DESIGN OF THE AIR COOLING SYSTEM FOR THE HIGH VOLTAGE POWER SUPPLY OF A ELECTRON ACCELERATOR

Feng Zhong#, Lei Yang, Jun Yang, Xiang Lei, State Key Laboratory of Advanced Electromagnetic Engineering and Technology (AEET), College of Electrical & Electronic Engineering, Huazhong University of Science and Technology, Wuhan 430074, P.R.China

Abstract

High voltage electron accelerators are widely applied in many fields of radiation processing, and the high voltage power supply is the critical equipment for the accelerator. For the requirement of high voltage, the design of locating the power supply in a steel barrel filled with atm SF_6 is commonly used. Considering the various losses of the power supply, an air-cooling system is needed. This paper presents the design of the air-cooling system for the high voltage power supply. The fluid simulation of SF_6 based on Fluent and the optimal design of the air duct's structure and the thermal efficiency has been done. The comparison and analysis of the simulation and the empirical formula result is also carried out. It illustrates the design of the air-cooling system can satisfy the demand of the heat radiation This paper also provides an effective method for the optimal design of the air duct's structure and the maximize efficiency of heat exchange.

INTRODUCTION

High voltage power supply is the critical equipment for high voltage electron accelerators.The air-cooling system is one of the important support systems for high voltage power supply.This paper presents air-cooling system design through the SF_6 cooling the heat elements and the optimal design of the air duct's structure and the thermal efficiency based on Fluent.

Because the output of the high voltage power supply is 20kW to 100kW, the heating loss cannot be ignored. The power supply is in an enclosed steel drum, so relaying only on nature cooling cannot guarantee the key devices of the power supply work in a safe temperature, we need to design a cooling system to ensure the safety. The internal loss of the high voltage power supply includes the heat of electrical components, primary coil, second coil and magnetic core and so on. The structure of cooling system we can see in the Figure 1.

The cooling system structure of high voltage power supply is shown in Figure 1. It is composed of a shell, finned tube, partition and inlet duct etc.The heat transfer process is as follows: A fan take the SF6 gas into the intake duct. The gas flow into the heat exchanger uniformly and is cooled. Then after cooled, the gas is sent to the tube and into the area which obtain electrical components to cool [1].

There are three types of heat transfer: conduction heat transfer, convection heat transfer and radiation heat transfer [2]. Fluent is a kind of advanced software. It is mainly used for fluid simulation [3].

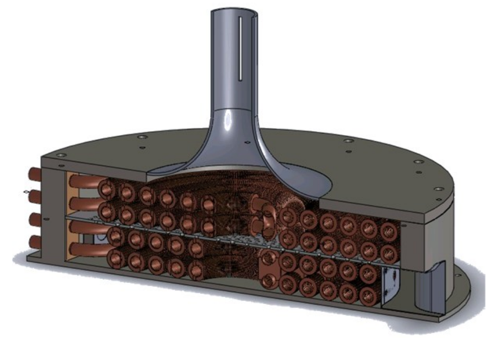

Figure 1: Structure of cooling system.

AIR-COOLING SYSTEM DESIGN

The Simulation Based on Fluid

First, we build a mock-up of the area based on gambit.

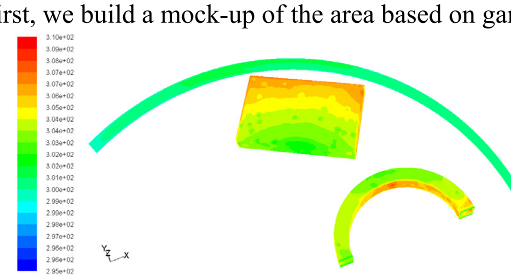

Figure 2: Contours of static temperature.

After a series of installations, such as meshing generation and assigning boundary conditions and so on, the calculation results reach a steady state after 505 iterations. We choose to display the cloud of temperature. The figure 2 is the display of temperature which the exchange result between SF_6 and heat elements based on Fluid.

From the picture, the cuboids means electric capacity and the half tourus means electric resistance. We can see that the highest temperature in the center area is 310K at the velocity of 2m/s.

The Calculation of Empirical Formula

Through the simulation calculation, we will use the empirical formula to test and verify. There are three areas in each layer of high voltage power supply. According to the estimation, the exchange area are about 0.13 m^2 and the heating power is 100W, so we assume that the SF_6 gas speeds is 2m/s and the temperature of inlet air are 23℃. As the temperature of electric capacity and resistance are unknown, we cannot get the qualitative temperature of

316845860@qq.com

ISBN 978-3-95450-122-9

SF_6.So we need to adopt the way of hypothesis-iterative computation. First, we assume the temperature of the heat elements is 32 ℃. t_m , t_∞ , t_w express the qualitative temperature of SF_6, the inlet temperature of SF_6 and solid surface temperature of heat elements respectively.

Then

$$t_m = \frac{1}{2}(t_\infty + t_w). \quad (1)$$

We get the qualitative temperature is 27.5℃.It is close to 300K, so we get the following thermal physical properties from the book about the SF_6 in the six atmospheres at the 300K.

$$\begin{aligned} &\lambda = 0.13W/(M \bullet K), \\ &\upsilon = 2.641\times10^{-6} M^2/S \\ &\Pr_m = 0.5, \rho = 36.0516 kg/m^3 \\ &C_P = 0.666 kJ/(kg\cdot k) \\ &\frac{vx_c}{\nu} = 5\times10^5 \end{aligned} \quad (2)$$

The distance of the turn point from laminar flow to turbulence flow to the leading edge is 0.66m.The actual length is 0.45 which is less than 0.66, so the laminar motion is the main form of SF_6 movement. But in fact the type of SF_6 movement is the mixture of laminar and turbulent motion. Considering the margin, we still regard it as the laminar motion. From the local definition of Re and formula of Nu:

$$\begin{aligned} &\mathrm{Re}_m = \frac{vx}{\nu} = 3.4\times10^5 \le 5\times10^5 \\ &Nu_m = 0.664\,\mathrm{Re}_m^{1/2}\,\Pr_m^{1/3} \approx 309.74 \end{aligned} \quad (3)$$

Through the definition of Nusselt Number on Laminar forced flow over a plate:

$$Nu_m = \frac{h_c l}{\lambda}. \quad (4)$$

we can get the result of hc is 89.27W/(m2.K).Hc represents the heat transfer coefficient.

Taking advantage of Newton cooling formula as follow:

$$100 = 89.27\times0.13\times[(t_w)_1 - 23]. \quad (5)$$

The calculation of temperature is 31.62 ℃ .It is corresponding to assumed temperature. So the temperature of electric capacity and resistance is 32℃.

Then, for the formula,

$$\phi_c = h_c A \Delta t. \quad (6)$$

The result is 104W, it meet the needs. Through the calculation of empirical formula, the result satisfies the requirement as long as the gas speed is not less than 2m/s. The solution of calculation and simulation is basically the same. So through the calculation of empirical formula and simulation based on Fluent, the speed of SF_6 is completely meeting the requirement under the 2m/s and the total flow is 39m³/h.Considering the safety margin, we take the 43m³/h.Through the calculation of empirical formula, it proved the simulation based on Fluent to be effective.

The Simulation of Central Tube

The main diameter of hub pipeline is 100mm and the total flow is 43m³ /h, so we confirm that the speed of hub pipeline is 1.6m/s.The height of each floor is 50mm and there are seven floors in total. So we build the physical model of central tube. Through changing aperture size in dozen times, we have gotten the best litter sizes of hub pipeline which make the flow of each floor uniform. This is the simulation pictures in the figure 3.

The calculation results reach a steady state after 180 iterations.

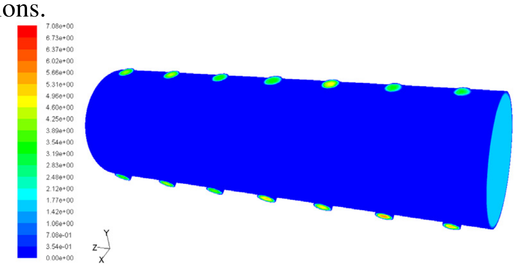

Figure 3: Velocity contours of central tube.

According to the velocity contours, we get the most suitable size table as follow:

Table 1: The Diameter of Pipeline and Flow

Number	Velocity(m/s)	Diameter(mm)	Flows (m³/min)
1	4.55	13	0.604
2	4.1	14	0.630
3	3.5	15	0.618
4	3.1	16	0.623
5	3.45	15	0.610
6	3.5	15	0.618
7	3.4	15	0.610

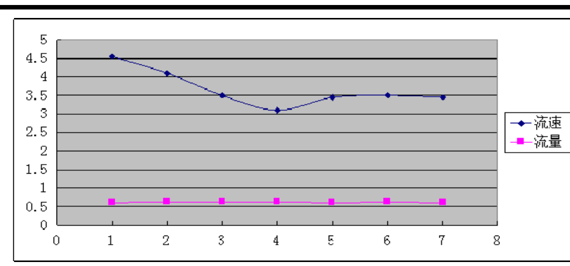

Figure 4: The velocity and flow of tube.

From the Table 1 and Figure 4, we can find that the flow of each layer is even.

The Simulation of Diversion Conduit

Because the heat exchanger of our high voltage power supply is disc-shaped. In order to make full use of heat exchanger and improve its efficiency, we design a diversion tube specially.

First we build and mesh half an entity model based on Gambit. Serial boundary conditions have been established. The speed of inlet is 11m/s. We need to get the pore size to achieve balance of flow and after more than ten times simulation, we get the suitable aperture.

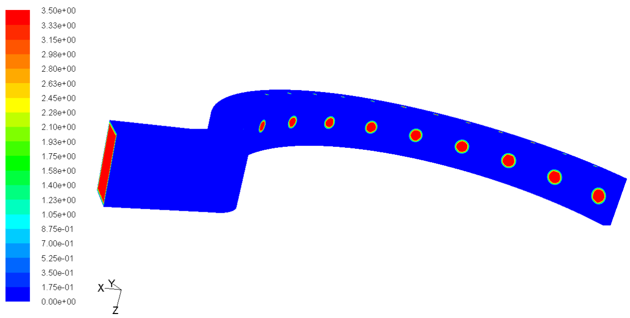

Figure 5: Velocity contours of diversion tube.

Then the calculation results reach a steady state after 135 iterations. And we get the velocity cloud of diversion conduit (see in the Figure 5) and create a part velocity and aperture table. The Table 2 shows the result. The total flow of the side hole is about 6.7 times more than the above hole and the flow of each hole is in uniform.

Table 2: The Part Diameter of Diversion Tube and Flow

Big hole flow(m³ /h)	**Velocity(m/s)**	**Angle(°)**	**Radius(m)**
4.018572	15.8	20	0.01
3.94227	15.5	35	0.01
3.96739	14	50	0.011
4.082	13	65	0.011
Little hole flow(m³ /h)	**Velocity(m/s)**	**Angle(°)**	**Radius(m)**
0.442112	8.8	15	0.005
0.42704	8.5	25	0.005
0.40192	8	35	0.005
0.40192	8	45	0.005

ENGINEERING PROCESS

After the design of cooling system is completed, we made some demands and submitted to the factory to product. The design is reasonable after actual test and the rest of work is to measure it. The Figure 6 and Figure 7 are the object pictures:

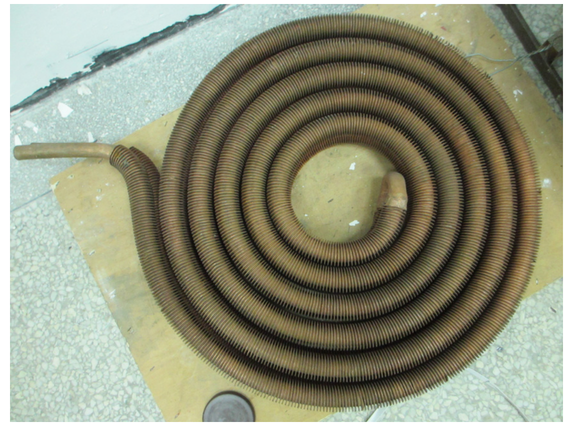

Figure 6: Heat exchanger.

Figure 7: Fan and diversion tube.

CONCLUSION

In the cooling system design process, we are vulnerable to cutbacks and there are also some disadvantages in the whole system. But the design of cooling system is successful. After having used the Fluent software to make SF_6 exchange with heat elements and the optimal design of the air duct's structure simulation and optimization, a high efficiency technology method is finally created. It is suitable.

REFERENCES

[1] Lei Yang, "Study on High-Voltage and High-Power DC Power Supply of Inslated Core Transformer" Wuhan: Huazhong University of Science and Technology 2011.

[2] Incropera, Frank, and David DeWitt. "Introduction to heat transfer."(1985).

[3] Version, Fluent. "4.2User's Guide. "Fluent Incorporated, New Hampsier (1993).

ISBN 978-3-95450-122-9