Zhongyi WANG, Jia HAN, Song WANG, Nan YE

College of Power and Energy Engineering, Harbin Engineering University

The Research of Solution Methods on the Marine Exhaust Ejector Device with Swerve

Abstract. The research of this paper is about solution methods on the marine exhaust ejector device with swerve. The whole system of the device is divided into two parts, including the filtration stage and the channel stage. The porous media model is established for the damping of filtration stage. The scaled model experiments are achieved through the similar conversion under the principle of keeping the same Mach numbers of the main air jet. The experimental results and numerical simulation results agree well. It can provide the theoretical basis for the research of the exhaust ejector with swerve and the large-size model.

Streszczenie Przedstawiona praca dotyczy metod rozwiązywania problemów związanych z okrętowym eżektorem wyciągowym z odchyleniem. Urządzenie składa się ze stopnia filtrującego i stopnia kanału. W stopniu filtrującym do tłumienia przyjęto model ośrodka porowatego. Skalowanie modeli eksperymentalnych osiągnięto przez przekształcenie według zasady utrzymania tej samej liczby Macha w głównym strumieniu powietrza. Wyniki eksperymentalne i symulacji numerycznych są zgodne. Praca może stanowić bazę teoretyczną do badań wyciągowego eżektora z odchyleniem oraz modelu w większej skali. Badania metod rozwiązań urządzenia okrętowego eżektora wyciągowego z odchyleniem

Keywords: gas turbine, exhaust ejector, porous media, similar conversion
Słowa kluczowe: turbina gazowa, eżektor wyciągowy, ośrodek porowaty

Introduction

The changes of the pressure around the high speed fluid can inject the circumjacent fluid. The device depending on this kind of principle is called the ejector [1].

In the work course of the marine gas turbine, the engine room maintains a high temperature. If the necessary technology measure is not adopted, the operating temperature of engine room will be too high to operate the instrument and equipment normally, and the fire can occur easily [2]. Therefore the important mission of lowering cabin temperature has been undertaken by the application of exhaust ejector [3].

The gas turbine is covered with the shell, and a fixed interval is remained between the cover shell and the gas turbine outside surface [4]. The air inlet is founded in the corresponding position of cover shell, and the air inside the engine room can flow through the hot surface of the unit [4[. The exhaust ejector can take the part of heat quantity from the hot surface with the injected cold air, and it also can reduce the temperature of engine room [5, 6]. The injected cold air is always called the second air [7].

Fig 1 shows the general arrangement plan of the exhaust ejector on the ship.
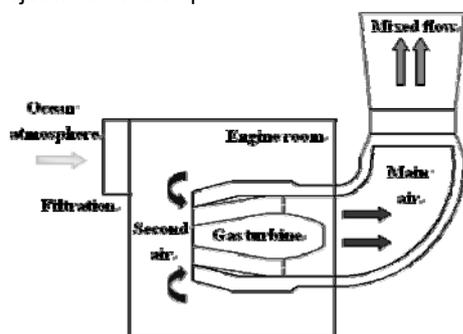

Fig.1. Arrangement plan of the exhaust ejector

The front end of the device locates inside the engine room, and the back end exposes outside. The hot gas exhausts from the gas turbine is called the main air. The waste gas of the gas turbine exhausts into the atmosphere through the bent pipe of 90 degrees. The smoke gas possesses high temperature and speed, and it has the effect of drawing the cold air of engine room. The second air can cool the surface parts of unit and mix with smoke gas. The second air passes the air filtration which locates in the front of the engine room, in order to enter the engine room from the ocean atmosphere.

Solution tactics

Considering the general arrangement plan, the whole system is separated into two parts, namely the filtration stage and the channel stage. The filtration stage produces the negative pressure of the second air entrance near the exhaust ejector. Along with the difference of work condition, the flow rate of the second air and the pressure loss of the air filtration are also dissimilar. In order to research the whole system by the way of numerical simulation, the filtration stage should be changed into the boundary condition. The work emphasis is the numerical simulation of channel stage. The whole solution course is shown as the Fig2.
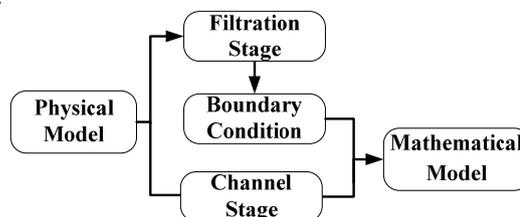

Fig.2. The solution course of whole system

Aerodynamic Characteristic of Filtration Stage

The second air entrance of the exhaust ejector locates inside the engine room. The second cold air will be drawn into the exhaust ejector by the exhaust gas through the air filter that locates in front of the engine room. (The structure of air filter is shown as the Fig 3.) When the air flows through the filter at a definite velocity, there will be a fixed pressure loss.
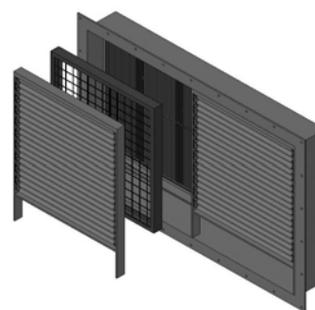

Fig.3. The air filter of engine room

In order to get the real performance of exhaust ejector, the definite negative pressure should be given to the second air entrance as a boundary condition, when computing the different work conditions. The maximum pressure loss of the air filtration of the engine room is limited to less than 400Pa. During the simulation of the original model, the different negative pressure will be set to the second air entrance in dissimilar work conditions.

Fig4 shows the relationship between negative pressure and flow rate of the second air under different work conditions, and the picture also includes the resistance of engine room air filtration. The ejection performance of a certain model under different work conditions can be got in this way, but at least three kinds of negative pressure should be set to the second air entrance. In order to get the flow rate of injected air, the result must be combined with the resistance of engine room air filtration. This method brings a lot of unnecessary troubles to the computational work; therefore, it is necessary to take some feasible measures to simplify the work of computing. The application of the porous medium model can convert the filtration stage into a kind of boundary condition. It can give the polynomial curve through the relationship of velocity and total pressure loss, combining with the filter resistance characteristics obtained from experiments. This kind of append condition should be set to the second air inlet boundary.
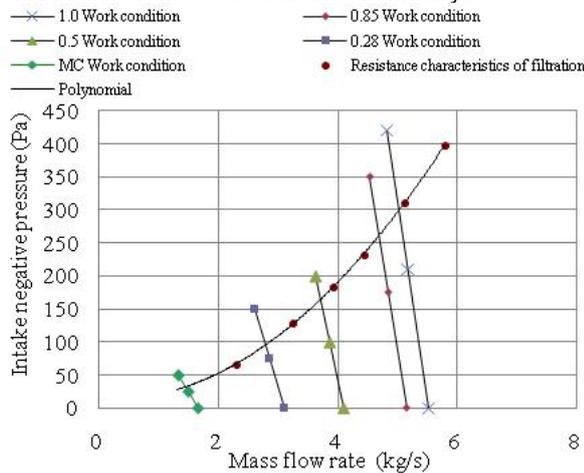

Fig.4. The curve about flow rate and negative pressure of the second air

Porous Media Conditions

Porous media are modeled by the addition of a momentum source term to the standard fluid flow equations. The source term is composed of two parts: a viscous loss term (Darcy, the first term on the right-hand side of Equation 1), and an inertial loss term (the second term on the right-hand side of Equation 1).

$$(1) \qquad S_i = -\left(\sum_{j=1}^{3} D_{ij} \mu v_j + \sum_{j=1}^{3} C_{ij} \frac{1}{2} \rho |v| v_j \right)$$

Where S_i is the source term for the i th (x, y, or z) momentum equation, $|v|$ is the magnitude of the velocity and D and C are prescribed matrices. This momentum sink contributes to the pressure gradient in the porous cell, creating a pressure drop that is proportional to the fluid velocity (or velocity squared) in the cell. To recover the case of simple homogeneous porous media

$$(2) \qquad S_i = -\left(\frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v| v_i \right)$$

Where α is the permeability and C_2 is the inertial resistance factor, simply specify D and C as diagonal matrices with $1/\alpha$ and C_2, respectively, on the diagonals (and zero for the other elements).

It also allows the source term to be modeled as a power law of the velocity magnitude

$$(3) \qquad S_i = -C_0 |v|^{c_1} = -C_0 |v|^{(C_1 - 1)} v_i$$

Where C_0 and C_1 are user-defined empirical coefficients. In the power-law model, the pressure drop is isotropic and the unit of C_0 is SI.

When we define the porous medium boundary condition, the resistance performance of the filtration can be used according to the data in table 1.The relationship of total pressure loss and velocity is used to draw up the synthetic quadratic polynomial with the resistance performance. The synthetic formula expression is shown as the follow form.

$$(4) \qquad \Delta P = 1.922 v^2 + 0.591 v$$

The permeability α and the inertial resistance factor C_2 can be got form the Equation 4, and they should be attached to the second air inlet boundary.

Table1 The Experimental Results of Air Filtration

Q (m³/h)	ΔP (Pa)	A (m2)	v (m/s)
6935.63	66	0.33	5.85
9744.19	128	0.33	8.22
11809.4	184	0.33	9.96
13393.63	232	0.33	11.3
15371.39	310	0.33	12.96
17423.16	398	0.33	14.69

In table 1, the Q is flow rate of filter; ΔP is pressure loss; A is area of the second air entrance; v is velocity of the second air entrance

Mathematical model and boundary condition

By using the Boussinesq vortex viscosity supposition, the Navier-Stokes equations of the compressible glutinous gas, neglecting mass force, may be described as follows:

Continuity equation:

$$(5) \qquad \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0$$

Momentum equation:

$$(6) \qquad \frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$$

Energy equation:

$$(7) \qquad \frac{\partial}{\partial t}\left[\rho \left(e + \frac{u_i u_i}{2} \right) \right] + \frac{\partial}{\partial x_j}\left\{ u_j \left[\rho \left(e + \frac{u_i u_i}{2} \right) + p \right] \right\}$$
$$= \frac{\partial}{\partial x_i}\left(\lambda \frac{\partial T}{\partial x_i} + u_j \tau_{ij} \right)$$

Equation of state:

$$(8) \qquad \frac{p}{\rho} = RT$$

In the formula:

$$(9) \qquad \tau_{ij} = \left(\mu + \mu_T \right)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left(\mu + \mu_T \right) \frac{\partial u_k}{\partial x_k}$$

τ_{ij} is viscous stress tensor; ρ is density; μ_i is component of velocity; P is pressure; e is unit mass internal energy; λ is thermal conductivity; T is temperature; μ is dynamically viscous coefficient; μ_T is turbulent viscosity coefficient.

When neglecting the gravity effect, by using a k-ε turbulence model to simulate vortex viscosity coefficient, the turbulence kinetic energy k and its rate of dissipation ε are obtained from the following transport equations:

$$(10) \qquad \rho \left(\frac{\partial k}{\partial t} + u_i \frac{\partial k}{\partial x_i} \right) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k - \rho \varepsilon - Y_M$$

$$(11) \quad \rho\left(\frac{\partial \varepsilon}{\partial t} + u_i \frac{\partial \varepsilon}{\partial x_i}\right) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_T}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. To evaluate G_k in a manner consistent with the Boussinesq hypothesis,

$$G_k = \mu_T S^2$$

Where S is the modulus of the mean rate-of-strain tensor, defined as:

$$S \equiv \sqrt{2S_{ij}S_{ij}}$$

The average strain rate tensor is given by the equation as follows:

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, $Y_M = \rho\varepsilon 2M_t^2$.

Where M_t is the turbulent Mach number, defined as:

$$M_t = \sqrt{\frac{k}{a^2}}$$

The turbulent viscosity, μ_T, is computed by combining k and ε as follows:

$$(12) \quad \mu_T = \rho C_\mu \frac{k^2}{\varepsilon}$$

The value about constant $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ and the turbulence Prandtl number σ_k and σ_ε of k and ε given by B.E.Launder and D.B.Spalding[8] are respectively: $C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92, C_μ=0.09, σ_k=1.0, σ_ε=1.3.

Table2 Main air parameters in each work conditions

Work condition	Temperature (K)	Flow rate(kg/s)
1.0	893	27.8
0.85	863	26.3
0.5	813	21.7
0.28	810	16.7
MC	833	9.2

Table 2 gives the gas parameters of the main air under different work conditions. The 1.0 work condition is Full-load conditions, and the MC work condition is the Low-speed cruise condition.

The boundary conditions are as follows:

Reference pressure is 101325Pa.

Main air gas inlet: mass flow import, gas parameter under different work conditions can refer to the data in Table 2, airflow direction is normal.

The second air inlet: It is confirmed combining the formula (4) by using the porous media condition, the temperature is 328K.

Exportation: pressure outlet, the pressure is 101325Pa, and the temperature is 300K.

Solid wall: adiabatic, and do not have the slipping.

Turbulence model: standard k-ε model.

Result of numerical simulation

Fig5 shows total pressure contours on longitudinal section of different work conditions using the porous media conditions. It shows the total pressure distribution circumstances of 1.0 and 0.85 operating modes of two pictures from left to right. The total pressure distributes on the vertical section is consistent, but the extremum is different.
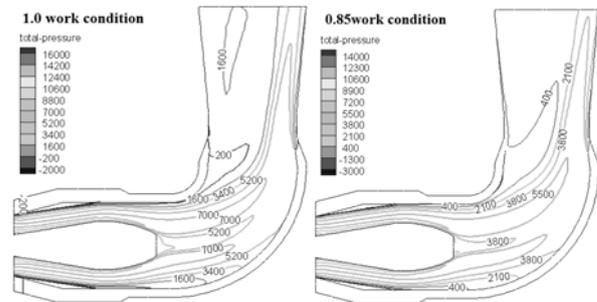

Fig.5. Pressure counters distribution of vertical section

Table3 shows the result of numerical simulation under the porous media conditions. The negative pressure value of the second air inlet which gets by the numerical simulation is close to the calculation result from formula. The relative error between the results is about 5%, which can be accepted. And the negative pressure of the second air inlet can be replaced by the method of porous media condition. The work load of numerical simulation can be reduced two thirds at least under each work condition. It has important significance for the numerical simulation of the marine gas turbine exhaust ejector. This kind of boundary condition treatment also can be used widely in the numerical simulation course of the complex fluid mechanical with pressure jump.

Table3 Numerical simulation result under porous media condition

Work condition	Inlet velocity (m/s)	Pressure loss under porous media condition (Pa)	Calculation result from formula (Pa)	Relative error (%)
1.0	12.46	321.00	305.99	4.68
0.85	11.62	279.73	266.61	4.69
0.5	9.22	177.19	168.76	4.76
0.28	6.93	101.51	96.55	4.89
MC	3.69	29.98	28.35	5.46

The negative pressure value of the second air inlet which comes from the numerical simulation accords with value of the intersection in Fig4 under the several work conditions. This appearance proves the correct porous media condition used in numerical simulation. The porous media model can be used correctly in numerical simulation to replace the component which has the specific total pressure characteristic. The whole system can be simulated using this method, and the physics model is changed to numerical model under the porous media condition.

Similar conversion

The geometry size of marine gas turbine exhausts injection is very large, thus the experiment of full ruler model is hard to accomplish under the condition of laboratory. The experiment of reduced rate model is used to do the research in laboratory. The similar norm must be observed in the experimental research, but the request is only satisfied on the special section in the engineering research.

The similar criterion which should be observed in model experiment of marine gas turbine exhaust ejector contains several criterion numbers. They are Geometry similar, Mach number similar and the Reynolds number similar. The Prandtl number, Froude Number, Strouhal number and Euler number also should be obeyed in research. Generally speaking, the Geometry similar is easy to achieve by the way of designing the experiment model under the fixed proportion.

It is difficult to satisfy the Mach number and Reynolds number at the same time in the experiment course. When

the Reynolds number is bigger than the critical Reynolds number, the change of loss efficiency ζ will be very little. This kind of area is called self-modeling area. The effect of viscosity frictional force is very small in this area. The value of Reynolds number will not influence the flow in the self-modeling area. The Reynolds can be express as the formula:

(13)
$$Re = \frac{uD}{v}$$

where v is coefficient of kinematic viscosity, u is the velocity, and D is characteristic length.

The Reynolds numbers on the surface of main air jet is $10^7 > 1 \times 10^5$ through computing result and dimensional analysis, and the flow is in the self-modeling area. The Mach numbers on the surface of main air jet in experimental model is identical with situation of the real model.

In this article the conversion between the different states and geometric dimension models should be under the principle of keeping the same Mach numbers. The parameter of the model with 1:1 rate and hot state should be confirmed in first step. The parameter of the model with 1:1 rate and cold state should be changed under the principle of same Mach numbers on the main air jet. The parameter of the model with 1:4.15 and cold model will be getting in the same way.

The rate of the experimental model is 1:4.15. The result of numerical simulation can be compared with the experimental result, in order to validate the correctness of numerical simulation. And the result of model with 1:1 rate and hot state also can be validated.

The different states models will be changed as the follow relationship, when the rate is 1:1 and neglecting the influence of viscosity is neglected. In the formula, the h means the hot state, and the c means the cold state.

(14)
$$Q_h = \rho_h A V_h, Q_c = \rho_c A V_c,$$

(15)
$$\frac{Q_c}{Q_h} = \frac{\rho_c V_c}{\rho_h V_h} = \frac{\dfrac{P_c}{RT_c} M_a \sqrt{\gamma RT_c}}{\dfrac{P_h}{RT_h} M_a \sqrt{\gamma RT_h}} = \frac{P_c}{P_h} \sqrt{\frac{T_h}{T_c}}$$

When the change of the pressure is not much, it can be neglected. The flow ratio in different states in the same rate is the inverse ratio of the square root about the temperature. It can get the main air flow value of the same rate about the cold model by using this formula, in order to replace the hot model. The different states models in the same ratio can be converted by using this formula.

In cold models, the flow rate of different scale models is decided by the ratio of area. We can get the relationship between the model with 1:1 rate with hot state and the model with 1:4.15 rate with cold state, through the conversion by the model with 1:1 rate with cold state. In other words, the big scale and hot model can be replaced with the little scale and cold model in experiment research. This can reduce the processing costs of experimental model. The experiment requirement of practical model on the ship also can be satisfied under the laboratory condition.

Table4   Statistic results of different rates and states

Ratio	Main air temperature (K)	Main air flows (kg/s)	Second air Flows (kg/s)	Inject coefficient (%)	Hot inject coefficient (%)
1:1	893	27.80	5.21	18.75	18.75
1:1	450	39.16	5.51	14.07	19.82
1:1	300	47.96	5.54	11.55	19.93
1:2	300	11.99	1.38	11.47	19.79
1:4.15	300	2.79	0.32	11.39	19.65

Table 4 gives the statistic results of different rates and state models. When the main air temperature is 300K, the inject coefficient of the ratio with 1:1 is bigger than the 1:4.15's. The difference of hot state inject coefficient is 1.18% when the models are the same scale.

Generally speaking, the performance of the model with 1:1 rate in hot state can be replaced with ones of the model with 1:4.15 rate in cold state. The experiment requirement of practical model on the ship also can be satisfied under the laboratory condition.

Experiment research

The air blower put the air with the fixed pressure into the main air pipeline, which is linked with the air blower, in order to simulate the main air with cold state. The different work conditions can be controlled by changing the flow rate of the air blower. The second air inlet of the exhaust ejector lies inside the pressurize tank, and there is an inlet pipeline for the second air on the top of the tank. The second air is pumped into the pressurize tank through the inlet pipeline which has several flow measuring instruments. The inlet of the exhaust ejector lies inside of the pressurize tank, and the outlet lies outside. The flow rate should be regulated under the fixed work condition during the experiment course. The parameter of the main air including the flow rate and temperature should be recorded. Under the same work condition, several group experiments with different negative pressure of the second air inlet should be done. The flow rate, temperature and the negative pressure of the second air should be recorded.
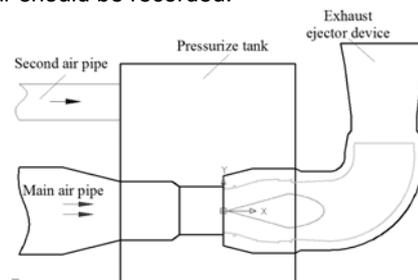

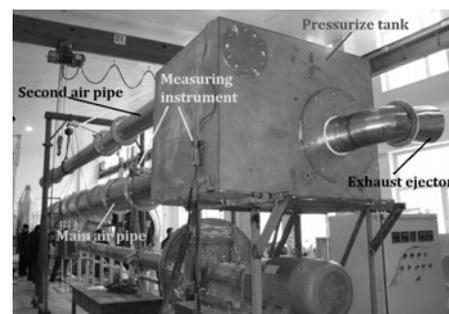Fig.6. The arrangement of experimental facilities

Fig.7. The local arrangement of experimental facilities

Fig 6 shows the arrangement of experimental facilities. And the Fig7 shows the local arrangement of the experiment equipments and the around devices.

The flow of main air should be adjusted by using the conversion relationship between the model of 1:1 rate with hot state and the model with 1:4.15 rate with cold state, in order to keep the Mach number coincident of the two kinds of state, under the laboratory condition

The numerical calculation will be done by using the several experimental data, in order to verify the accuracy of the numerical computation. If the trend between the numerical simulation result and experimental data is consistent, the inject capability of the exhaust ejector under the 1.0 work condition can be replaced by the result of numerical simulation.

Table 5 shows the computation data comparing with the experimental results under the several work conditions. The trend between the computation data and experiment result is consistent. The contrast situation is shown as the Fig 8.

Table 5 The compare of computation and experiment result

Work condition	Inlet resistance (Pa)	MF (kg/s)	SFNS (kg/s)	ICNS (%)	SFE (kg/s)	ICE (%)
MC	-56	0.898	0.099	10.97	0.083	9.20
0.28	-120	1.616	0.224	13.84	0.172	10.67
0.5	-250	2.012	0.251	12.45	0.214	10.62
0.85	-360	2.587	0.334	12.91	0.285	11.00

In table 5, MF means main air flows, SFNS means second air flows under numerical simulation, ICNS means inject coefficient under numerical simulation, SFE means second air flows under experiment, ICE means inject coefficient under experiment.
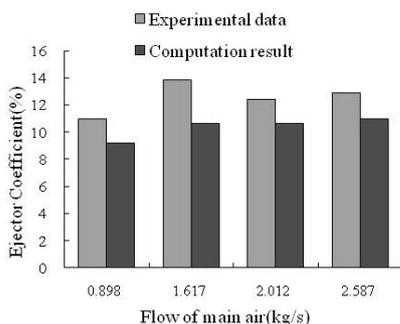

Fig.8. The compare of computation and experiment result

Conclusions

The performance of the marine gas turbine exhaust ejector has been researched by combining the method of numerical simulation and experimental research, and the conclusions are summarized as follows:

1. The whole system is divided into two parts, including the filtration stage and the channel stage. The porous media model is established for the filtration stage damping, and the whole system can be simulated successfully.

2. The similar conversion relationship between the exhaust ejector device models at different scales and working conditions is established, under the principle of keeping the same Mach number on the special section.

3. The performance of exhaust ejector device can be tested under the laboratory conditions, and the trend of experimental results with numerical simulation is consistent.

The ejector performance data under different exhaust ejector device structure parameters should be accumulated in the future. The appropriate database of design should be established, and it can provide the design reference for the exhaust ejector device. The measurement problem about the flow field velocity and pressure distribution in experimental research should be solved in the further work.

REFERENCES

[1]. Li Liguo, Zhang Jingzhou, Aviation ejector mixer. *National Defense Industry Press*, Beijing 2007,10-13.
[2]. Wang Zhongming, Marine gas turbine installations. *National Defense Industry Press*, Beijing 1981,243-245.
[3]. Li Dongming, Wang Lin, Wen Xueyou, *Journal of engineering for thermal energy and power*, Vol. 17(3) (2002), p. 226-230.
[4]. S.Bottenheim, A.M.Birk, D.J.Poirier, Performance of A Longitudinal-to-Slot Gas Turbine Exhaust Duct with A 90 Degree Bend. *Proceedings of ASME Turbo Expo 2004: Power for Land, Sea and Air*.GT2004-53373.
[5]. Wang Xiaoling, *Ship and boat*, (5) (2005), p. 35-40.
[6]. Asim Maqsood, A Study of Subsonic Air-Air Ejectors with Short Bent Mixing Tubes. *Department of Mechanical and Materials Engineering, Queen's University*. Kingston, *Ontario, Canada* 2008, p. 5-15.
[7]. Asim Maqsood, A.M.Birk, Improving the Performance of a Bent Ejector with Inlet Swirl. *Proceedings of ASME Turbo Expo 2008: Power for Land, Sea and Air*. GT2008-51209.
[8]. Wu Jianghang, Han Qingshu, Computational fluid dynamics theory, method and application. *Science Press*, Beijing 1988, p. 55-60.

Authors: *Wang Zhongyi, lecturer, College of Power and Energy Engineering, Harbin Engineering University, Tel:+8645182519347, E-mail:b205030024@126.com*