International Journal of Engineering and Applied Sciences (IJEAS)
ISSN: 2394-3661, Volume-4, Issue-9, September 2017

Study on Heat Transfer Characteristic of Furnace Bottom for Waste Heat Recovery

Ruiyu Yang, Yanxia Wang

Abstract—**The temperature of Calcined Petroleum Coke particles is as high as 1000℃,which is of great significance in waste heat recovery applications. Water cooling type furnace bottom specified for Tank Calcination Furnace waste heat recovery system can significantly reduce heat loss and decrease thermal stress. In this paper heat transfer characteristic is investigated numerically and experimentally followed by analysis of temperature uniformity, numerical simulation results have good agreement with experimental data. Influence of parameters that velocity and inlet temperature on heat transfer characteristic are investigated and optimization design is obtained. The results are presented as follows: water cooling type furnace bottom significantly reduce heat loss and improve temperature uniformity, heat recovery proportion is up to 72.2% and average heat transfer coefficient ranges from 460~550 w·(m^2·℃)$^{-1}$; as inlet temperature increase heat transfer quantity decrease gradually and outlet temperature increases linearly with constant temperature increment; as fluid velocity increases heat transfer quantity increases and outlet temperature decrease linearly; as fluid velocity increases heat transfer coefficient and pressure drop increase; the optimized channel structure improves flow pattern and heat transfer is presented.**

Index Terms—**Waste heat recovery; Heat transfer characteristic; Temperature uniformity; Optimization design**

I. Introduction

With the increasing demands for energy and emerging energy crisis investigation of heat transfer technique and waste heat recovery attract more attentions in recent years. High efficiency utilization of energy is of great significance for sake of higher energy utilization proportion in industrial applications. Generally, waste heat resources can be classified by two methods: for heat resource medium, they can be classified into solid waste heat, fluid waste heat and gas waste heat; for temperature levels of waste heat medium, they can be classified into high temperature (higher than 600℃), intermediate temperature (between 200℃ and 600℃)and low temperature (below 200℃). Among those waste energy resources high temperature particle waste heat received great attentions for inherent utilization difficulties and potentials for heat recovery in engineering applications. Particle waste heat is by-product of large quantities of industrial products, such as carbon products, steel ore, slag and other particles in chemistry industry. Investigations on waste heat recovery of high temperature particles are carried out in wide range of engineering applications and great economic profits are obtained. High temperature particle heat recovery investigation focus on packed bed heat transfer mechanism and characteristic, heat exchanger performance and optimization.

Ruiyu Yang, Mechanic Department, Shandong University of Technology,Zibo,China,15898714209
Yanxia Wang, Mechanic Department, Shandong University of Technology,Zibo,China,13573307775

A. Background

Calcined Petroleum Coke is an important industrial material, which is produced by calcination at the temperature as high as 1300 ℃ and widely used in industrial electrodes. The Calcined Petroleum Coke is discharged into heat exchanger in cooling process at temperature of 1000℃, taking nearly one third of total energy consumed in whole process, so it is of great significance to recycle waste heat of particles so as to reduce energy loss. Tank Calcination Furnace has advantages such as low mass loss, high calcination quality and no need for surplus fuel supply, so it is widely selected as calcinations furnace in China. In discharging process Calcined Petroleum Coke must be cooled till the particle temperature below 200 ℃, and recycled waste heat can be used for making vapour. The furnace bottom burdens temperature as high as 1000℃, high thermal stress and excessive heat loss even wear occurs. Compared with previous furnace bottom the newly designed water cooling type one can significantly decrease heat loss, meanwhile improve temperature uniformity so as to reduce thermal stress, so detailed information of fluid flow and heat transfer needs to be investigated. A water channel of rectangle section is inserted in furnace bottom for sake of waste heat recovery by convective heat transfer of flowing water. Investigation on heat transfer characteristics are carried out numerically and experimentally so as to present fluid flow pattern, heat transfer and temperature uniformity in details.

B. Existed Investigation

The investigations on waste heat recovery consist of heat transfer models or mechanism, heat recovery performance, heat exchanger design for particle heat recovery and its optimization. High temperature particle heat transfer received significant attentions since it is basic problem of heat recovery but no systematic and generic conclusions are obtained. Packed bed, moving bed and fluidized bed are major application fields of packed particle heat transfer. The heat transfer characteristic and mechanism are complicated and uncertain due to multiple heat transfer modes exist in actual case, such as convective flow of gas, radiation and conduction between particles are involved. Experimental and numerical investigations are carried out for particle bed heat transfer, varieties of parameters are considered to analyse heat transfer performance of particle bed. Parameters of packed particle bed consist of particle diameter, porosity, thermal properties, flow velocity, inlet particle temperature and others.

From the point of design and engineering applications, it is essential and meaningful to investigate flow and heat transfer process in packed particle bed as well as heat recovery performance. Thorough understandings of heat transfer mechanism and models are developed in study of packed bed heat transfer. Numerical simulation can present flow and heat transfer in details but lack of accuracy, experimental results are reliable but no detailed information are shown.

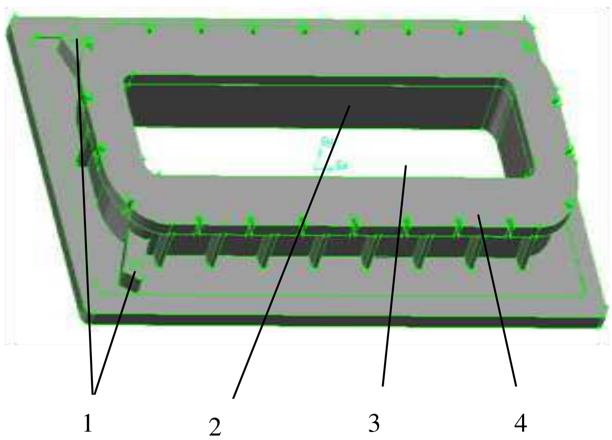
In the present paper, heat transfer characteristic of single furnace bottom is investigated numerically, heat recovery performance and uniformity of temperature distribution are presented experimentally to verify numerical simulation results, influence of factors on heat transfer and fluid flow are analyzed and optimization design is presented based on fluid flow pattern. The present work focus on influence of factors on heat transfer and fluid flow, heat recovery performance and temperature uniformity, the results have significances in engineering applications.

II. GEOMETRICAL FEATURE

A. *Physical Model*

Table 1: Material properties of each part

Material Property	**Furnace bottom**	**Refractor layer**	***Water***
Conductivity /W·(m· K)$^{-1}$	36	0.8	0.6
Density /kg·m^{-3}	7570	1560	1000
Specific Heat /J·(kg· ℃)$^{-1}$	470	876	4200



1. Inlet and outlet 2. Refractory layer 3. Discharge channel 4. Bottom side

Fig 1: Physical model of furnace bottom

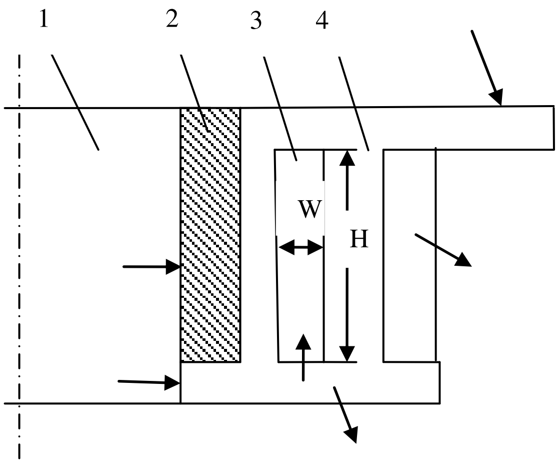
Geometry of furnace bottom is presented in Fig 1 and material properties of each part are given in Table 1. Three dimensional physical model is established in ANSYS Workbench software by parametric modeling method. The computational domain consists of furnace bottom, water cooling channel and refractory layer. Fluid region and two solid regions are established and then assembled, when model transformed into FLUENT software coupled walls are created automatically between different materials to guarantee heat transfer balance on interface of two bodies. The height of established model is 214mm, diameter of the water channel inlet is 42mm, thickness of refractory layer is 50mm, size of discharge channel is 1660mm×360mm, size of the rectangle water model, some simplification assumptions are presented as follows: (1) region of packed particle bed and silicone bricks on upper surface are eliminated;(3) fluid flow at inlet is of good uniformity and thermal properties of fluid are regarded as constant; (4) thermal resistance between different materials is neglected.

B. *Thermal Balance Analysis*

Compared with original furnace bottom, water channel of rectangle section is created in water cooling type and insulation materials are mounted on bottom side of it. Consequently, heat dissipation to environment is blocked, less heat flow away and most heat is captured by water. Fig 2 shows heat transfer pattern in furnace bottom, the arrows denote heat flow directions.

Calcined Petroleum Coke particles move at low velocity as 0.2m/h through discharge channel and heat transfer occurs on interface of particle and wall, it can be regarded as steady process. The heat input of furnace bottom is illustrated as follows: (1) heat flux conducted to upper surface of furnace bottom through upper silicone bricks, Q_1;(2) heat transferred through refractory layer into furnace bottom, Q_2;(3) particle heat conducted directly to inner surface of furnace bottom, Q_3 . The heat output is given as follows: (1)heat dissipation to surrounding air by natural convection, Q_4; (2) heat taken away by forced convective heat transfer on surface of water channel, Q_4;(3) heat loss from bottom surface, Q_6. The internal energy increments of furnace bottom resulting from temperature increase, Q_7. Based on above analysis, the thermal balance equation is given below:

$$Q_1+Q_2+Q_3=Q_4+Q_5+Q_6+Q_7 \tag{1}$$



1. Discharge channel 2. Refractory layer 3. Water channel 4. Furnace bottom

Fig 2: Schematic of heat transfer process

III. NUMERICAL SIMULATION

A. *Governing Equations & Boundary Conditions*

In the present work, water flow at inlet is considered as uniform flow, the gravity effect and viscous dissipation are neglected. The governing equations for continuity, momentum and energy are given in the following forms:

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{2}$$

Momentum balance equation:

$$\frac{\partial(\rho u)}{\partial t}+div(\rho uU)=div(\mu gradu)-\frac{\partial P}{\partial x}+S_u \tag{3}$$

$$\frac{\partial(\rho u)}{\partial t}+div(\rho uU)=div(\mu gradu)-\frac{\partial P}{\partial x}+S_u \qquad (4)$$

$$\frac{\partial(\rho w)}{\partial t}+div(\rho wU)=div(\mu gradw)-\frac{\partial P}{\partial z}+S_w \qquad (5)$$

Energy equation:

$$\frac{\partial(\rho T)}{\partial t}+div(\rho UT)=div(\frac{\lambda}{c_p}gradT)+S_T \qquad (6)$$

Conduction equation in solid region:

$$\rho_s c\frac{\partial T}{\partial t}=\frac{\partial}{\partial x}\left(\lambda\frac{\partial t}{\partial x}\right)+\frac{\partial}{\partial y}\left(\lambda\frac{\partial t}{\partial y}\right)+\frac{\partial}{\partial z}\left(\lambda\frac{\partial t}{\partial z}\right) \qquad (7)$$

Where ρ is the density of fluid, ρ_s is the density of solid, u, v, w is fluid velocity in the direction of x,y,z respectively, p is the pressure, t is the time, T is the temperature, grad is gradient of variation, S_T is the dissipation term caused by turbulent viscosity, λ is the thermal conductivity, c is the specific heat, c_p is specific heat, U is velocity vector, S_u, S_v, S_w is dissipation term in direction of x, y, z respectively.

In actual heat recovery cases, a couple of furnace bottom are placed by series connection. To simplify calculation and reduce calculation consumption, numerical simulation is carried out for single furnace bottom to obtain detailed information of heat transfer. The grid independence test is carried out to testify solution accuracy, it indicates that 1.8 million mesh elements satisfy the need of calculation. For simplicity of calculation transform the volume flow rate (1050L/h,1150L/h,1250L/h,1350L/h,1450L/h) into velocity (0.216m/s,0.231m/s,253m/s,0.267m/s,0.289m/s) respectively. The boundary conditions are listed as follows: at channel entrance the velocity is considered to be parallel to flow direction with a certain value and temperature is assumed to be constant; the exit is considered as pressure outlet; on inner surface of refractory layer the heat flux is constant(q=150 w/m^2); on inner surface of furnace bottom the temperature is consumed as constant(T_w=500℃), which is measured results in experiments; on upper surface of furnace bottom the heat flux is constant(q=100w/m^2); the bottom surface is considered as perfectly insulated; on the outer surface exposed to air the convection coefficient is given as constant(h=10 w·(m^2·℃)$^{-1}$).

B. *Numerical Simulation Setup*

Based on criterion of fluid flow state and the calculation of Reynolds number, the flow state is considered as turbulent flow. Computational Fluid Dynamic software FLUENT is used to solve governing equations, steady coupled solver is selected to handle equations using Finite Volume Method. The SIMPLE algorithm is used to cope with the coupling of pressure and velocity. The second- order upwind accurate scheme is used for discretization of the convective and diffusive terms. The convergence criterion for iterations is 10^{-6} for energy equations, for others it is 10^{-3}.

C. *Numerical Simulation Results & Disccusion*

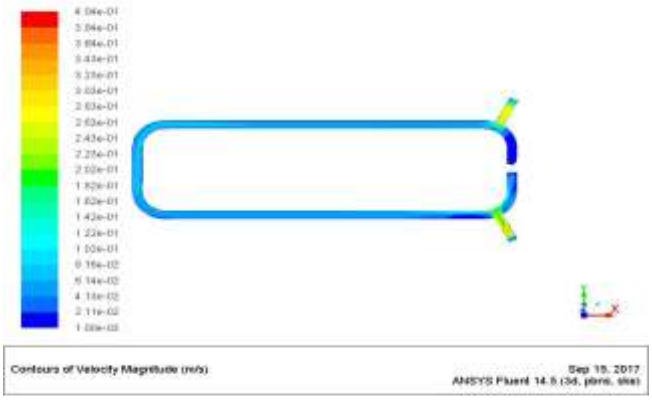


Fig 3(a): Fluid velocity contour on upper side

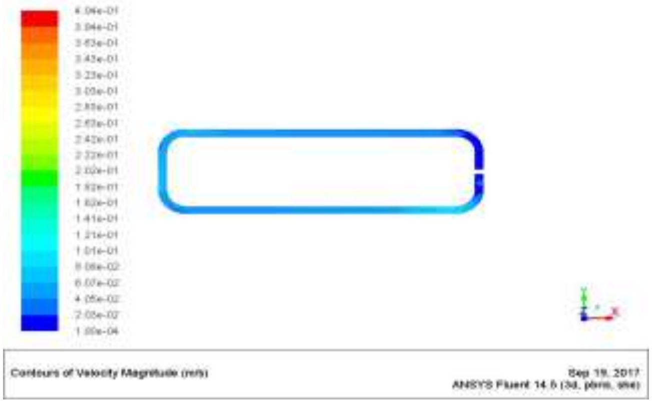


Fig 3(b): Fluid velocity contour on bottom side

Fig 3 presents fluid velocity contour in water channel, it shows that fluid velocity is relatively high near circular channel, swirl flow occurs at inlet of rectangle channel due to fluid flush on wall, velocity decrease along flow path. Secondary flow is observed at turning corner, which can effectively thick the flow boundary layer. From the comparison of velocity on different section, it is shown in fluid flow contour that flow pattern on upper side is better than bottom side since velocity is higher and turbulence stronger on upper side, flow stagnation occurs at the ending of rectangle channel. At exit of channel the flow velocity increase greatly causing pressure drop. The proposed channel optimization is that moving channel entrance to bottom side of channel and outlet to ending side of rectangle channel.

Fig 4 presents temperature distribution, it is shown that temperature ranges from 196℃ to 56℃ in most regions, maximum temperature difference is 140℃. it indicates that water channel can significantly reduce heat loss.

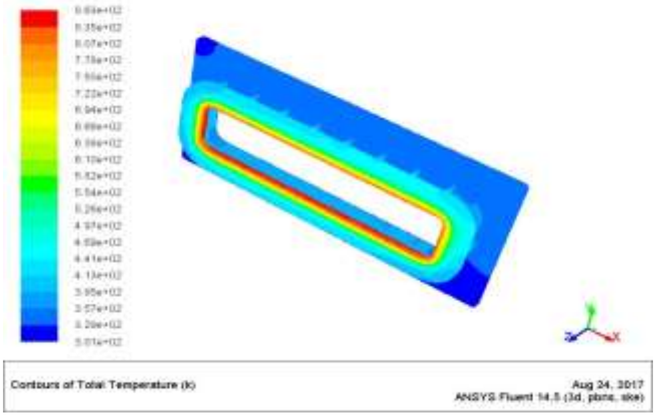
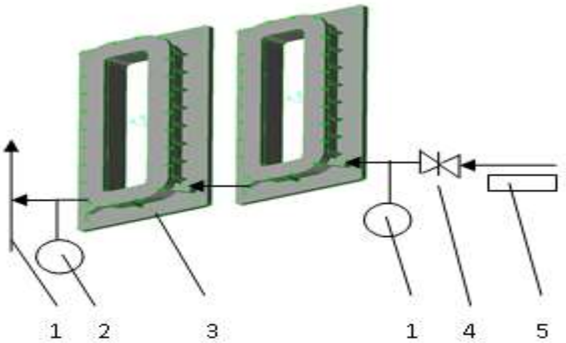


Fig 4: Temperature contour

IV. EXPERIMENT VERIFICATION

A. Experimental Items

To verify reliability and accuracy of numerical simulation, experiment is carried out. The experiment schematic is shown in Fig 5 and description is as follows: two furnace bottoms are connected together with stainless tube, valve is used to supply water flow at certain velocity measured by ultrasonic flowmeter, thermometer is used to measure outlet (inlet) temperature. Laser thermometer is used to measure temperature on surface and temperature distribution is shown in Fig 6.



1. Convergence tube 2. Thermometer 3. Furnace bottom 4. Valve 5. Ultrasonic flowmeter

Fig 5: Schematic of experimental apparatus

Heat transfer quantity is calculated by heat taken away by water in an hour

$$Q=cM(T_{out}-T_{in}) \tag{8}$$

where c is specific heat of water, M is mass flow rate of fluid, T_{out} is water outlet temperature, T_{in} is inlet temperature.

The focus of experiment is to investigate heat recovery performance variation with velocity and inlet temperature of fluid, followed by temperature uniformity analysis. The experiment is carried out under the condition that inlet water temperature of 46℃ and fluid velocity of 1250L/h, temperature is measured by laser thermometer. Heat recovery results obtained from experiments are presented in Fig9, 10 and comparison is done.

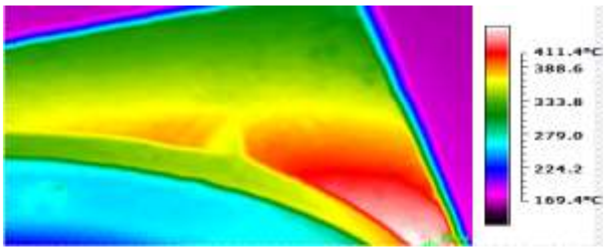


Fig 6(a): Temperature contour of original one

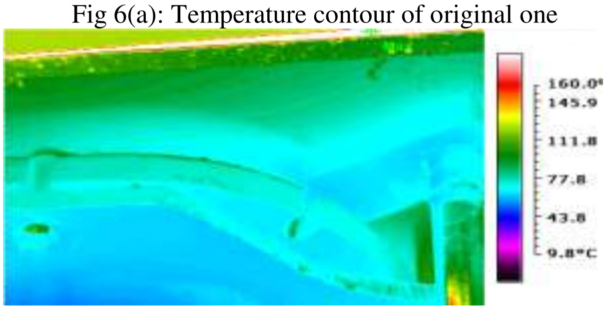


Fig 6 (b): Temperature contour of Water cooling type

It is shown in Fig 6 that for original one the highest temperature occurs at upper surface in large area and maximum temperature difference is 240℃ while for water cooling type the highest temperature occurs at upper side in small region and maximum temperature difference is 150℃, it has good agreement with numerical simulation results. The experimental results indicate that water cooling can decrease the temperature apparently.

Total heat recovery quantity obtained from test is 97812KJ/h, heat flux is 8.13kw/m^2, average convective heat transfer coefficient is caculated below:

$$h=Q/(A\Delta T)=27.17/(2\times1.67\times18)=452w\cdot(m^2\cdot℃)^{-1}$$

where h is average convective heat transfer coefficient, Q is total heat recovery quantity, A is area of interface of channel, ΔT is temperature increment of fluid.

Here heat recovery proportion η is defined as follows:

$$\eta=1-Q_n/Q_m \tag{9}$$

where Q_m is the heat loss from original furnace bottom, Q_n is the heat loss from water cooling type.

$$\eta=1-11549.7/41515.1=72.2\% \tag{10}$$

B. Discussion

Numerical simulation in present work is carried out for single furnace bottom, but in actual cases water flow through two furnace bottoms then into manifold. The following outlet fluid temperature and heat recovery quantity refer to two furnace bottoms, which corresponds with actual cases.

Influence of inlet temperature and fluid velocity on heat recovery is presented in Fig 9, 10 respectively, e denotes experiment, n denotes numerical simulation, Q denotes heat recovery quantity, T denotes outlet fluid temperature.

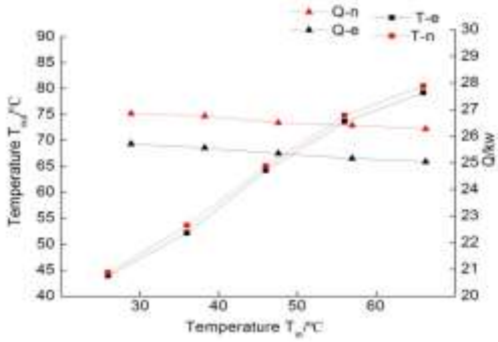


Fig 9: Heat recovery variation with inlet temperature

It is shown in Fig 9 that outlet temperature decreases and heat transfer quantity increases with the fluid inlet temperature linearly, thus heat recovery performance independent on inlet temperature. It can be seen from Fig 10 that outlet temperature decreases greatly and heat recovery quantity increases gradually with fluid velocity. Errors between numerical and experimental results are below 10%, so the numerical simulation results are reliable.

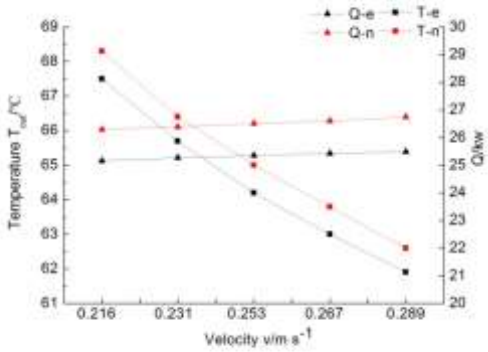


Fig 10: Heat recovery variation with fluid velocity

Detailed information of heat transfer characteristic is presented below, pressure drop and average convective heat transfer coefficient is analyzed and results are shown in Fig 11(a) (b) respectively, it indicates that average convection heat transfer coefficient and pressure drop increase with fluid velocity. The pressure drop ranges from 100 Pa to 180Pa and heat transfer coefficient ranges from 460~550 w·(m^2·℃)$^{-1}$.

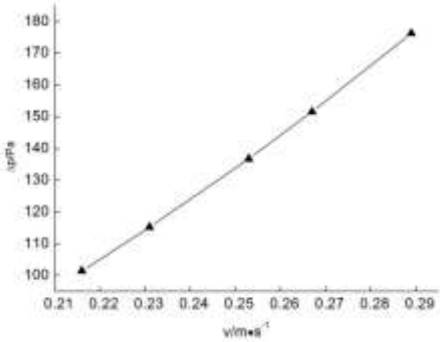


Fig 11(a): Δp variation with v

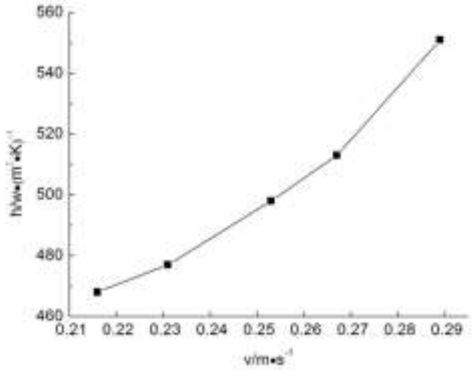


Fig 11(b): h variation with v

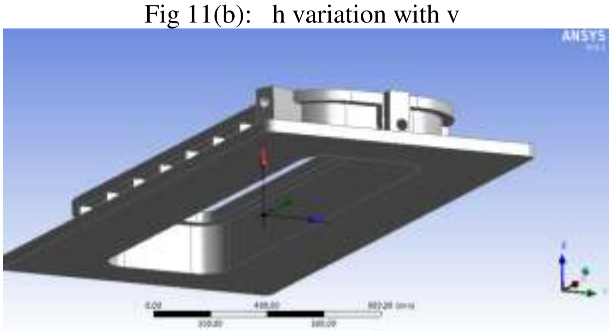


Fig 12: Optimized channel structure

To improve fluid flow pattern and heat transfer enhancement, channel structure is modified. The optimized channel is shown in Fig 12. Turbulence on bottom side is strengthened, flow stagnation is dismissed, and pressure drop decrease due to arrangement of entrance and outlet of channel.

V. Conclusion

This study presents detailed information of heat transfer and fluid flow pattern, heat transfer and temperature uniformity are analyzed. Numerical and experimental results show that fluid flow pattern is not good enough to enhance heat transfer, temperature uniformity and heat recovery performance is apparently improved.

Acknowledgment

The research presented in this paper was supported financially by Provincial Science Foundation of China through Grant No. ZR2013EEQ005

References

[1] Mojtaba Mokhtari, M. Barzegar Gerdroodbary, Numerical study of mixed convection heat transfer of various fin arrangements in a horizontal channel, Engineering Science and Technology, 20(2017):1106-1114

[2] I.K. Adegum, T.S. Jolayemi, (2017). Numerical simulation of forced convective heat transfer in inclined elliptic ducts with multiple internal longitudinal fins, Alexandria University, 12, pp. 1211-1225

[3] Zhigang Feng, Michaelides, (2009). Heat transfer in particle flows with Direct Numerical Simulaition, International Journal of Heat and Mass Transfer, 52, pp. 777-786

[4] Yongmin Zhang, Qing Wei, (2017). CFD simulation of bed-to-wall heat transfer in a gas-solids bubbling fluidized bed with an immersed vertical tube, Chemical Engineering and Processing, 6, pp. 17-28

[5] Qinyuan Yuan, Jing Ding, (2016). Heat transfer and energy storage performance of methane reforming with carbon dioxide in semi-cavity reactor, Procedia Engineering, 157, pp. 365-373

[6] Zineb Erregucragui, Nourreding Boutammache, (2016). Packed-bed thermal energy storage analysis: Quartzite and Palm oil performance, Energy procedia, 99, pp. 370-379

[7] Xiangdong Xu, Xingjuan Zhang, (2015). Study on heat transfer characteristic of compact heat exchanger based on experimental data, Procedia engineering,121, pp. 293-299

[8] Xin Yan, Yan Huang, (2017). Investigations into heat transfer and film cooling effect on a squealer-winglet blade tip, International Journal of Heat and Mass Transfer,115, pp. 955-978

[9] Anas Ei Maakoul, Azzeddine Laknizi, (2017). Numerical design and investigation of heat transfer enhancement and performance for an annulus with continuous helical baffles in a double-pipe heat exchanger, Energy Conversion and Management, 133, pp. 76-86

[10] Mahmoud A. Alzoubi, Agus P. Sasmito, (2017). Thermal performance optimization of a bayonet tube heat exchanger, Applied Thermal Engineering, 111, pp. 232-247

[11] Long Chen, Yidu Zhang, (2017). Heat transfer optimization and experimental validation of anti-icing component for helicopter rotor, Applied Thermal Engineering, 127, pp. 662-670

[12] Nianben Zheng, Peng Liu, (2017). Turbulent flow and heat transfer enhancement in a heat exchanger tube fitted with novel discrete inclining grooves, International Journal of Thermal Sciences, 111, pp. 289-300

Ruiyu Yang Graduated from Shandong University of Technology, acquire bachelor degree of engineering, majored in Thermal Energy and Power Engineering. Start from 2013 till now, engaged in investigation of high temperature particle waste heat utilization, skilled in numerical simulation of heat transfer and static analysis related with mechanic problems.

Yanxia Wang Graduated from Shandong University and got bachelor degree, majored in Internal Combustion Engine. Acquired master degree in China University of Agriculture in 2002, majored in mechanic, worked in Shandong University of Technology mechanic department, professor, taught theoretical mechanic, material mechanic, etc. Engaged in National '863' Program and Provincial Science Foundation, twenty papers were published in SCI and EI database.