SSRG International Journal of Civil Engineering
ISSN: 2348 – 8352 /doi:10.14445/23488352/IJCE-V7I9P102
Volume 7 Issue 9, 7-16, September 2020

Finite-time thermodynamic analysis and optimization of water-cooled multi-split heat pipe system (MSHPS)

Feihu Chen[#1,] Shuguang Liao*[2] Quan Zhang[#3]
[#1,]Civil Engineering College, Hunan University, Hunan, Changsha,410000, China
[2,]Changsha Maxxom high-Tech Co., Ltd. Hunan, Changsha,410008)

Received Date: 31 July 2020
Revised Date: 07 September 2020
Accepted Date: 13 September 2020

Abstract

This study focuses on the evaporation terminal's heat transfer characteristics and the cooling delivery unit (CDU). Flow distribution of the water-cooled multi-split heat pipe system (MSHPS) is used in the data center. The finite-time thermodynamic analysis and the exergy method is employed. Then the thermodynamic simulation software SIMULINK was hired to build the simulation model of the combined system. It shows that the period of working fluid passing through the heat exchanger is about 6 seconds. When the liquid to gas ratio is greater than 0.72, the heat transfer efficiency reduces greatly. The IT server is suggested to arrange at the location below 1.3 meters. And a CDU of 12 kW has a heat transfer of about 74W in 6 seconds. The optimum flow rate of CDU is 0.82 kg/s, and the corresponding heat transfer efficiency is 0.81. A certain cooling capacity distribution unit (CDU), which connects 2 heat pipe evaporator terminals of 10kW, was calculated, and the working fluid is R22. The error of the mass flow of the branch is about 6.7%. The Cop and exergy efficiency η of the MSHPS is 4.6 and 0.43 in Wuhan city (29° 5′ N and 115 ° 5′ E). The Cop and exergy efficiency η、using time and energy-saving potential in China's major cities were investigated and studied. The results are of great significance for the operational control and practical application of an MSHPS and other pipe-net systems

Keywords — *finite time thermodynamic analysis; exergy method, IDC room, heat pipe air-conditioning system, MATLAB/SIMULINK Software, refrigerant distribute characteristic*

I. INTRODUCTION

According to the great development of information technology, the information data center (IDC) room is built and expanded. To maintained the normal operation of the run, the air-conditioning system is needed. The air conditioning takes about more than 40% of the total energy consumption of the room[1]. The traditional air-cooled method can not fit the demand, so the water-cooled technique is employed to design the high-efficiency cooling process. But for the safety of the room, the water pipe is prohibited from going into the IDC room. Some are even not promised to the corridor of the room. On the other hand, the room's heat load has characteristics: there is only heat load in the room, the wet load is almost zero. And it needs refrigerating all year. The traditional air-conditioning system treats heat and wet load simultaneously, then to keep the humidity of the room, the reheat component is added after the air-conditioning process. It wastes a lot of energy. On the other hand, in the air distribution side, the traditional air conditioning system in the room, the cooled air is distributed by the windpipe. The distance is long, and there is a cooling capacity loss in the process. To tackle these problems, the heat pipe air-conditioner is developed. The heat pipe has high heat transfer efficiency. The heat pipe system tackled the heat exhaust by the calculator severely and combined the cold and heat channel closed technical. The PUE of the room can reach about 1.3[2].

Many scholars study energy performance. In the heat source and evaporator model: Yue. et al. developed a complete CFD model for the parallel tubes with simplified louvered fins of the evaporator structure was established to consider the thermal enhancement while reducing the computing costs. The CFD model was validated by comparing the cooling capacity and outlet temperature of MSCSHP with experiments[3]. Zou. et al. focused on the onsite test about the self-adaptive capacity of an MSHPS in a real data center under 25%, 50%, 75%, and 100% heating loads and various fan failures. The results show that the MSHPS abnormally operated under low heating loads, but it still met the cooling demands due to its superior self-adaptive capacity[4]. Ling et al. carried out experimental research on the MSHPS and pointed out that the system's optimal filling rate is between 33% and 42%[5]. Zeng. et al. mainly studies the heat transfer performance of the micro-channel backplane heat pipe air conditioning system and conducts heat transfer performance experiments in a standard enthalpy difference laboratory[6].

Furthermore, many scholars have done much research to check the pipe network characteristics, such as liquid separation parameters. A novel multi-branch heat pipe was investigated by the experiment. The proposed heat pipe consists of three branches—two with an evaporator and one with a condenser. The optimal working fluid filling ratio and ideal heat load were obtained[7]. Chen et al. used the thermodynamic simulation software cycle pad to simulate and calculate the thermodynamic properties and energy-saving characteristics of 72 backplane heat pipes connected by 6 CDU in a telecommunication room in Changde. The results show that the heat pipe's annual average energy saving is more than 26%[8]. However, the simulation method is steady-state. In the IDC room's energy efficiency measurement methods: In the influence of the environment aspect: A thermodynamic approach for evaluating energy performance (productivity) of information technology (IT) servers and data centers is presented. This approach is based on the first law efficiency to deliver energy performance metrics defined as the ratio of the useful work output (server utilization) to the total energy expended to support the corresponding computational work. These energy performance metrics will facilitate proper energy evaluation. It can be used as indicators to rank and classify IT systems and data centers regardless of their size, capacity, or physical location[9]. A distribution parameter annual energy consumption model during the effect of the dynamic heat dissipation characteristics of servers, lake water temperature, outdoor weather conditions, and cooling plant thermal performance will be established to evaluate the pthis cooling plant'sormance and energy efficiency of the different load factors[10].

The simulation model is hard to be built, and the task of the experiment is large. To build a simple method to evaluate the heat-pipe combined system, thermodynamic simulation software *SIMULINK* is employed. In this paper, the finite-time thermodynamic method is applied to establish an hourly simulation model of CDU, which can reflect the system's actual operation, such as the influence of inlet water temperature fluctuation on the heat transfer performance CDU. The system's network node model will be established based on the two-term flow model to solve the refrigerant flow distribution characteristic and the key parameters of the water-cooled multi-spilit heat pipe system used in the data center solved by MATLAB simulation method in this paper. Then the energy and exergy efficiency will be calculated and compared with the traditional air-conditional system.

II. THE PHYSICAL MODEL OF THE WATER COOLED MULTI-SPILIT HEAT PIPE SYSTEM

A. The system model of the water-cooled multi-spilit heat pipe system

A typical heat pipe air-conditioning system can be seen in Fig. 1. The IDC room area is 384 m^2, the indoor temperature demand is 24°C, and humility is 50%. 72 backboard heat pipe air conditioners were designed in the system. The outdoor temperature is 35.4°C. The temperature of the chiller water in and the outlet is 12/17°C. And the chilling water machine can be shut down when the outdoor temperature is below 10°C in winter, and the free cooling mode can be used[11].

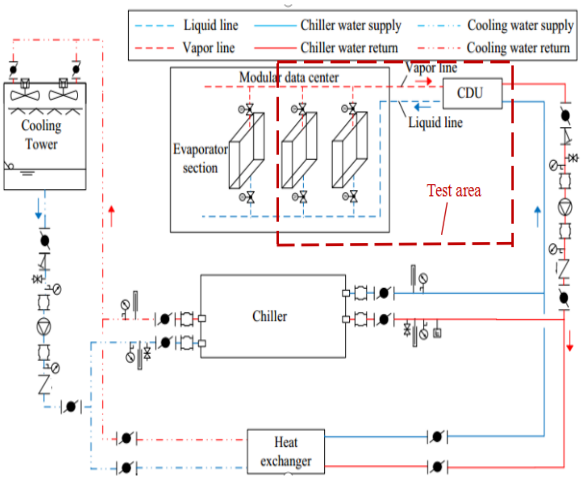


Fig. 1 The backboard heat pipe air-conditioning system in the *IDC* room

The average outdoor temperature of the room is shown in Tab. 1.

TABLE 1

The average outdoor temperature of the IDC room

Temperature gap	≥30	≥20, <30	≥10, <20	≥0, <10	<0
Temperature distribute efficiency	11.5 %	33.3 %	27.1%	26.2%	1.9%

B. The Physical model of heat pipe evaporator

The 6 kW heat pipe evaporator is used as the object. The microchannel heat exchanger is used. The height of the heat exchanger is 2 m, and the width is 0.6 m. The winding rate of the fan is 2000 m^3/s, and the refrigerant is R22.

C. The Physical model of CDU

The parameter of the CDU is shownn in table 2.

TABLE 2
The parameter of the CDU

Parameter	**Side of refrigerant**	**Waterside**
Mass flow (kg/h)	187.6	1714
Pressure drop (kPa)	2100	-
Working pressure(bar)	9.08	0.1
Temperature(℃)	20	14/19
Tracheal branch(mm)	19	
Liquid pipe branch(mm)	16	
The heat transfer area(m^3)	0.9	
CDU outline size (high) ×(wide) ×(thick)	0.525 ×0.111×0.12	
Installation height(m)	1.8	

D. The Physical model of the network

When the temperature difference is less than 6℃ inside and outside the room, the main refrigerating engine is opened, the heat of the supply of cold water and the hot pipe working medium is exchanged, and the heat emitted from the machine room is taken away. The heat is discharged into the outdoor environment by the condenser[12].

E. The Physical model of the cooling source

The compressor's cooling capacity is 130kW; the condenser temperate is 35℃; the temperate of the cold water is 12℃。

III. MATHEMATICAL MODEL OF THE HEAT-PIPE AIR-CONDITIONING SYSTEM

A. The mathematical model of the evaporator of the heat-pipe

Referring to the microchannel heat exchanger model, based on the new Gungor-Winterton correlation formula[13], the hourly simulation model of the heat pipe evaporator is established.

$$\dot{h}_{tp} = 0.0455(S+\dot{F})R_{e1}P_r^{-1}We_{lo}^{-0.17}\frac{\lambda_1}{D_h} \tag{1}$$

$$S = 1+4400BL^{0.86}Fr_{lo}^{-0.22} \tag{2}$$

$$\dot{F} = 1.12\left(\frac{\dot{x}}{1-\dot{x}}\right)^{0.86}\left(\frac{\rho_l}{\rho_g}\right)^{0.41} \tag{3}$$

The formula $\dot{h}_{tp}$: Hourly two current heat transfer coefficient; R_{e1}: Liquid Reynolds number; P_r : Liquid Prandt number; BL : Boiling number; We_{lo}: Liquid Weber number; λ_l : Liquid thermal conductivity; D_h : Heat pipe diameter; $\dot{x}$: Dry degree; ρ_l : Liquid density; ρ_g : Gaseous density.

The total heat exchange:

$$Q = m\cdot\int_0^1 \dot{h}_{tp} d_x \tag{4}$$

The efficiency of the system:

$$\eta = \frac{Q}{T} \tag{5}$$

The T in the formula is a cycle, that is, the heat transfer time of the working fluid in the heat pipe evaporator:

$$T = \frac{L_{ev}}{v} \tag{6}$$

Medium: V is the velocity of the working fluid:

$$v = \frac{m}{\pi\cdot(\frac{D}{2})^2} \tag{7}$$

Medium: m is mass flow.

The formula (1) - (7) constitutes a heat transfer model for heat pipe microchannel heat exchangers[14].

B. The mathematical model of the CDU

See in Fig. 2. The CDU's heat exchange is equal to the enthalpy drop from state point 7 to state point 8[15].

$$\dot{Q}_{cond} = \dot{H}_7 - \dot{H}_8 \tag{8}$$

In the formula $\dot{Q}_{cond}$ is the heat transfer for CDU by time. The $\dot{H}_7$ 、 $\dot{H}_8$ are enthalpy values for state points 7 and 8.

According to the designed evaporation and condensation temperature, the state of T-S diagram 2, 7, and 8 points can be determined.

The exergy loss of the CDU:

$$\dot{I}_{cond} = \dot{m}_{ref}\cdot\dot{i}_{cond} \tag{9}$$

And:

$$\dot{i}_{cond} = \dot{m}_{ref}\cdot(\dot{h}_7-\dot{h}_8)-\dot{T}_0\cdot(\dot{s}_7-\dot{s}_8) \tag{10}$$

The formula : the hourly loss of refrigerant for unit quality, the loss of working hours, and the mass flow rate of working medium. $\dot{T}_0$: The entropy of hourly ambient temperature and $\dot{s}_7$ 、 $\dot{s}_8$: working hourly entropy.

The formula (8) - (10) constitutes a finite time thermodynamic model of CDU, seen in Fig.2.

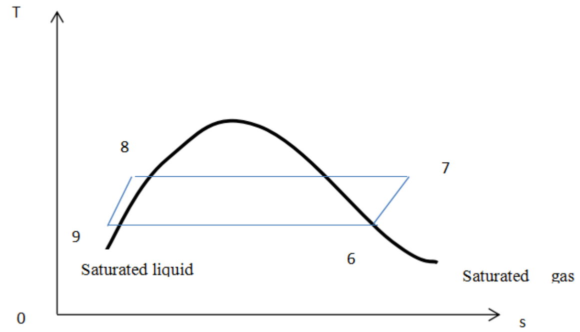


Fig. 2 The T-S diagram of the evaporator of the heat pipe

C. The mathematical model of a network system

According to the law of conservation of mass, the sum of the flow of each pressure node in the fluid network is 0:

$$\sum_{i=1}^{m} q_i = 0 \tag{11}$$

In the form, the flow of the first branch into the node.

Momentum equation based on fluid motion:

$$p_i - p_{i+1} = R_i \cdot q_i^2 \tag{12}$$

The resistance coefficient of the pipeline is the pipe inlet and outlet pressure, Pa.

It is linearized, and the computing equations of network nodes are obtained.

In the process of computing, the following assumptions are made:

(1) the branch has a fixed intercepting area.

(2) there is no heat exchange with the outside world;

(3) there is no compressibility in the working quality in the node.

(4) the diversion coefficient of the branch is variable. It is a function of the parameters of fluid pressure, flow rate, and valve opening.

Because the momentum equation contains a square term, the software can not be directly solved and linearized.

$$q_i = \frac{\sqrt{p_i - p_{i+1}}}{\sqrt{R_i}} \tag{13}$$

Calculation of boundary and local resistance:

$$p_i - p_{i+1} = (\lambda_i \frac{l_i}{d_i} + \varepsilon_i) \cdot V_i^2 \cdot \rho / 2 \tag{14}$$

Where λ_i is the resistance coefficient along the way; l_i is the length of I pipe; d_i is the diameter of I pipe, m ; ε_i is the local resistance coefficient of I pipe; V_i is the flow velocity of I pipe，m/s; ρ is the density of working medium, which is ρ_l in the liquid state and ρ_g is in the gas state.

Among them, A is a linear diversion coefficient:

$$\sqrt{\frac{\pi^2 \cdot d_i^4}{2 \cdot \rho \cdot (\lambda_i \frac{l_i}{d_i} + \varepsilon_i) \cdot (p_i - p_{i+1})}}$$

D. The mathematical model of the cooling source

a) The mathematical model of the compressor

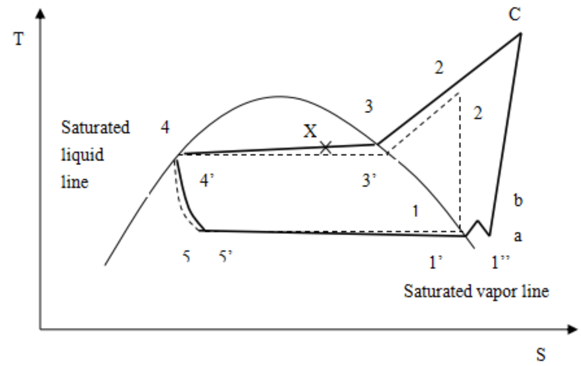


Fig. 3 The T-S diagram of the cooling source

According to Fig.3, the refrigerant's mass flow comes to the compressor at state 1. And the state of the outlet is state 3. Because of the irreversible process of the compression. There are exergy loss and entropy generations. The exergy balance equation is as follows[16]:

$$\dot{e}_1 + \dot{w} = \dot{e}_2 + \dot{i}_{comp} \tag{15}$$

The exergy input by the compressor is:

$$\dot{e}_{in,comp} = \dot{w} = \dot{m}_{ref}(\dot{h}_2 - \dot{h}_1) \tag{16}$$

The entropy generation of the irreversible process is:

$$\dot{s}_{gen,12} = \dot{s}_2 - \dot{s}_1 \tag{17}$$

Then the exergy loss of the compressor is:

$$\dot{I}_{comp} = \dot{e}_1 - \dot{e}_2 + \dot{w} = \dot{m}_{ref} T_0 (\dot{s}_2 - \dot{s}_1) \tag{18}$$

In the ideal reversible process of the compress, the power consumption is :

$$\dot{w}_c = \dot{m}_{ref} \cdot (\dot{h}_{2'} - \dot{h}_1) \tag{19}$$

The formula can define the parameter of the the real outlet state 3 of the compressor:

$$\dot{\eta}_{is,c} = \frac{\dot{h}_{2'} - \dot{h}_1}{\dot{h}_2 - \dot{h}_1} \tag{20}$$

The formula is the compress efficiency, and it is the decentralized compressor. Suppose it is 0.85.

$$\dot{h}_2 = \frac{\dot{h}_{2'} - \dot{h}_1}{\dot{\eta}} + \dot{h}_1 \tag{21}$$

b) The mathematical model of the condenser

The heat of condensation is equal to the enthalpy drop from the 2 states to the state point 4, seen in Fig.3.

$$\dot{Q}_{cond} = \dot{H}_2 - \dot{H}_4 \tag{22}$$

According to the T-S diagram, the state of the 4 points can be determined.

$$\dot{I}_{cond} = \dot{m}_{ref} \cdot \dot{i}_{cond} \tag{23}$$

And:

$$\dot{i}_{cond} = \dot{m}_{ref} \cdot [(\dot{h}_2 - \dot{h}_4) - \dot{T}_0 \cdot (\dot{s}_2 - \dot{s}_4)] \tag{24}$$

c) The mathematical model of the valve

As shown in Fig. 3, the throttling process line is 4~5. The exergy loss of the throttle valve is from the exergy balance equation and adiabatic throttling equation.

$$\dot{I}_{\exp} = \dot{T}_0 \cdot \dot{m}_{ref} (\dot{s}_5 - \dot{s}_4) \tag{25}$$

d) The mathematical model of an evaporator

In the evaporator, the circulating refrigerant absorbs heat from the low-temperature heat source to transform into a gaseous state and enter the compressor. For example, the process line 5~1 in Fig.3. When the cooling capacity is utilized, the exergy loss is estimated from the exergy balance equation [17].

$$\dot{I}_{ev} = \dot{m}_{ref} \cdot [(\dot{h}_5 - \dot{h}_1) - \dot{T}_0 \cdot (\dot{s}_5 - \dot{s}_1)] + \dot{m}_{ev,w} \cdot [(\dot{h}_{evwo} - \dot{h}_{evwi}) - \dot{T}_0 \cdot (\dot{s}_{evwo} - \dot{s}_{evwi})] \tag{26}$$

The formula m_{evw} is the flow of cold water, h_{evwi} 、 h_{evwo} is the enthalpy of cold water for inlet and outlet, and h_{evwi} 、 h_{evwo} is the entropy of cold water import and export.

E. The mathematical model of the energy efficiency

The COP and exergy efficiency η is :

$$COP = \frac{\int_0^t \dot{q}_{hw} \cdot dt}{\int_0^t \dot{w} \cdot dt} \tag{27}$$

$$\eta = \frac{\int_0^t \dot{e}_{hw} \cdot dt}{\int_0^t \dot{e}_w \cdot dt} \tag{28}$$

The actual climate parameters, such as temperature, are taken as a function of time. As an input parameter, the hourly temperature of the whole year is integrated in this period. The unit's average temperature can be obtained by dividing the total amount by the time of the whole year. As shown in the formula:

$$T = \frac{\int_0^t \dot{T} \cdot dt}{t} \tag{29}$$

IV. THE SIMULATION MODEL OF UNIT TYPE WATER COOLED MULTI CONTINUOUS HEAT PIPE SYSTEM

A. The simulation model of the evaporator of the heat pipe

According to formula 3, the formula of F is calculated. A simulation model for solving F is established. The constant module 1.12 (constant) is multiplied by X divided by 1 (Constant1) minus the 0.86 the power of X and multiplied by the 0.41 th power (Math Function1) [18]. Seen in Fig.4.

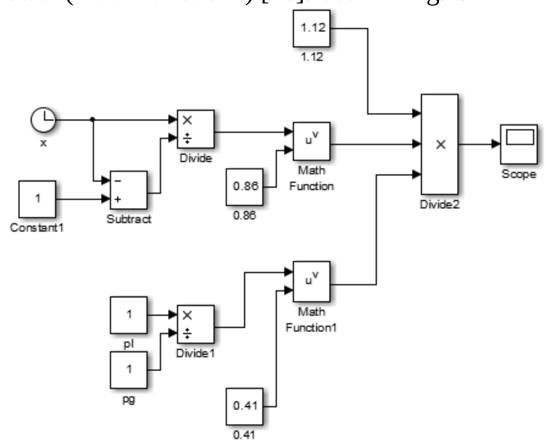


Fig.4 The simulation model of F value in the heat transfer coefficient of two-phase flow

Similarly, a simulation model of heat transfer coefficient and heat transfer can be established.

B. The simulation model of the CDU

According to the mathematical model of the heat exchanger, a simulation model is established. According to the equivalent heat transfer between the refrigerant and waterside, the refrigerant side is a phase change process. The heat transfer $\dot{Q}_t$ is the latent heat of phase change (multiplied by refrigerant mass flow rate $\dot{m}_{ref}$. The heat transfer at the waterside is the difference between the inlet and outlet temperature ($\dot{T}_{wo} - \dot{T}_{wi}$) times the specific heat (C_w) and the flow rate. As seen in Fig.5.

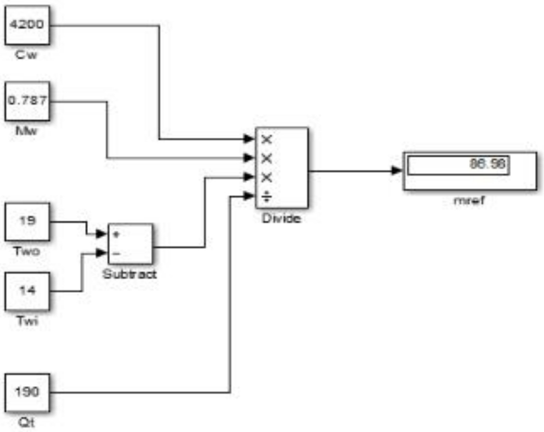


Fig.5 The simulation model for solving the flow of refrigerant in CDU

C. The simulation model of the compressor

According to the evaporating temperature, the entropy of the refrigerant at state 1 can be gotten. Then the entropy of the refrigerant at state 2 can be gotten; then, according to the Te\Tc and compressor efficiency η, the differ of the $(h_2 - h_1)$ can be gotten. Then put the $(h_2 - h_1)$ sublet the accumulate of the initial temperature T_0 multi $(s_2 - s_1)$, at last, let the calculation result multi the mass flow of the refrigerant m_{ref}. The exergy loss of the compressor can be gotten. See in Fig.6[19].

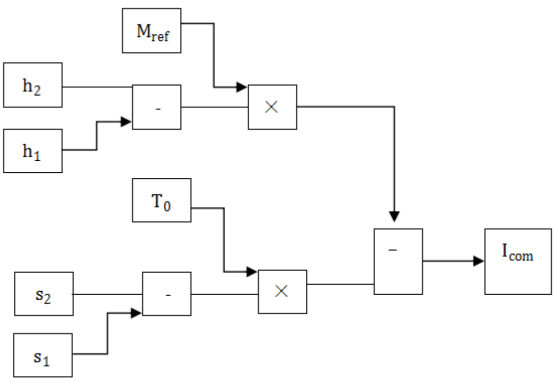


Fig.6 The simulation model of the compressor

D. The Simulation Model **of the** *condenser*

The condensing heat is equal to the enthalpy drop from state 3 to state 4.

According to the T-S diagram, the state at the outlet of the condenser can be defined.

According to the condensing temperature Tc, the exergy loss of the condenser can be gotten. Then the value of the simulation model $M_{ref} \cdot [(h_2 - h_4) - T_0 \cdot (s_2 - s_4)]$ can be calculated, see in Fig.7.

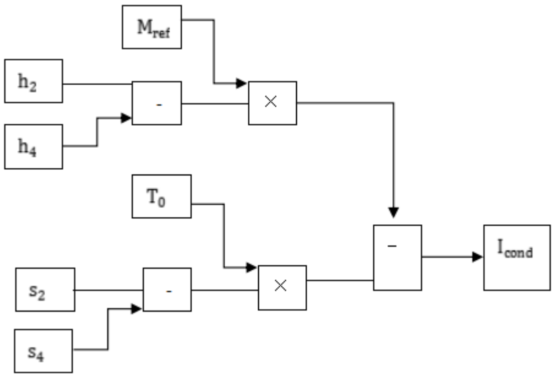


Fig.7 The simulation model of the condenser

E. The simulation model of an expansion valve

As seen in figure 1, the throttling process is 4-5. Due to the exergy balance equation and the adiabatic throttle process, the valve's exergy loss $T_0 \cdot m_{ref} \cdot (s_4 - s_5)$ is got.

According to the condensing temperature, the entropy at state 4,s_4 can be gotten. Then the entropy at state 5 (s_5) can be calculated by the evaporation temperature. Let s_4 sublet s_5, the results multi the base

point temperature T_0 and the mass flow of the refrigerant. The exergy loss of the expansion valve can be gotten. As seen in Fig.8.

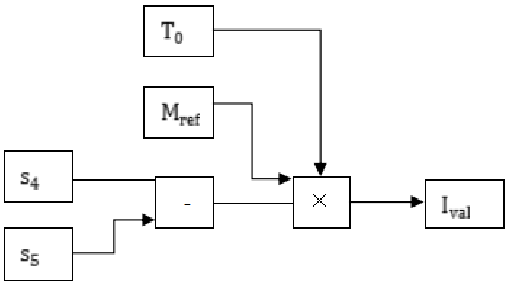


Fig.8 The simulation model of the valve

F. The simulation model of the evaporator

According to the inlet temperature and outlet, the inlet and outlet chilled water's enthalpy, and entropy can be calculated. Then put the environment temperature T0 multi the, then let the difference of the enthalpy of inlet and outlet sublet, at last. Still, the result of the chilled water's mass flow, the exergy loss of the chilled water side can be gotten. Similarly, the exergy loss at the refrigerant side can be gotten, then put the two add together, the exergy loss of the evaporator $m_{ev,w}[(h_{evo} - h_{evi}) - T_0(s_{evo} - s_{evi}) + m_{ref}[(h_5 - h_1) - T_0(s_5 - s_1)]$ is gotten. As seen in Fig .9.

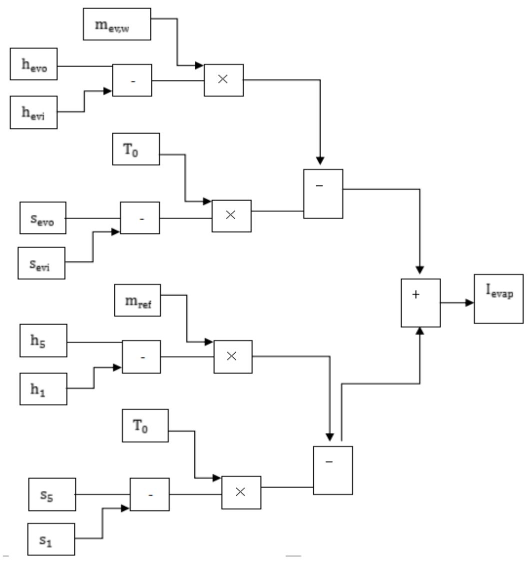


Fig.19 The simulation model of the evaporator

V. Experimental test

A. The purpose of the experiment

In the field, the server's heating capacity is different in time and space, and the distance from CDU is quite different. It is easy to cause uneven liquid distribution in the pipe network, resulting in dry burning and other problems. For this reason, the software for the design of the water-cooled multi-spilit heat pipe system (MSPHPS) is compiled. It is a reference for the design of system parameters. In order
to verify the accuracy of the software, it is necessary to test, verify, and modify the accuracy of the software until it meets the engineering accuracy requirements. Seen in Fig.10.

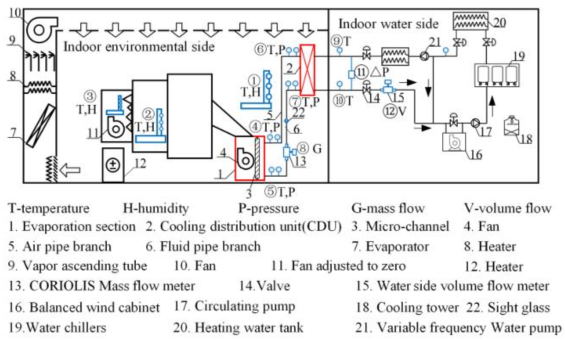


Fig. 10 Schematic diagram of the backplane heat pipe test platform of enthalpy difference laboratory

B. Test of construction of system liquid separation test bench

Two backplates of heat pipe air conditioner are set, and the model is 6kW. The structure is a copper tube and aluminum fin structure. The inlet air temperature of one block is 35 ℃, and that of the other block is reduced from 37.5 ℃ to 32.5 ℃. The wind speed is the same. The accuracy of the calculation results is verified by measuring the flow meter flow and total flow of the branch pipe. Seen in Fig.11.

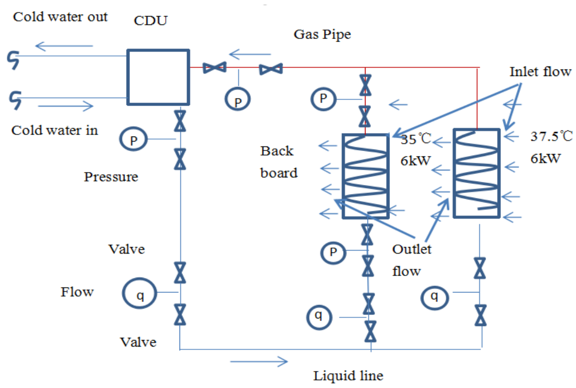


Fig.11 The flow and pressure test diagram

C. The experimental equipment

The inlet air temperature of backplate 1# is 35, ℃and the outlet air temperature is 21℃. The refrigerant is R22. The phase transition temperature of the designed refrigerant is 19 ℃. The corresponding pressure is 885.7 Pa. The inlet air temperature of backboard 2# increased from 32.5℃to 37.5℃, and the outlet air temperature was 21 ℃. Seen tin Tab.3.

Tab. 3 The style of the experimental equipment

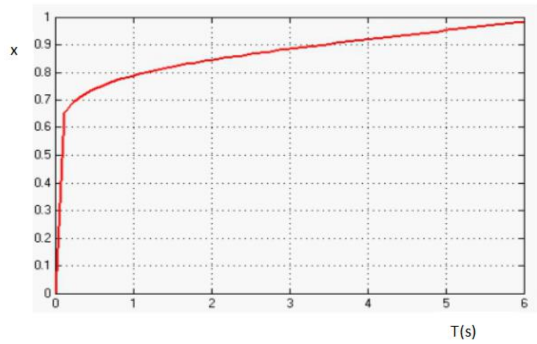
Name	Model	Measurement range	Resolution	Remarks
Flowmeter	OMEGA	0.001-9.999m/s	1%	Doppler flow meter
thermometer	NTC-3M10K	-20℃ - 150℃	0.05℃	
Pressure gauge	SL1808T	-0.1-250MPa	grade0.4	

D. Initial parameter setting

The initial calculation values are shown in Table 1: the inlet air temperature of backplane 1 is 35 ℃, and that of backplane 2 is 37.5 ℃. Seen in Tab.4.

VI. EXPERIMENTAL TEST

It can be seen from the relationship between the liquid to gas ratio of the heat exchanger and the height and operation time of the heat exchanger. It shows that the liquid to gas ratio is higher than 0.72; the heat transfer efficiency reduces greatly. When arranging the IT server, it is advisable to concentrate on the location below 1.3 meters, seen in Fig.12.



Test conditions: the inlet air temperature of backplane 1# is 35 ℃, and that of backplane 2#increases from 32.5 ℃ to 37.5 ℃.The outlet air temperature is 21 ℃.

Fig. 12 The liquid to the gas ratio (x) vs. the operating time in the evaporator

The calculation results show that heat transfer gradually slows down with time. CDU has a heat transfer of about 74J after 6 seconds of heat transfer. Seen in Fig.13.

TABLE 4 The parameter setting

Take the meteorological conditions of a place (such as Wuhan city) as an example as input.

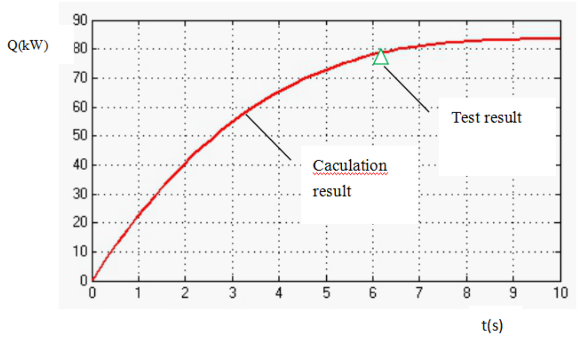


Fig. 13 The total heat transfer vs. the heat transfer time of the heat pipe exchanger

The heat transfer efficiency of CDU is calculated by changing the flow rate of cold water. As shown in the diagram, the optimum flow rate of CDU is 0.82, and the corresponding heat transfer efficiency is 0.81. Seen in Fig.14.

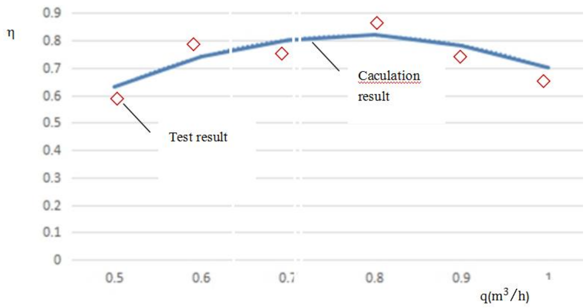


Fig. 14 The η of the evaporator vs. the mass flow of the cooling water

The flow calculation and test values of each branch are as follows. Seen in Fig.15.

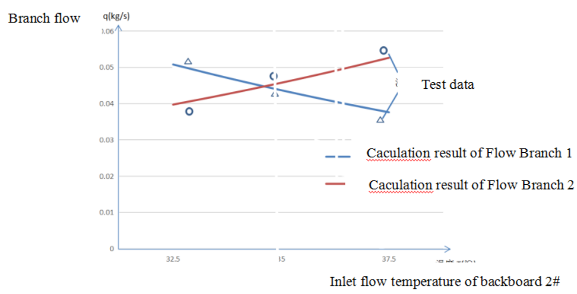


Fig. 15 The flow calculation and test values of each branch

The Cop and exergy efficiency of the system are shown in Fig. 16.

Serial number	Parameter	Value
1	Inlet air temperature of backplate 1 (℃)	35
2	Inlet air temperature of backboard 2 (℃)	37.5
3	Total mass flow rate of refrigerant (kg / s)	0.0901
4	Flow of branch 1 (kg / s)	0.0371
5	Air volume of branch 1 (kg / s)	0.4983
6	The diameter of branch 1 (CM)	15
7	Inlet pressure of branch 1 (kPa)	885.7
8	the outlet pressure of branch 1 (kPa)	887
9	Flow of branch 2 (kg / s)	0.0530
10	Air volume of branch 2 (kg / s)	0.6040
11	The diameter of branch 2 (CM)	15
12	Cold water inlet temperature (℃)	12
13	Cold water outlet temperature (℃)	17
14	Cold water flow (kg / s)	0.8124

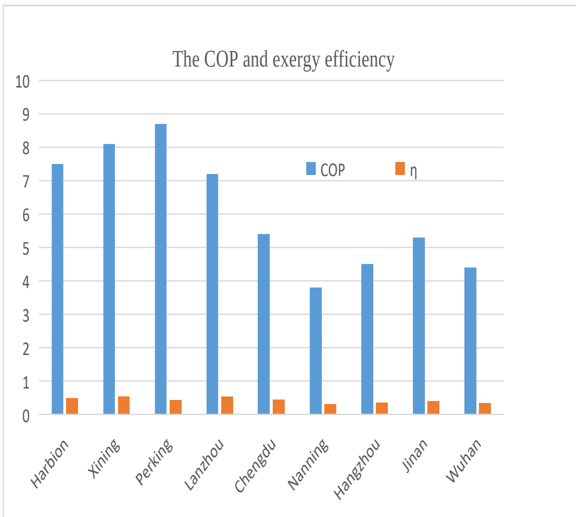


Fig. 16 The Cop and exergy efficiency of the system

The energy-saving potential in the major city of China is seen in Fig.17.

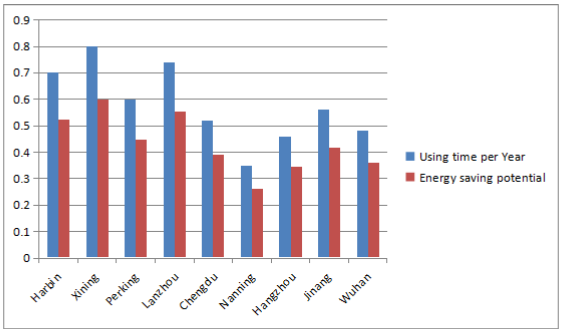


Fig.17 The using time and energy-saving potential of the typical city in china

VII. CONCLUSIONS

(1) In this paper, a time integration model for heat transfer in heat exchangers is established based on finite time thermodynamic analysis. The model can reflect the actual heat transfer process of the heat exchanger. The model can be used to optimize the charging capacity of heat exchangers. And the model can reflect the relationship between heat transfer and heat exchanger height. Provide judgment and reference for the layout of IT equipment.

(2) The hourly simulation model of CDU is simple, and the simulation error is small. Compared with classical thermodynamic analysis, it can reflect the actual operation of the system. The simulation precision is high. Moreover, the model can provide a reference for the design and optimization of CDU parameters.

(3) Compared with the traditional air conditioning system, the flow rate of working fluid in the unit water-cooled multi-connected heat pipe system is much lower than that in the traditional refrigeration system, which fails the empirical formula of "one driving more" in the refrigeration system. The existing calculation of flow rate and thermal resistance are independent engineering projects, and there is no unified conclusion, so it can not be widely applied. In this paper, the liquid separation model is established according to the network node flow method. Using the M language of MATLAB software, the program for solving the network node pressure and flow rate of the unit water-cooled multi-connected heat pipe system is compiled. The adaptive performance of the system under different loads is simulated. The test results show that the calculation method meets the requirements of engineering accuracy and can be used as a reference for the design of refrigeration requirements under different load conditions.

(4) The model provides a new method for the simulation of the thermal system.

ACKNOWLEDGMENT

This paper was supported by the Hunan province postdoctoral Innovation Fund.

REFERENCES

[1] Lingen Chen, Fengrui Sun, and Chih Wu. Optimal allocation of heat-exchanger area for refrigeration and air-conditioning plants. Applied Energy,77(3)(2004) 339-354.
[2] Hassan, M. A. M. Investigation of heat pipe performance as heat exchanger using alternative refrigerants. International Journal of Energy Engineering, 139(1) (2013) 18-24.
[3] Chang Yue, Quan Zhang, Zhiqiang Zhai. et al. Numerical investigation on thermal characteristics and flow distribution of a parallel micro-channel separate heat pipe in the data center. International Journal of Refrigeration.,98(2019) 150-160.
[4] S. Zou, Q. Zhang, Y. Yu, C. Yue, J. Wang, X. Ma, Field study on the self-adaptive capacity of multi-split heat pipe system (MSHPS) under non-uniform conditions in the data center. Applied Thermal Engineering.,160(2019).
[5] LiLing et al., Simulation of a microchannel separate heat pipe (MCSHP) under low heat flux and low mass flux. Applied Thermal Engineering, 119(5)(2017) 25-33.
[6] Liping Zeng, Xing Liu, Quan Zhang, et al. Research on Heat Transfer Performance of Micro-Channel Backplane Heat Pipe Air Conditioning System in Data Center. Applied science,(10)(2020) 583-598.
[7] Yanghua Cai, Zongtao Li, Jingmei Zhai. et al. Experimental investigation on a novel multi-branch heat pipe for multi-heat source electronics channel heat exchangers. International Journal of Heat and Mass Transfer. (104)(2017) 467-477.
[8] Feihu Chen, Shuguang Liao. Simulation Research on heat pipe air conditioning system in the data center based on thermodynamic simulation software [J]. Refrigeration and energy, (2)(2016) 28-34.
[9] Tao Ding et al., Experiment research on influence factors of the separated heat pipe system, especially the filling ratio and Freon types. Applied Thermal Engineering,118(25)(2017) 357-364.
[10] Li Ling, Quan Zhang, Liping Zeng. Performance and energy efficiency analysis of data center cooling plant by using lake water source. 10th International Symposium on Heating, Ventilation and Air Conditioning, ISHVAC2017.
[11] Jeon, Sung-Taek, and Cho, Jin-Pyo. An experimental study on the performance of the separate type heat pipe in accordance with the refrigerant charge. Journal of the Korea Academia-Industrial cooperation Society, 16(3) (2015) 1600-1604.
[12] Chao Dang Li, Jia Qian, yi lu. Investigation on thermal design of a rack with the pulsating heat pipe for cooling CPUs. Applied Thermal Engineering,110(5)(2017) 390-398.
[13] L.Y. Zhang, Y.Y. Liu. Experimental investigation and economic analysis of gravity heat pipe exchanger applied in communication base station. Applied Energy. 194(15) (2017)499-507.
[14] Tao Ding, et al. Application of separated heat pipe system in data center cooling. Applied Thermal Engineering..109, Part A (25)(2016) 207-216.
[15] Hainan Zhang , et al. Numerical investigation on an integrated system of mechanical refrigeration and thermosyphon for free cooling of data centers. International Journal of Refrigeration, 60(2015) 179-182.
[16] ChengchuYan, et al. A seasonal cold storage system based on separate type heat pipe for sustainable building cooling. Renewable and. Sustainable Energy Review,58(2016) 666-673.
[17] Mustafa AliErsöz, and AbdullahYıldız. Thermoeconomic analysis of thermosyphon heat pipes. Renewable Energy. 85 (2016) 880-889.

[18] Xingxing Zhang et al., Comparative study of a novel liquid-vapor separator incorporated gravitational loop heat pipe against the conventional gravitational straight and loop heat pipes – Part II: Experimental testing and simulation model validation. Energy Conversation and Management. 93(15(2015) 228-238.

[19] Asoka RG, Aishwarya N, Rajasekar S, Meyyappan N, Dynamic Simulation of Double Pipe Heat Exchanger using MATLAB Simulink SSRG International Journal of Chemical Engineering Research 4(1)(2017) 6-13.