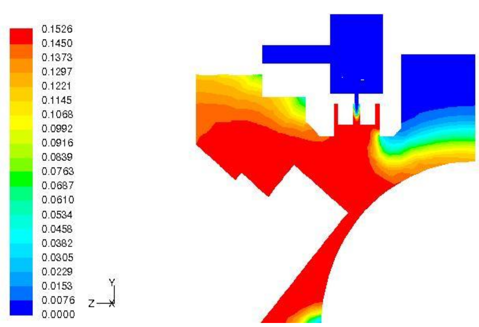


Fig. 31: ***Mass fraction of carbon dioxide***

Check of burning – maximum value after burning of whole mass of fuel, the reached maximum corresponds to the equation of burning.

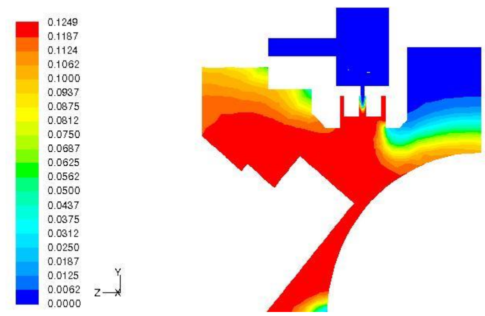


Fig. 32: ***Mass fraction of water vapor***

Check of burning – maximum value after burning of whole mass of fuel, the reached maximum corresponds to the equation of burning, the same character as previous Figure.

From received fields follows that for check of combustion is sufficient to observe the concentration of fuel (methane), of oxidizer (oxygen) and of one component of flue gases (water vapor or carbon dioxide).

Without insertion of treated material is generally visible the undesirable cross flow in model, instead the desirable flow direction in the lower edge of the area.

VI. SUMMARY AND CONCLUSION

From given parameters the efficiency of actual device is very low, of 16% approx. Some of presented proposals could increase the efficiency and reliability of described process and the product quality, too.

A. Numerical simulation

This feasibility study presents results of simulated solutions, with some conclusions for practical use. The advantage of

ISSN: 2456-1851

Volume:02, Issue:06

numerical simulations is that they are able relative simply, without demanding experiments to determine actual state and to verify influence of proposed modifications and of fully new ideas, too.

Generally said, any simulation approximates the reality, only, therefore the main formulated conclusions should be verify by experiment.

Solved problem has complicated geometry in general, with high number of small holes etc. So some important simplification is used and results of such simplified solutions are used for qualified evaluation for the complex solution. For instance:

Used symmetrical one-half model shortens the solution time.

Next problem is the meshing of small holes, first in models with heat conduction in solids, where the use of tetrahedral mesh elements makes problem – somewhere they arise unreal temperatures. Meshing of round solids by hexahedral elements is not possible.

Similar is the problem meshing around singular points (contacts of volumes etc.) – they must be substituted by adding of next small volumes – they allow to create mesh, but practically without influence on the result.

The simulation with combustion needs more of calculating time, compared with a case without combustion. For determination of principal features of the flow field, the "cold" models without combustion were used, "colored" by higher temperature of inlet flow. Next prolongation of each calculations means the implementation of radiation, which should be taken into account at high temperatures of flames etc.

B. Combustion

Air excess – the mixture of gas fuel and air is coming in the burner with small excess of air. Both items must be controlled, because the heat output of the burner is usually controlled in the range of 3:1 and more, depending on the mode of operation. So it is necessary to measure not only volume flows of each item, but also their pressure and temperature, to be sure, that the correct mixture in any moment is reached and so the quality of both combustion followed by product heating will be the highest. The suitable procedures and measuring devices are available in any handbook, for instance [4].

Theoretical optimum is the combustion at the stoichiometric ratio of air excess of $\lambda = 1$, when the highest temperature of flue gases is reached. However, from the reason of operational reliability, the air excess should be slightly higher, to be sure that the volume of combusting air is sufficient. At the air deficiency, the combustion is not perfect, the temperature of flue gases is lower, and the treated material is damaged by soot. At high air excess the uselessly high volume of air is heated (the air contains nearly of 80% of nitrogen, which does not take a part in the

ISSN: 2456-1851

Volume:02, Issue:06

combusting process), so the both flue gases temperature and efficiency are lower.

Burner distance from the heated material must be optimal; approximately the tops of flames are in the area of treated material. If the distance is too high, the heating effect is lower. If the burner is too close, the flame is constricted by adjoining solids, does not receive its right air volume and the combustion is not perfect.

The disadvantage of the observed solution is that the burner is situated up and treated material down, while the hot flue gases are moving up.

The inlet of secondary air into burning mixture is not necessary, due to used excess of air in the inlet mixture. Therefore, obstacles in the area of flames could not make a problem. The proposed side partition could not make a problem, because they reduce the cross flows of surrounding air in the heated area and more, affect as reflecting surfaces for partial reflection of radiation heat back into heated area.

Heating of inlet pipe for mixture fuel and air mixture is disputable. The preheated mixture decreases the heat necessary for reaching of combustion temperature, but in the same time, the mixture volume in inlet is increasing, so the flow resistance is increasing, the mass flow is decreasing and the heat input of burner is decreasing, too.

Use of oxygen instead air for creation of combustion mixture means that here is not any ballast nitrogen. Such mixture of ratio 2:1 approx. contains relatively more fuel, so the heating output of the same burner volume is higher.

C. Structural design

Simulation results of several structural designs give possible influence on the operation of the entire system, for instance:

Holes diameters – the precise manufacturing is necessary. For no-tolerated diameter of 0.8±0.05 mm the limiting values of the cross section, therefore of the volume flow, therefore of the heat output, too, are changing of ±12% of nominal value. Each manufacturing impurity on the hole edge decreases the flow substantially, too [5].

Complicated sieves, installed in individual channels decrease substantially the cross sections of combusting mixture, in average of 60-70%, but it is not included the substantial deformation of individual sieve eyes in sharp bends of the sieve surface. By this the flow resistance is increasing, the input of fuel mixture and heating output are decreasing. On the real channel width are situated two eyes of given sieve, only, therefore it is supposed to use flat sieve, only, across at the channel end. The effect of such sieve see above.

In the working area (lower wedge), where the treated material is moving, flows only small volume of hot flue gases. The lower outlet cross section is narrow and is partially clogged by treated material. In such typical

ISSN: 2456-1851

Volume:02, Issue:06

stagnation area, the flowing of flue gases is low and material here is not fully heated by convection. The main arriving part of flue gases divides here into two parts; the most of them escapes in the surroundings and heats the adjoining solids. On the left side the flue gases heat the channel of the fuel inlet, on the right side there is arriving the cold air from the surroundings. The proposal is to use side partitions or covers, which can partially to prevent it.

<u>Heat transfer by radiation</u> is typical for high temperatures of flue gases. Therefore, the proposed covers can be used as reflection surfaces of radiation heat back into the working area, too.

<u>High temperature of surfaces</u> can be suppressed by increased reflexivity of such surfaces. Actually used duralumin and brass oxidize quickly in the burner atmosphere and so their reflexivity decreases quickly, too. Stainless steel, chromium or nickel coating will be better, as presents for instance the commercial solution of burner, available in the catalogue [1] etc.

ACKNOWLEDGMENT

Our acknowledgment is given to VUTS Liberec – Center for Development in Machinery Research for the support in the framework of the grant NPU-LO1213 “National program of sustainability”, granted by Czech ministry for education, youth and sport.

REFERENCES

[1] Gas burners S. Reich – catalogue NEDFORM

[2] Software Fluent v. 6.3.26 etc.

[3] Adámek, K., “Permeability of textile layers (Prodyšnost textilních vrstev), Proc. of the XVI. int. conf. Applic. of experimental and numer. methods in fluid mechanics, TU in Žilina, 2008

[4] Handbook of measuring technics (Příručka měřicí techniky), SNTL Praha, 1967, and others

[5] Adámek, K., “Staffetendüsen hergestellt im Feingussverfahren”, Proc. of the 6. Weberei Kolloquium “Peripherien beeinflusst den Nutzeffekt“, ITV Denkendorf, 1990

Next reference are not mentioned here, because presented results were received by own simulations and measuring, using standard knowledge of both fluid and thermo mechanics, well known in professional community since many years ago. It is a pity that they are not fully used in industrial practice. Many different designs of burners are available on the web, but the aim of this paper is not to advertise it.

