Proceedings of the ASME 2012 Summer Heat Transfer Conference
IMECE2012
July 8-12, 2012, Puerto Rico, USA

HT2012-58225

NUMERICAL STUDY OF HEAT TRANSFER IN HEAT PIPES OF FLUE GAS WASTE HEAT RECOVERY

Qingqing Yong
College of Energy and Power Engineering
University of Shanghai for Science and Technology
Shanghai, China

Mo Yang*
College of Energy and Power Engineering
University of Shanghai for Science and Technology
Shanghai, China

Yuwen Zhang
Department of Mechanical and Aerospace Engineering
University of Missouri
Columbia, Missouri, USA

Mei Lu
College of Energy and Power Engineering
University of Shanghai for Science and Technology
Shanghai, China

ABSTRACT

In this paper, the advantages and precautions of using heat pipes to recycling flue gas waste heat in a power plant are discussed. A mathematical description of the model is established and its temperature and flow fields are simulated numerically. On this basis, effects of changing parameters and working conditions on the simulation results are studied.

INTRODUCTION

Energy issues are the most concerned topic in the world today. With the increasing tension of energy, scientists devoted to research of new energy development and power-saving emission reduction in existing equipment.

Power plant is energy hog. How to make more efficient use of energy in power plants, development and utilization of waste heat is undoubtedly a effective way [1]. On the premise of equipment safety and feasibility and in the field of researching flue gas residual heat recovery, using heat-pipe technology to recycle the heat is more and more popular in various power plants [2].

In theory this method is no doubt effective. In heat pipe heat exchanger, the heat transfer between cooling fluid and heating fluid, couples with working medium of evaporation and condensation processes of phase change in heat pipes, thereby the overall performance of heat pipe heat exchanger depends on the performance of heat pipe symbol itself and the fluid flow and heat transfer characteristics of shell and tube. These two aspects of integrated effect decide heat pipe heat exchanger of numerical simulation research has quite large of difficulty.

The heat transfer performance of heat pipe heat exchanger research has been general concerned in heat pipe territories scholars. Using traditional heat exchanger design theory, that is logarithmic average temperature difference method and effective heat transfer unit method on heat pipe heat exchanger for heat transfer calculation, has been reported largely [2~4], but using numerical analysis method to research heat transfer performance in heat pipe heat exchanger is uncommon reported. In reality, often because designers didn't research the temperature field and velocity field distribution clearly in heat pipe heat exchanger, just to satisfy a basic heat transfer area. In such condition, the designed heat pipe exchangers in actual operation have a variety of issues, such as hydrochloric acid [2], the tube vibration violent effects, and thermal deviation [6], etc.

This paper is based on the project of using heat pipe technology to recycle the Flue Gas Residual Heat in a power plant for background, using numerical simulation using

*Corresponding author(email: yangm66@gmail.com)

software-FLUENT under different conditions to get the temperature field and velocity field distribution of heat pipes heat exchanger. These provide a reference for research and engineering application.

Considering the feasibility of the project, the study is based on the solution of the steady 2D compressible Navier-Stokes equations by using finite volume method. The control equation is solved with SIMPLE algorithm. The grids are structured non-uniform staggered under Cartesian coordinate system; the turbulence model chooses the RNG k-ε model. The discrete phase model is random orbital model. The radiation model is P-1 model.

PHYSICAL MODEL

The working principle of heat pipe as shown in figure1, the heat pipe is generally compose by shell , capillary porous materials and working medium. The working media after being purified will be filled into tube with capillary wick material and being pumped into the high vacuum, then being sealed from the front end. One end of the heat pipe is evaporator section and the other is condenser section. When the evaporator section is heated, the working media of the capillary wick material absorb heat and be vaporized into steam, and the steam will flow to condenser section, be cooled in condenser section, release latent heat of vaporization of steam and be condensed into liquid. The liquid goes back to the evaporator section along the porous material under the action of capillary force, and then is reheated to evaporator into steam again. And this process is circulated like this again and again. The heat will be transmitted continuously from evaporator section of heat pipe to condenser section, as shown in Figs. 2 and 3.

THERMAL SYSTEM

The thermal power plant is composed of 15MW steam turbine-generator set and three 75t/h circulating fluidized bed boilers. The coal flue gas temperature is 140°C, about 80000 Nm3/h flow, coming out from the tail of economizer. After calculated , metal acid dew point is approximately 115℃。

Calculation formula： $t_{ld} = t_{sl} + \dfrac{\beta\times\left(k\times s^{n}\right)^{1/3}}{1.05^{a\times A^{n}}}$

condensation temperature Corresponding to the steam partial pressure: t_{ls}=60.68°C

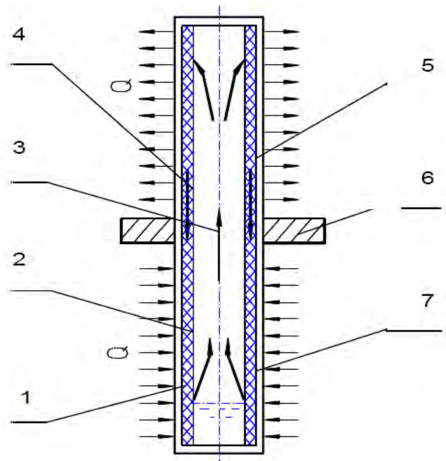
β=121；k=0.85；a=0.95

Receive base conversion of sulfur content: s^n=(1000 × 0.7)/5000=0.14;

Conversion of ash：A^n=(1000×19.82)/5000=3.964

Flue gas acid dew point calculated at 110°C [11]

So it can consider installing heat pipe heat exchanger to extracts energy, then 140℃flue gas reduced to 120°C. On the other end of heat pipes water from 50℃ can be heated to 90°C as boiler feed water. As shown in the Figs. 2 and 3 for heat pipe heat exchanger model. Because staggered tube bundle heat transfer performance is better than of bundles in line. This project uses a staggered arrangement design. The flue gas side model is shown in Fig. 4.



1- tube-shell, 2-wick, 3-vapour, 4-liquid, 5-condenser section, 6-clapboard, 7-evaporation section

Fig.1 The working principle of heat pipe

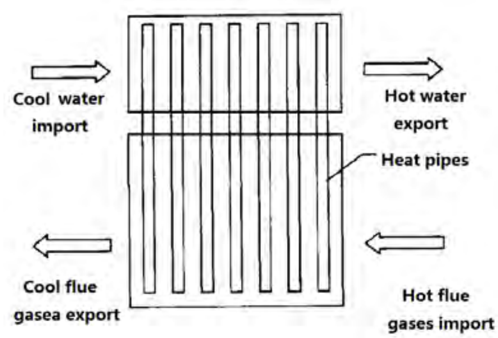


Fig. 2 Heat transfer model of heat pipe heat exchanger

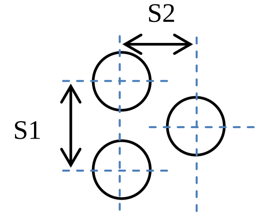


Fig. 3 Side model of heat pipe heat exchanger

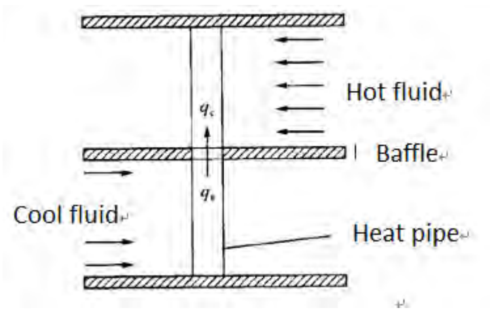


Fig. 4 Staggered tube bundles Model

The Process of thermal system flow diagram of reclaiming residual heat of the flue gas with heat pipe is showed as fig.5. The residual heat of flue gas is far from being recycled and utilized adequately. Using this low grade waste heat heating boiler feed water, pulled out of the system as a whole, it has reduced high grade steam from steam turbine to low- pressure-heater and high-pressure-heater for heating the boiler feed water. In other words, such improvement reduces the heat load of steam boiler, achieves the goal of saving energy and increases the enterprise economic benefits.

The structural parameter is shown in Table 1. The contrast of thermodynamic parameters among heat pipe evaporator and condenses is shown in Table 2.

Table 1 Structural parameter

Item	Unit	Date
Tube diameter	mm	Φ32
S1	mm	77
S2	mm	67
Evaporation segment length	mm	3400
Condensed segment length	mm	1400
Number of rows×columns	---	24

Table 2 Design parameters

side	item	unit	date
Flue gas side	Import temperature	K	413
	export temperature	K	393
	Quantity of flow	Nm^3/h	80000
	Resistance	Pa	487
Water side	Import temperature	K	323
	export temperature	K	363
	Quantity of flow	Nm^3/h	16.6
	heat recovery	kw	774

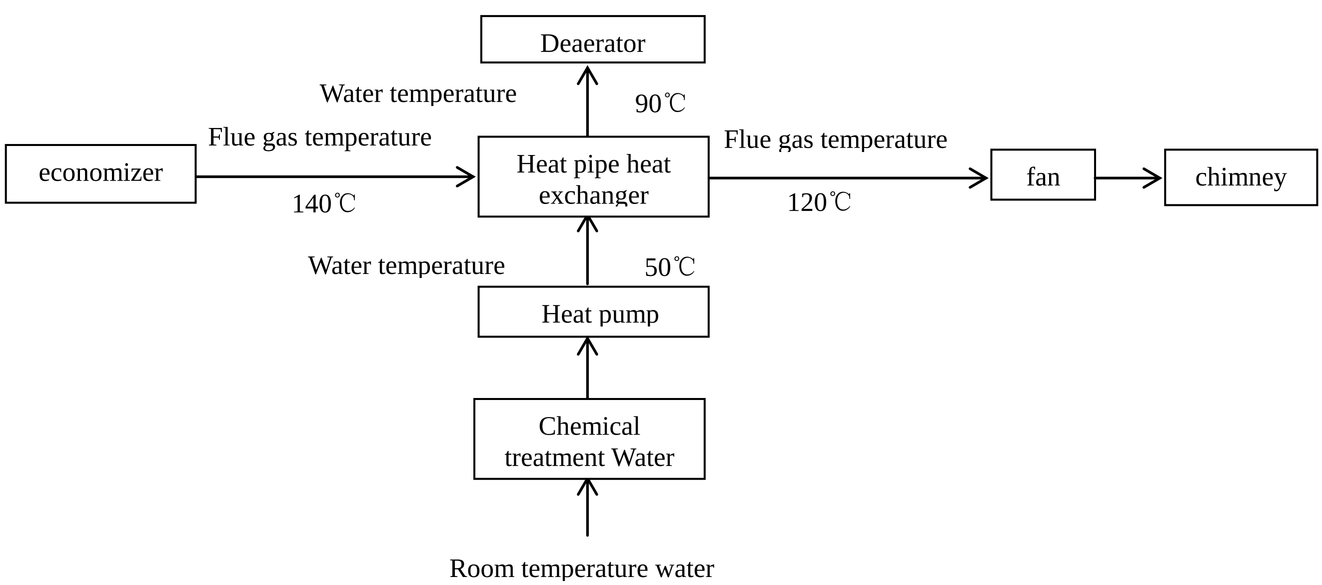


Fig. 5 Process flow diagram of the system of reclaiming residual heat of the flue gas with heat pipes

MATHEMATICAL MODEL AND DESCRIPTION

Problem Description

In this system, heat pipe heat transfer of flue gas model abstractions and water-side model abstractions as fluid cross-flow control model from the other side. Because the viscosity of water is larger than flue gas, and the designed velocity of the water is very small, while the other side heat exchange tubes Channel is smooth, in order to reduce losses, the other side is used as import, which greatly reduces the frictional and local resistance. The physical model of flue gas side is shown as fig.6 and model of water side is shown as fig.7.

Assume that the flow is periodically fully developed into the people, as two-dimensional, laminar flow, often sexual. Fluid and solid wall be as slip-free assumption. Flow channel geometry shows periodic variation on the main direction. When entrance is long enough, this periodic geometry of flow in a channel will get into the so-called periodically fully developed [9].

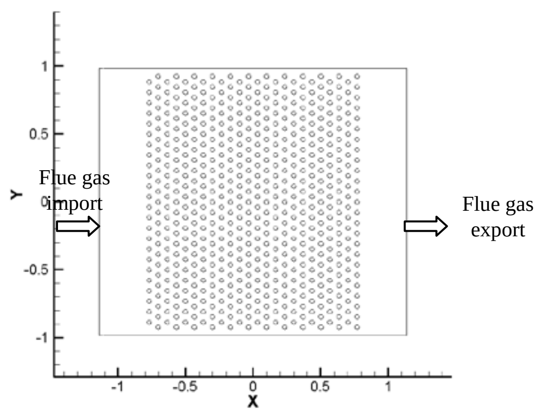


Fig. 6 physical model of flue gas side

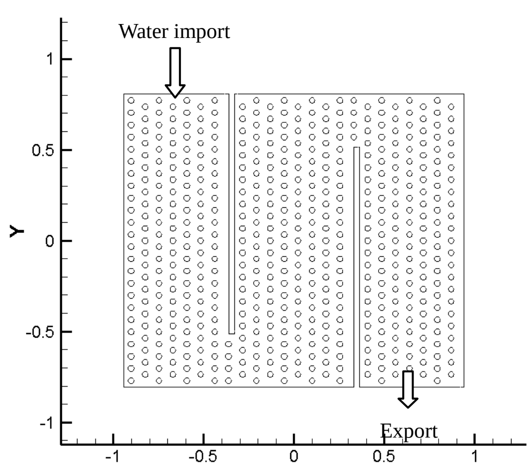


Fig. 7 physical model of water side

Governing Equations

Considering the feasibility of the project, the study is based on the solution of the steady 2D compressible Navier-Stokes equations by using finite volume method. The gas phase model is the RNGk-ε model. The discrete phase mode is random orbital model. The radiation model is P-1 radiation model, which is based on heat flux method. The control equation is solved with SIMPLE algorithm. The grids are structured non-uniform staggered under Cartesian coordinate system; the turbulence model chooses the RNG k-ε model. The discrete phase model is random orbital model. The radiation model is P-1 model. Those can be shown general equation, which are expressed as:

$$\frac{\partial}{\partial x}(\rho u\phi)+\frac{\partial}{\partial y}(\rho u\phi)+\frac{\partial}{\partial z}(\rho u\phi)=$$
$$\frac{\partial}{\partial x}(\Gamma_\phi\frac{\partial\phi}{\partial x})+\frac{\partial}{\partial y}(\Gamma_\phi\frac{\partial\phi}{\partial y})+\frac{\partial}{\partial z}(\Gamma_\phi\frac{\partial\phi}{\partial z})+S_\phi+S_{P\phi} \quad (1)$$

The equation of motion for the discrete phase mode can be described by:

$$\frac{\partial u_p}{\partial t}=F_D(u-u_p)+\frac{g_x(\rho_p-\rho)}{\rho_p}+F_x \quad (2)$$

The radiated transfer equation is written as:

$$\frac{dI_\lambda}{ds}=-(k_{a,\lambda}+k_{s,\lambda})I_\lambda+k_{a,\lambda}I_{b,\lambda}=\frac{k}{4\pi}\int_0^{4\pi}P(\theta)I_\lambda d\Omega \quad (3)$$

Boundary conditions

Import: given heat pipe heat exchanger heat fluid inlet temperature, velocity .

Flue gas side:

Temperature: 413K, 403K

Velocity: 4m/s, 6m/s, 8m/s.（Description: These levels of velocity Converted into the narrow section velocity respectively are：6.5 m/s ，10 m/s，13 m/s. They are up to the standard of heat exchanger design recommended velocity range [5,7,8]）

Heat flute from heat pipes to flue gas:-3981w/m^2

Water side:

Temperature: 323K

Velocity: 0.2m/s, 0.6m/s, 0.8m/s.

Constant temperature of heat pipes external wall: 373K

Export: export speed, pressure and temperature are determined by mass conservation. Shell walls: the use of impermeable, non-slip and adiabatic condition

Description: in the heat pipe exist complex phase change problems, so tube wall boundary conditions differ at the flue gas side and water-side

RESULTS AND DISCUSSION

Results and discussion of flue gas side

Physical model of flue gas side numerical simulation of temperature field, flow field and pressure can be abstracted as shown in Figs. 8-16

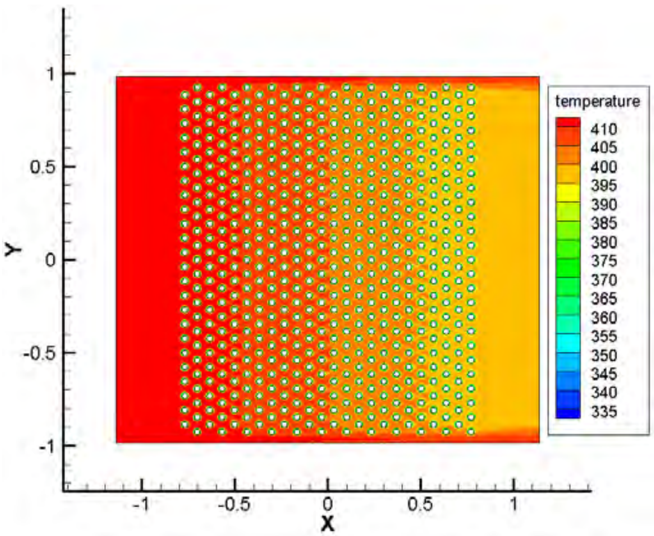


Fig. 8 Temperature field of 413K，4m/s

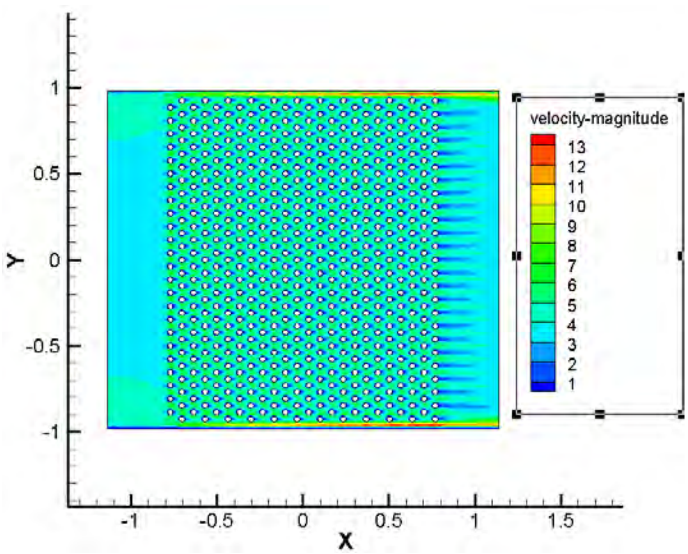


Fig. 9 Velocity field of 413K, 4m/s

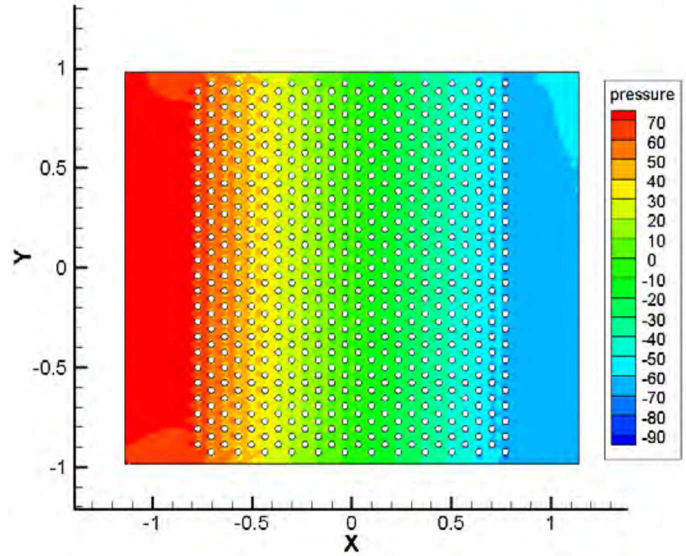


Fig. 10 Pressure field of 413K, 4m/s

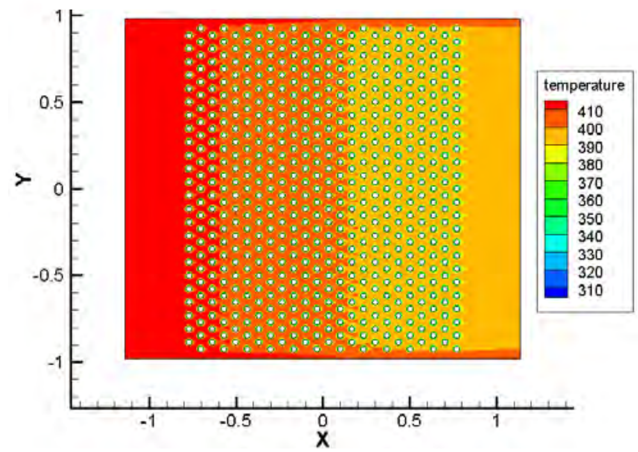


Fig. 11 Temperature field of 413K，6m/s

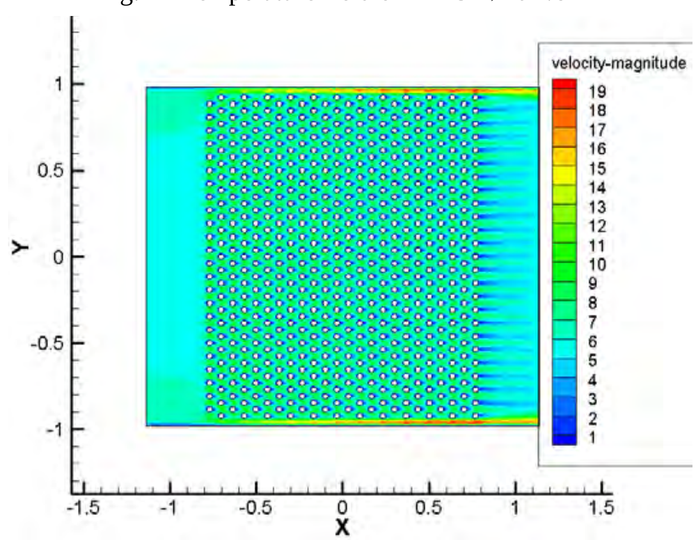


Fig. 12 Velocity field of 413K, 6m/s

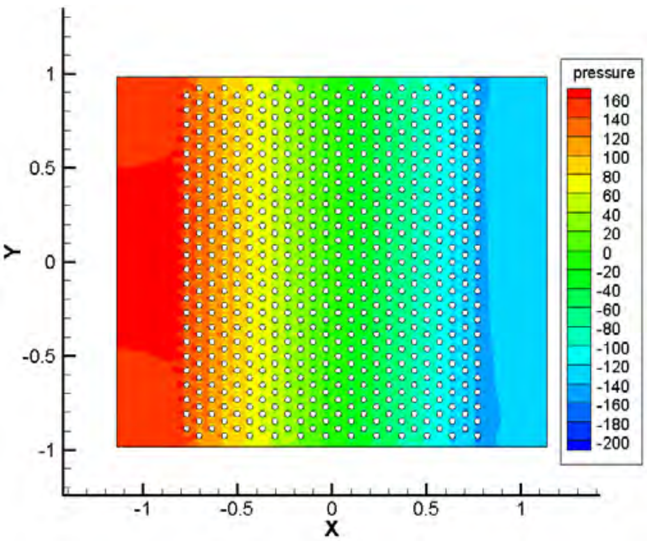


Fig. 13 Pressure field of 413K, 6m/s

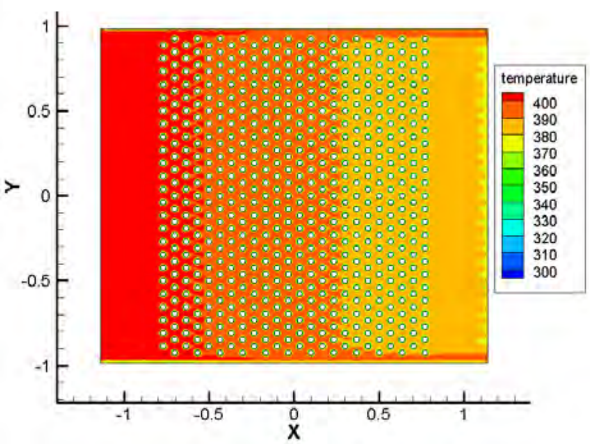


Fig. 14 Temperature field of 413K，8m/s

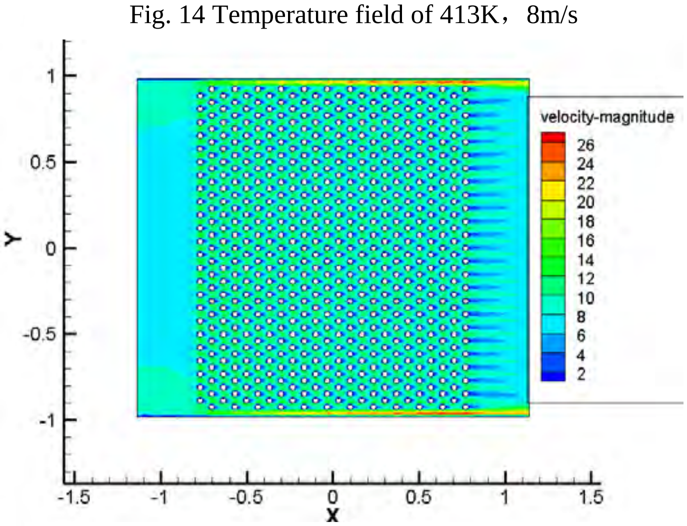


Fig. 15 Velocity field of 413K，8m/s

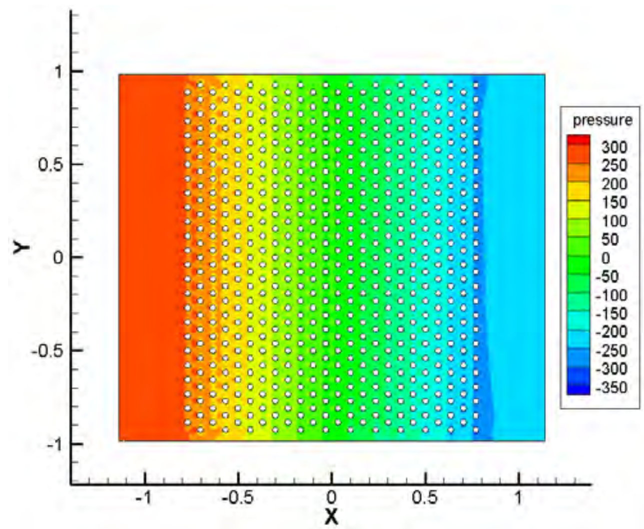


Fig. 16 Pressure field of 413K，8m/s

The import flue gas temperature is 413K, respectively to the velocity for 4m/s, 6m/s, 8m/s. numerical simulation of temperature field, velocity field and pressure field. From the distribution of temperature field can reach the following conclusions:1. These three conditions of the average temperature is 393k, the same as design values. 2, the strongest heat transfer at velocity of 8M/s. Export average flue gas temperature is 375k. This proves that Re is proportional to Nu.

$$Nu=0.358Cs.CzRe^{0.6}Pr^{0.33}.$$

But at this point is below the metal heat exchanger wall acid dew point. This temperature is not so desirable. 6m/s outlet and 4m/s outlet flue gas temperature almost unanimous, but we can see fourth heat exchanger pipe flue gas temperature becomes 405k at the speed of 6 m/s, and 12th pipe flue gas temperature becomes 405K at the speed of 6 m/s.so, 6 m/s for optimal velocity.

From Pressure field can gets these information: the pressure drop of 4 m/s is 140Pa; pressure drop of 6m/s is about 290Pa; pressure drop at 8 m/s velocity has been reached about 525Pa, at this case, spends big expense. It is clear that wind speeds greater pressure loss bigger. Judging from the velocity field distribution, Velocity distribution is steady and smooth. Tube speed hysteresis occurs in the last row. Comprehensive energy-saving point comparison, Velocity of 6m/s is the most optimum speed.

Results and discussion of water side

Physical model of flue gas side numerical simulation of temperature field, flow field and pressure can be abstracted as shown in Figs. 17-28.

The import water temperature is 323K, respectively to the velocity for 0.2m/s, 0.4m/s, 0.6m/s, 0.8m/s numerical simulation of temperature field, velocity field and pressure field. From the distribution of temperature field can achieve these following information:1.the export temperature are 368K at 0.2 m/s, 364K at 0.4 m/s, 362K at 0.6 m/s, 358K at 0.8 m/s. Heat transfer of 0.2 m/s maximum, but the possibility of liquid water phase change occurs. For this reason, design export water temperature values in 363k. In this way, 0.4 m/s velocity is the best value.

From the velocity field can be seen, 0.2 m/s and 0.4 m/s velocity distribution is smoother than velocity of 0.6 m/s and 0.8 m/s. From Pressure field can gets these information: Compared to the overall pressure loss than the flue gas are not significant. It is clear that velocity greater then pressure loss bigger. Pressure drop at 8 m/s velocity reached about 28pa, spends expense. Judging from the velocity field distribution, Velocity distribution is steady and smooth. Tube speed hysteresis occurs in the last row. Comprehensive energy-saving point comparison, velocity of 0.4m/s is the most optimum speed.

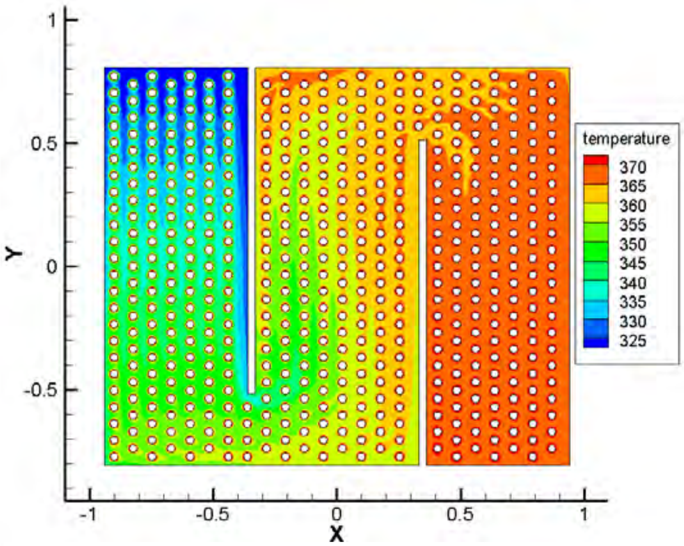


Fig. 17 Temperature field of 0.2m/s

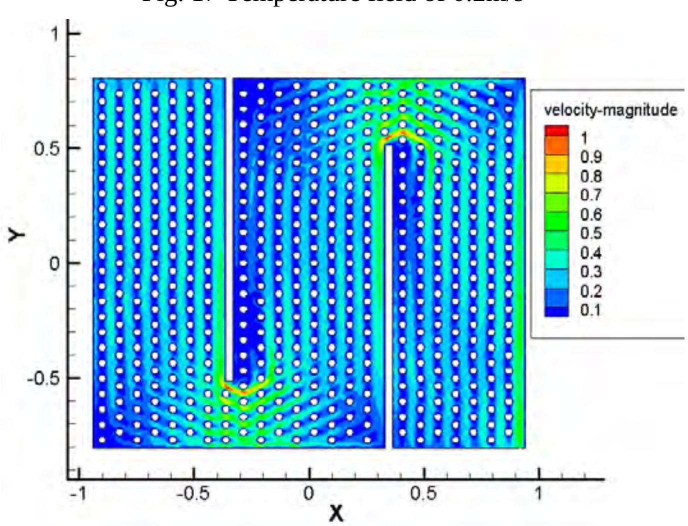


Fig. 18 Velocity field of 0.2m/s

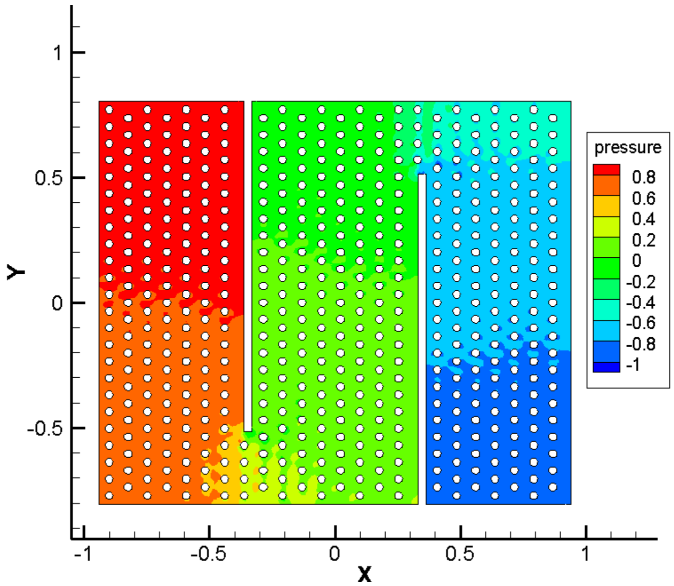


Fig. 19 Pressure field of 0.2m/s

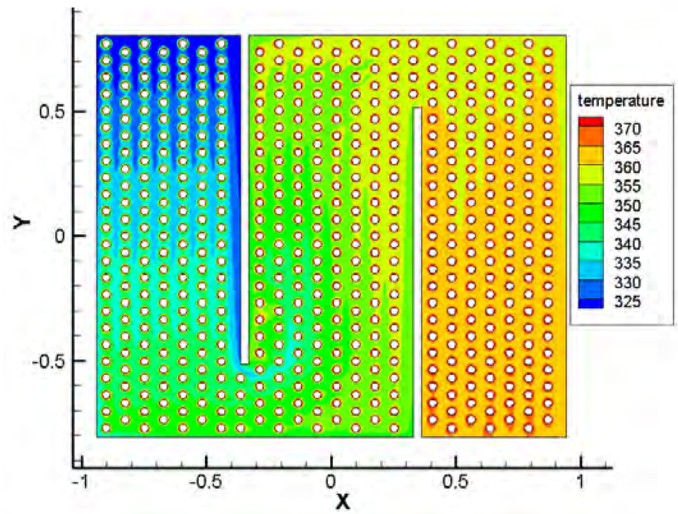


Fig. 20 Temperature field of 0.4m/s

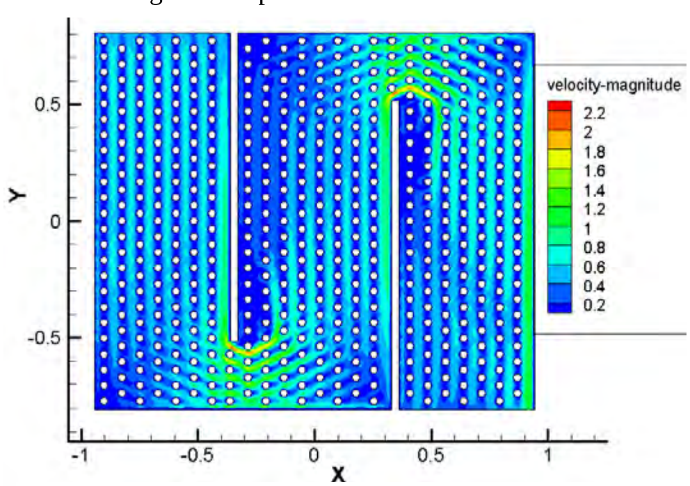


Fig. 21 Velocity field of 0.4m/s

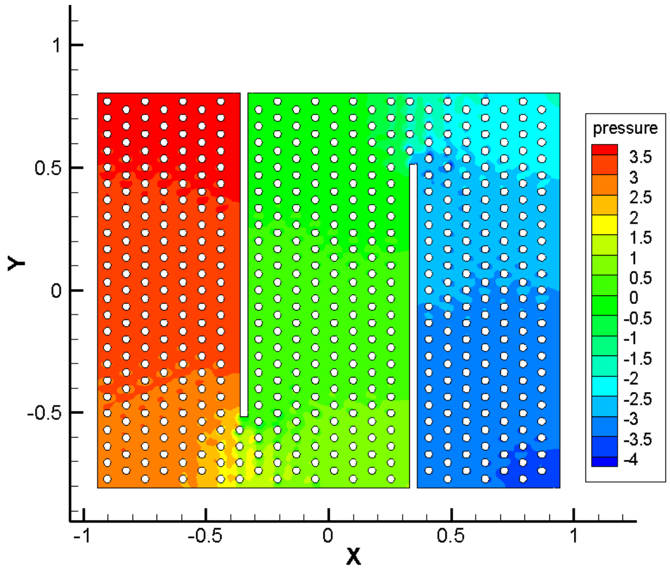


Fig. 22 Pressure field of 0.4m/s

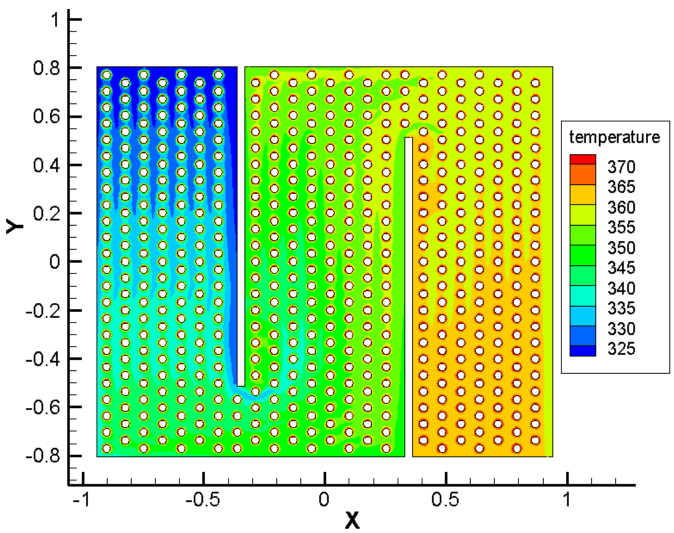


Fig. 23 Temperature field of 0.6m/s

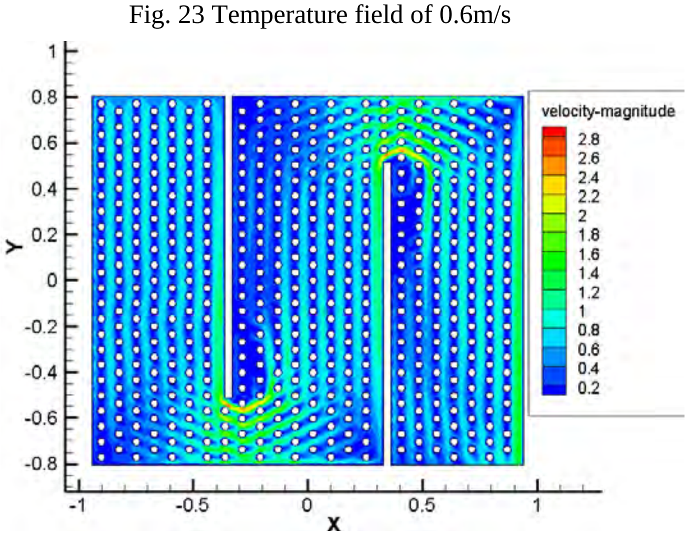


Fig. 24 Velocity field of 0.6m/s

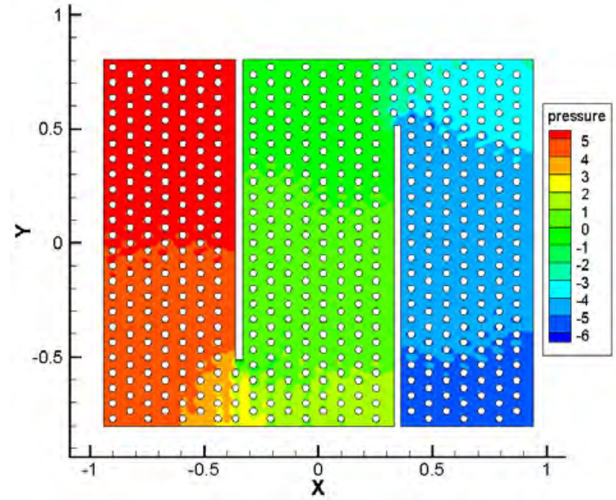


Fig. 25 Pressure field of 0.6m/s

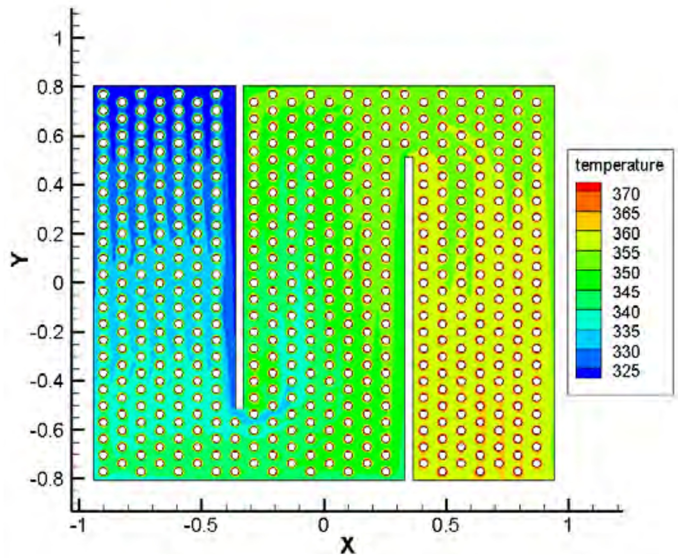


Fig. 26 Temperature field of 0.8m/s

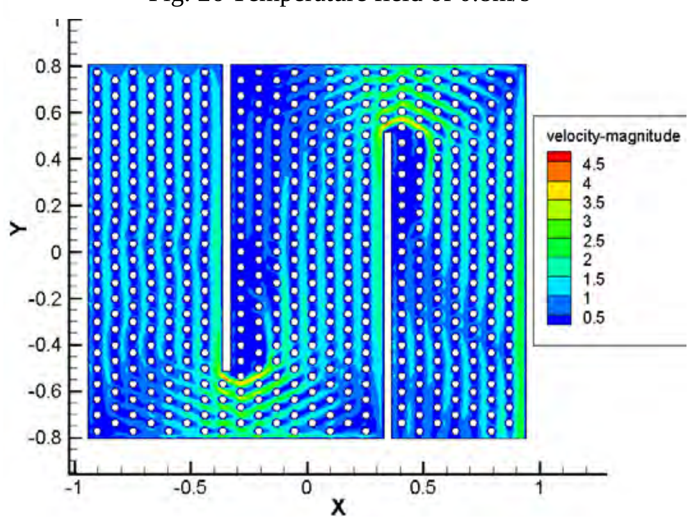


Fig. 27 Velocity field of 0.8m/s

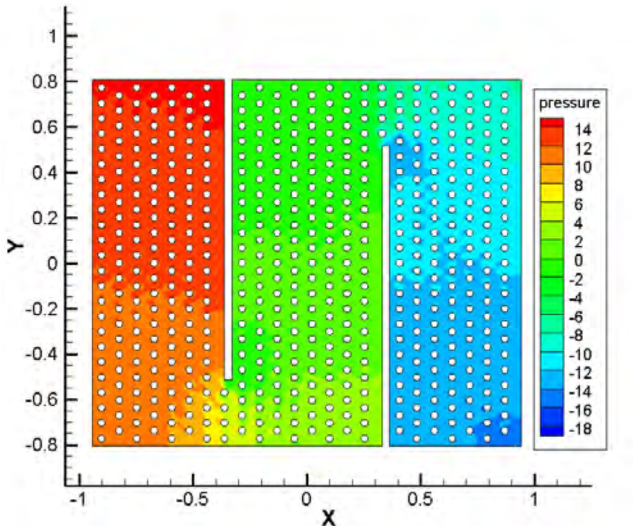


Fig. 28 Pressure field of 0.8m/s

CONCLUSIONS

This article uses numerical simulation methods to simulate temperature, velocity and pressure field in heat pipe heat exchanger in a power plant. After Comparison and analysis on the different working conditions, the best conditions are identified. Temperature field and velocity field and pressure velocity are obtained. Field distributions are provided for designersas a reference.

NOMENCLATURE

ϕ	Velocity- u, v, z , turbulent kinetic energy- κ , turbulent dissipation rate- ε , time-average mixture fraction- f , enthalpy- h .
Γ_{ϕ}	diffusion coefficient
S_{ϕ}	source terms caused by gas phase
$S_{p\phi}$	source terms caused by solid phase
U_p	velocity of solid phase
ρ	density of gas phase
ρ_p	density of solid phase
F_x	additional force
$F_D(u-u_p)$	traction force

ACKNOWLEDGMENTS

The present work is supported by the National Natural Science Foundation of China (No. 51076105) and the National Natural Science Foundation of China (No. 51129602)

REFERENCES

[1] Zhuang J., Zhan H., 2000, Heat Pipe Technology and Engineering Application, Beijing: Chemical industry press

[2] Zhang R., Xin D.D., Sun X.F., “Application of Heat Pipe on Decreasing Temperature of Exhaust Gas in Power Plant”, Power System Engineering, 2011

[3] Amode J O, Feldman K T·“Preliminary Analysis of Heat Pipe Heat Exchangers for Heat Recovery,” ASME Paper No·75-WA/HT-36.ASME: 1976

[4] Lee Y, Bedrossian A.“The Characteristics of Heat Exchangers Using Heat Pipes or Thermo siphons,” Int·J·Heat Transfer,1978,21(4):221—229

[5] Li T.H., and Hua C.S., Heat pipe design and application, Beijing: Chemical Industry Press, 1987

[6] Xiong M.Z., “Calculation of fluid-induced vibration in heat exchangers,” Chemical Industry and Engineering, 2007

[7] Faghri, M., and Rao, N., “Numerical computation of Flow and Heat Transfer in Finned and unfinned Tube Banks,” Int. J. Heat Mass Transfer, Vol. 30, No. 2, pp. 363-372, 1987

[8] Shi M.Z., and Wang Z.Z., principle and design of heat exchanger, Southeast University Press, 1996

[9] Patankar, S. V., Liu, C. H. and sparrow, E. M. , “Fully Developed Flow and Heat Transfer in Ducts Having Stream wise periodic Variations of Cross-Sectional Area,” ASME J. Heat Transfer, Vol. 99, pp. 180-186, 1977

[10] Yang M., Gao S.W., Jin H., Tao W. Q. “Cross-flow membrane periodically fully developed flow numerical simulation of tube bundle,” Journal of Northeast Dianli University, 14 (3): 38~43, 1994

[11] Gao, Z. Y., Song, W., Fan, L. J., Sun, X. Z., Chen, D. F., and Du, W. Y., 2009, “Numerical Simulation of NO Emission of a Dual Circle Tangential Firing Boiler of a 1000MW Ultra supercritical Unit,” Proceedings of the CSEE, **29**, pp.12-18.