Numerical Study on Flow-Induced Vibration of Ejector Structure

NIE Xutao[1*], *LIU Zongzheng*[1], *QIN Chaojin*[2], *ZHANG Wei*[1]

1. China Aerodynamics Research & Development Center, Mianyang 621000, P. R. China;
2. Chengdu Star-Union Ltd., Chengdu 610000, P. R. China

(Received 26 March 2019; revised 28 April 2019; accepted 18 May 2019)

Abstract: High-speed airflow in wind tunnel tests usually causes dramatic vibration of ejector structure, which may lead to fatigue and even destruction of the wind tunnel. Therefore, analyzing and solving the flow-induced vibration problem is a tough and indispensable part of the wind tunnel security design. In this paper, taking a kind of two-stage ejector as the study object, multiple numerical simulation methods are adopted in order to carry out research on the analysis technique of the flow-induced vibration characteristics of ejector structure. Firstly, the structural dynamics characteristic is analyzed by using the ejector structural dynamics numerical model, which is built on the basis of finite element method. Secondly, the complex flow phenomenon is explored applying numerical fluid-dynamics model of the inner flow field of the ejector, which is constructed on the basis of finite volume method. Finally, based on the two numerical models above, the vibration response of the ejector structure induced by the high-speed airflow is computed via the fluid-solid coupling technique. The comparison of the simulation results with the actual vibration test indicates that these numerical simulation methods can accurately figure out the rule of flow-induced vibration of ejectors.

Key words: ejector; flow-induced vibration; fluid-solid coupling; numerical simulation

CLC number: TP391 **Document code:** A **Article ID:** 1005-1120(2019)05-0769-10

0 Introduction

The ejector uses the turbulent diffusion effect of high-speed primary gas to take away the upstream gas to form a certain negative pressure environment and complete the gas transportation, which can be regarded as a fluid pump[1-2]. Compared with other types of fluid pumps, the ejector is simpler in structure, lower in cost and can replace fans or compressors. As a driver or exhaust device for transonic wind tunnels, the ejector effectively improves the efficiency of aerodynamic test[3-4].

When the ejector is working, the high-pressure primary gas would accelerate through a group of nozzles to supersonic state and enter the suction chamber, bringing the low-pressure gas in the suction chamber into the mixing chamber. After exchange and turbulence of momentum, the two gases are decelerated and supercharged through the diffuser section and then discharged into air or the next stage. The intense gas mixing inevitably produces complex unsteady flow phenomena, including supersonic shear, strong shock waves, air flow separation, and pressure pulsation, which are bound to trigger structural vibration of the ejector and endanger the safety of wind tunnel equipment. Therefore, the structure vibration of ejector is a typical type of flow-induced vibration problem.

In the middle of last century, Blevins[5] firstly proposed the concept of flow-induced vibration, and summarized lots of flow-induced vibration phenomena in practical engineering. Later, some researchers carried out many investigations on low-frequency flow-induced vibration. For instance, Goyder[6] studied the fluid vibration problem of heat exchanger pipes. Zdravkovich[7] and Sumner[8] analyzed the

*Corresponding author, E-mail address: nie_xu_tao@163.com.

How to cite this article: NIE Xutao, LIU Zongzheng, QIN Chaojin, et al. Numerical Study on Flow-Induced Vibration of Ejector Structure[J]. Transactions of Nanjing University of Aeronautics and Astronautics, 2019,36(5):769-778.
http://dx.doi.org/10.16356/j.1005-1120.2019.05.008

vibration problem induced by the flow around cylinder. With the help of the computation technology, researchers developed the fluid-solid coupling simulation method to explore the structural vibration induced by high-frequency and high-speed airflow[9-11]. By contrast with the traditional techniques, the fluid-solid coupling method can complete the data mapping between fluid domain and solid domain based on the high precision interpolation algorithm, thus accurately exerting the aerodynamic loads on the structure mesh and obtaining high fidelity structural vibration response[12-14].

In this paper, several numerical simulation methods are adopted to analyze the structural dynamic characteristics of the ejector and to simulate the flow phenomena in the inner flow field. On this basis, the vibration response of the ejector structure under complex airflow load is calculated by means of the fluid-solid coupling technique. Finally, the flow-induced vibration test of the ejector structure is performed to verify the accuracy of the numerical simulation method.

1 Analysis of Structural Dynamics Characteristics

The research object is a two-stage ejector, as shown in Fig.1. The structure of each stage is identical, including the ejector nozzles, suction chamber, mixing chamber and diffuser section. Each stage of the ejector has six ejector nozzles arranged in an annular direction, which are used to accelerate high-pressure primary gas. In the suction chamber, a large amount of original low-pressure gas is taken away by the high-pressure gases, resulting in a great pressure drop, and the upstream gas is continuously filled, thus completing the gas transportation and pressurization.

The intense mixing of wide range pressure airflow forms an unusually complex unsteady flow field in the suction chamber, probably forcing the ejector nozzles to oscillate dramatically. The practical experiences show that after many wind tunnel tests, fatigue fracture occurs frequently on the noz-

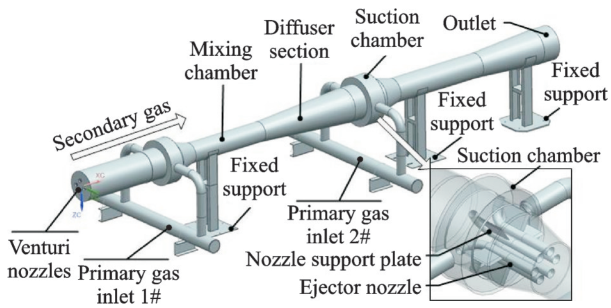


Fig.1 Composition of ejector structure

zle support plates and indispensable welding repair needs to be carried out. This not only consumes manpower and material resources, but also greatly affects the test task progress. Consequently, in order to improve the equipment service life, it is necessary to focus on studying the flow-induced vibration characteristics of ejector structure and lay a crucial foundation for the later structural optimization or vibration suppression.

Firstly, finite element numerical simulation technique is applied to analyze the structural dynamics of the ejector suction chamber. In general, the structural dynamics equation is given as follows

$$\boldsymbol{M}\ddot{\boldsymbol{X}}(t)+\boldsymbol{C}\dot{\boldsymbol{X}}(t)+\boldsymbol{K}\boldsymbol{X}(t)=\boldsymbol{f} \tag{1}$$

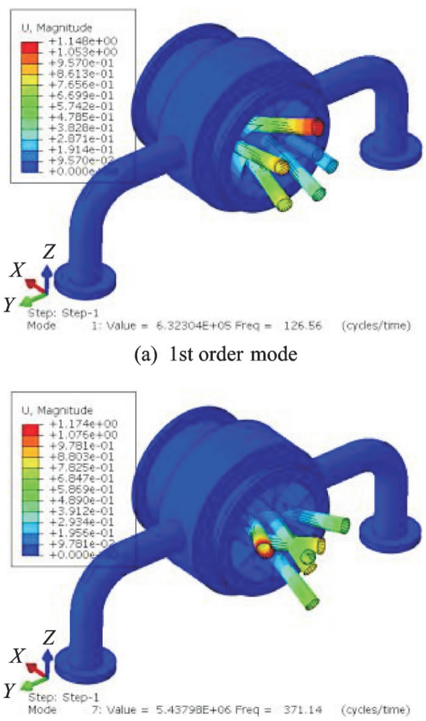
where $\boldsymbol{M}$ is the mass matrix, $\boldsymbol{C}$ the damping matrix, $\boldsymbol{K}$ the stiffness matrix, $\boldsymbol{f}$ the external load vector, $\boldsymbol{X}(t)$ the structural response vector, and t the time variable.

Neglecting the structural damping and using the Fourier transform method, Eq.(1) is reestablished in the frequency domain and then the structural modal equation can be obtained.

$$\boldsymbol{M}\ddot{\boldsymbol{X}}(\boldsymbol{\omega})+\boldsymbol{K}\boldsymbol{X}(\boldsymbol{\omega})=\boldsymbol{0} \tag{2}$$

where $\boldsymbol{\omega}$ is the angular frequency. The equation solution is composed of the natural frequency and the corresponding modal shape.

Based on the above equation, the finite element software ABAQUS is used for numerically calculating the structural modes of the ejector suction chamber. The vibration results are shown in Fig.2. Among them, the modal modes from the 1st order to the 6th order are similar, which represent lateral bend of the support plate with a frequency range of 126.6 Hz to 127.3 Hz. Additionally, the modal modes from the 7th order to the 12th order are similar, which appear to be radial bend of the ejector nozzle tube with a frequency range of 371.1 Hz to 375.5 Hz.



(a) 1st order mode

(b) 7th order mode

Fig.2 Mode shapes of ejector suction chamber

It can be seen that compared with the suction chamber shell, the nozzles and their support plates have lower structural stiffness, which are easier to generate larger structural vibration by airflow load.

2 Numerical Simulations of Flow Phenomena in Inner Flow Field

Simulating the flow phenomena in the flow field and extracting the airflow loads are the essential precondition for calculating the vibration response of the injector structure. Here the following issues should be taken account of:

(1) The supersonic flow with high inverse pressure gradient inevitably yields the strong shock wave structure, and the numerical format used to capture strong shock wave generally has a large numerical viscosity. Thus, in order to reduce the influence of numerical viscosity, the numerical format applied in the flow simulation should have the self-regulating ability of numerical viscosity.

(2) Supersonic flow has significant compressible and turbulent effects, and also has some backflows appearing in local areas. Therefore, a suitable turbulence model should be selected to accurately simulate the mutual interference of turbulent mixing layer, turbulence/shock layer and mixing layer.

(3) Two mixed airflows have different speeds, temperatures and densities. Moreover, the shock wave structure is small in scale and has strong viscous interference. Hence, it is imperative to build large-scale computing grid and employ high-efficient parallel processing technology.

(4) During the calculation process, the wave structures in the inner flow field are constantly varying. Meanwhile, compression waves or expansion waves would reflect at the boundary. As a result, the iterative convergence of the flow field may be quite slow. Therefore, for improving the simulation efficiency, it is necessary to choose the efficient method and set the reasonable boundary condition.

The control equations for the inner flow field of the ejector include:

Quality conservation equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho u_i\right) = 0 \tag{3}$$

Momentum conservation equation

$$\frac{\partial}{\partial x_i}\left(\rho u_i\right) + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial t_{ij}}{\partial x_j} \tag{4}$$

$$\tau_{ij} = \left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] - \frac{2}{3}\mu\frac{\partial u_l}{\partial x_l}\delta_{ij} \tag{5}$$

Energy conservation equation

$$\frac{\partial}{\partial t}\left(\rho E\right) + \frac{\partial}{\partial x_i}\left(u_i\left(\rho E + p\right)\right) = \frac{\partial}{\partial x_i}\left(k_{\mathrm{eff}}\frac{\partial T}{\partial x_i} + u_j\left(\tau_{ij}\right)_{\mathrm{eff}}\right) \tag{6}$$

Turbulence equation

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho u_i k\right) = \frac{\partial}{\partial x}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right) + S_k \tag{7}$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho u_i \varepsilon\right) = \frac{\partial}{\partial x}\left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right) + S_\varepsilon \tag{8}$$

Equation of state

$$\rho = \frac{P}{RT} \tag{9}$$

where ρ is the density, t the time, x the displacement vector, u the velocity vector, p the pressure, μ the viscosity coefficient, τ_{ij} the stress tensor, E the energy, k_{eff} the effective thermal conductivity, T the temperature, k the turbulent pulsating kinetic energy and ε the dissipation rate.

The computational fluid dynamics software FLUENT, which is developed on the basis of finite volume theory, is adopted to solve the control equations above. Furthermore, the implicit solver built on density base and k-ε RNG turbulence model are selected.

The fluid grid model, which directly determines the computational efficiency and simulation accuracy, is regarded as an important basis for flow field simulation. Here, the grid model of the inner flow field of the ejector is constructed by using three methods, including non-consistent hybrid grid (NHG), consistent hybrid grid (CHG) and Cutcell. The independence verification of these fluid grids is implemented and Table 1 lists the comparison results.

Table 1 Independence verification of fluid grid

Grid	A	B	C	D	E
Grid type	NHG	NHG	CHG	Cutcell	Cutcell
Number/ 10^6	2.78	16.38	3.16	8.91	11.74
Outlet flow quantity/ $(\mathrm{kg \cdot s^{-1}})$	9.313	9.473	9.739	9.549	9.567
Stability	Bad	Normal	Bad	Good	Good
Iterative step	5 000	3 000	4 000	1 400	1 000

In the NHG model, the nozzles and their plates are meshed with unstructured grids, while the other areas are modeled with structural grids. The nodes at the interface between these two types of grid are not one-to-one matched. Whereas, in the CHG model, the nodes on the grid interface are one-to-one. As for the Cutcell grid model, both the main area and the boundary layer area are built with Cartesian grids and some prismatic grids are created for transition at the grid interface[15].

As known from Table 1, the Cutcell grid model has significant advantages in comparison to the hybrid grid model, such as computational stability, precision, and convergence speed. Thus, the grid model E shown in Fig.3 is selected to simulate and explore the flow field in the ejector.

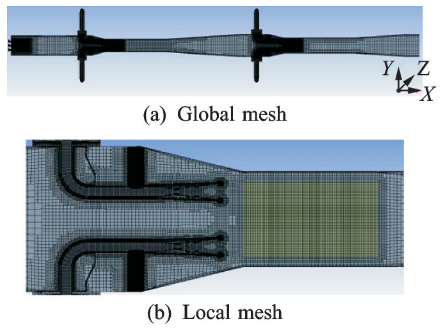


Fig.3 Fluid grid of ejector inner fluid field

After that, the boundary conditions of the flow field are set according to the actual operating parameters, in which the total pressure is 0.9 MPa at the 1st stage ejector inlet, 1 MPa at the 2nd stage ejector inlet, and 101.5 kPa at the ejector inlet. Besides, the static pressure at the ejector outlet is set at 101.5 kPa.

Finally, the FLUENT solver is started to calculate the transient flow characteristics of the flow field. Some results are shown in Figs.4—6.

Fig. 4 shows the distribution of static pressure in the flow field, which indicates that a large quantity of gas in the main area is taken away under the action of high-pressure airflow, resulting in a significant pressure drop. For instance, the pressure is as low as about 0.7 kPa in the area near the venture nozzles, and also is reduced to about 10 kPa and 30 kPa separately in the upper area of the 1st stage ejector nozzles and the 2nd ejector nozzles.

Fig. 5 shows the global and local contours of the Mach number distribution of the flow field. It can be clearly seen that the primary airflow of the 1st stage and 2nd stage ejector is accelerated to supersonic speed through the ejector nozzles, and the

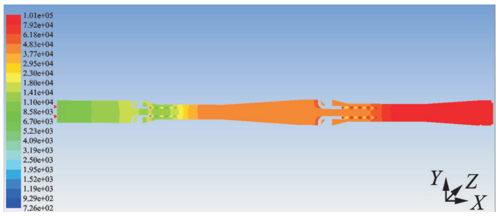
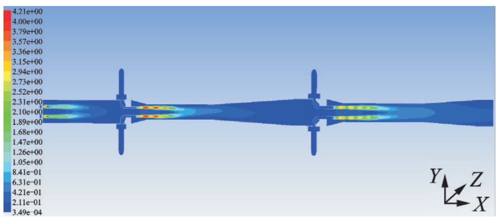
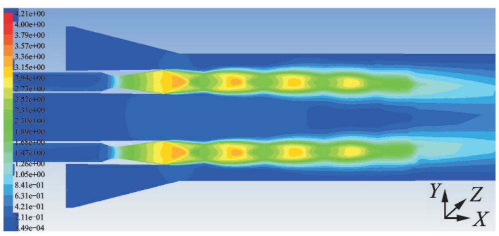


Fig.4 Static pressure contour of ejector inner fluid field

Mach number is about 4.2 and 3.5, respectively. In addition, the shock wave structures appear in the exit area of the ejector nozzles obviously.



(a) Global contour



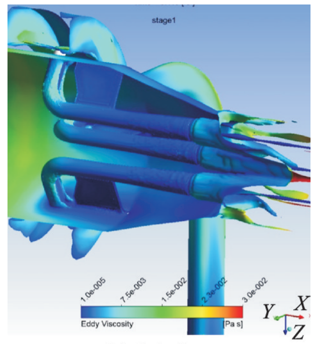
(b) Local contour

Fig.5 Mach number contour of ejector inner fluid field

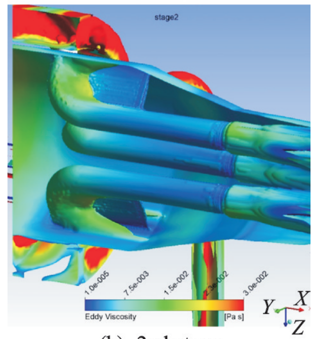
The vortex distribution in the flow field near the ejector nozzles, as shown in Fig. 6, illustrates that the vortex oscillations occur distinctly in this area.

The above simulation results suggest that after a series of processes such as ultrasonic acceleration, turbulent mixing, and pressurization discharge, the complex unsteady flow field, in which strong shock waves and vortex oscillations emerge simultaneously, has been established inside the ejector. This would cause the pressure fluctuation in the flow field.

The frequency spectrum curve of the lift coefficient near the nozzle support plates is worked out by Fast Fourier Transform, as shown in Fig. 7. It can be concluded that the pulsating pressure of the airflow is a type of broadband load, with a maximum peak near 40 Hz and a second peak near 140 Hz. Moreover, the pulsating pressure amplitude fluctuates slightly and changes little after 200 Hz.



(a) 1st stage



(b) 2nd stage

Fig.6 Eddy contour of ejector inner fluid field

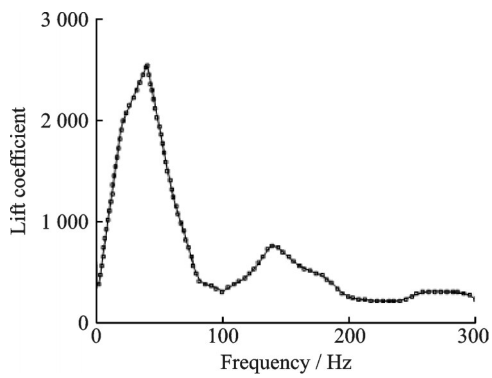


Fig.7 Frequency spectrum curve of lift coefficient in ejector

3 Flow-Induced Vibration Simulation

The flow-induced vibration problem of the ejector belongs to fluid-solid coupling problem. Generally, fluid-solid coupling is either one-way coupling or two-way coupling. When the structure deformation induced by the fluid pulsation is too little to have a significant impact on the flow field, it can be regarded as one-way coupling. Otherwise, it should be taken as two-way coupling. The structural dynamics results narrated in Section 1 suggest that the structural stiffness of the ejector is relatively high. Consequently, the ejector structure would not produce large deformation and not affect the flow field to a great extent. Thus, the one-way coupling technique is employed properly for analyzing the flow-induced vibration of the ejector.

The numerical simulation of fluid-solid coupling not only involves the structural dynamics simulation and the fluid dynamics simulation, but also needs to realize the data exchange between the fluid domain and the solid domain. Fig. 8 describes the general simulation flow of the one-way fluid-solid coupling.

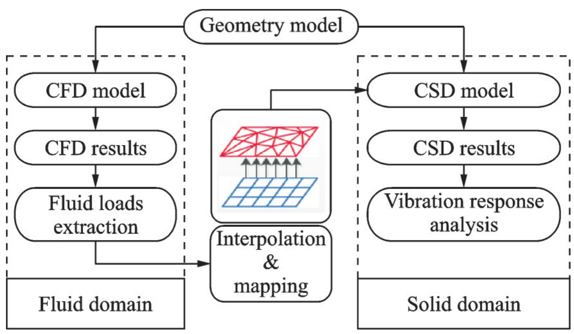


Fig.8 Simulation procedure of one-way coupling problem

Firstly, according to the geometric model of the research object, the computational fluid dynamics (CFD) model and the computational structural dynamics (CSD) model are established respectively.

Secondly, the fluid dynamics calculation is launched based on the CFD model and the pressure information of each node on the domain interface is extracted from the results.

Thirdly, with the interpolation algorithm the pressure information of each fluid node is mapped to the corresponding structure node as its pressure load.

Finally, based on the CSD model structural dynamics calculation is carried out to get the vibration response of the structure.

Following the simulation flow above, the MpCCI software is applied on the high-performance computer cluster to effectively complete the fluid-solid coupling simulation of the ejector, combining with the CFD model and the CSD model established in the previous sections. Some simulation results are shown in Figs.9—11.

Fig. 9 presents the pressure distribution in the coupling region of the fluid domain and the solid domain at the same time, which demonstrates that the pressure distribution in these two domains is so consistent that the error is less than 1%. Thus, the interpolation algorithm adopted in this work is precise enough to accurately map the aerodynamic load to the structure grid.

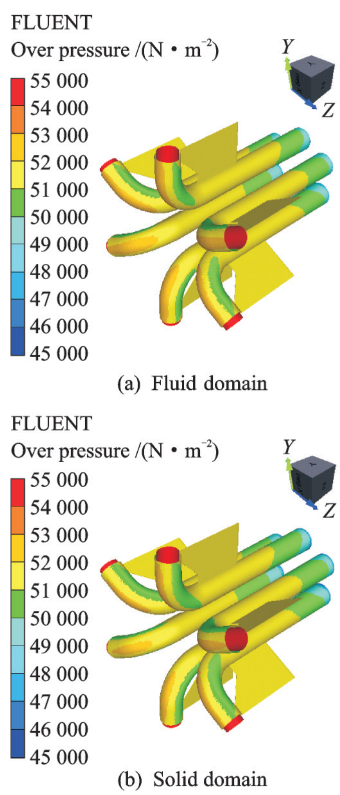
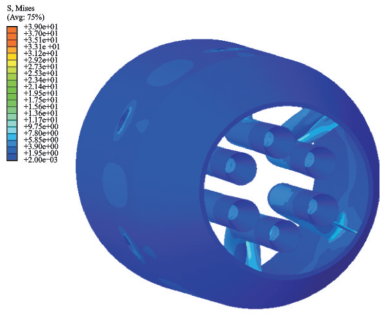


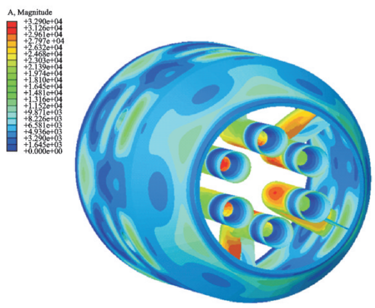
Fig.9 Pressure contour of fluid-solid coupling field

The transient structural response contours of the ejector are shown in Fig. 10, including equivalent stress and acceleration. It can be seen that the maximum equivalent stress is about 39 MPa, which appears at the conjunction area between the support board and the chamber wall. On the other hand, the maximum acceleration is about 33 m/s^2, which occurs in the middle of the ejector nozzle.

The acceleration response of monitoring points arranged on the support plates is extracted from the simulation results, and then is processed by fast Fourier transform to obtain its acceleration spectrum curve, as shown in Fig. 11. It can be found that there are three distinct peaks in the spectrum curve, which are respectively about 0.011g, 0.065g, 0.122g, corresponding to frequencies of approxi-



(a) Equivalent stress



(b) Acceleration

Fig.10 Transient response contour of ejector structural vibration

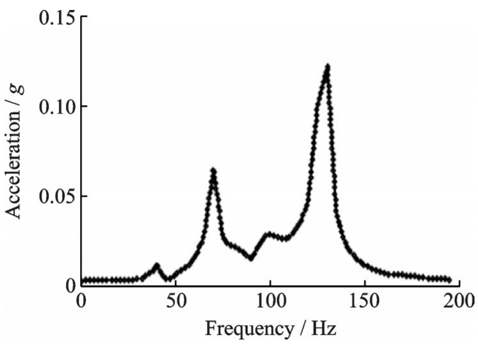
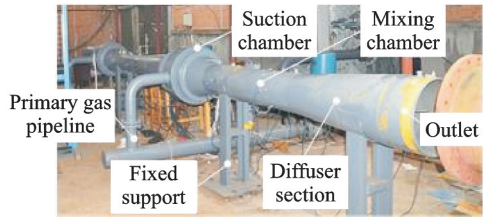


Fig.11 Acceleration frequency spectrum curve of monitoring point

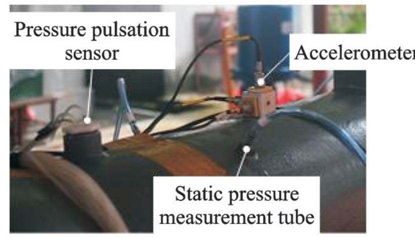
mately 40, 70,130 Hz.

4 Structural Vibration Test

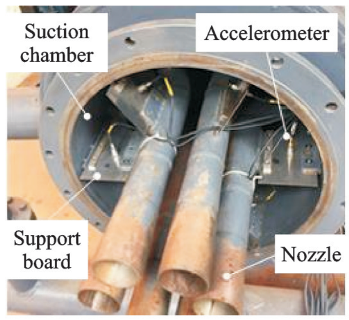
According to the 3D structure model shown in Fig. 1, a set of ejector structure system including pipelines is actually developed, as shown in Fig. 12 (a). Additionally, A variety of sensors, as shown in Fig. 12 (b), are deployed to construct an ejector vibration test platform for monitoring the variation of flow field parameters and measuring the vibration response of the ejector structure. Moreover, the test results can be used to verify the accuracy of the simulation method. Some monitoring points are arranged on the support plates, as shown in Fig.12(c).



(a) Ejector structure



(b) Sensors



(c) Monitoring point

Fig.12 Vibration test platform of ejector structure

The selected sensors mainly include:

(1) Static pressure measurement tubes, which are arranged along the main pipeline of the ejector, as shown in Fig.12(b), for real-time static pressure measurement in the mainstream area.

(2) Pressure pulsation sensors, which are arranged along the main pipeline of the ejector, as shown in Fig. 12(b), for real-time pressure pulsation measurement in the inner flow field.

(3) Vibration accelerometers are, which arranged on the main pipe of the ejector as well as the support plates of the nozzle, as shown in Fig. 12 (c), for real-time recording of the vibration acceleration.

The two-stage ejector system is started and operated by controlling the valves set on the ejector pipelines. In the meantime, the structural vibration tests are carried out and some results are obtained and plotted in Figs.13—15.

The test data and simulation results, as shown in Fig. 13, illustrate the static pressure distribution along the main pipeline of the ejector. It can be discovered that the static pressure escalates from the inlet to the outlet except in the front section of the 2nd mixing chamber, where the static pressure drops slightly. And besides, the simulation results are relatively large.

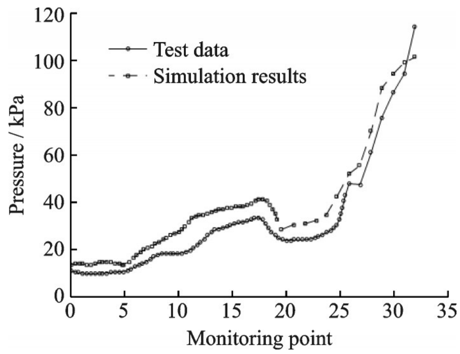


Fig.13 Static pressure distribution of ejector fluid field

The test results of the flow field pressure pulsation are processed and shown in Fig.14. It is obvious that there are two peaks in the spectrum curve, which are respectively about 0.9 kPa and 0.37 kPa, corresponding to frequencies of about 30 Hz and 120 Hz.

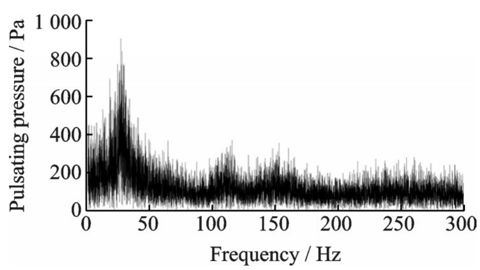


Fig.14 Test results of ejector pulse pressure

By comparison with the results in Fig.7, it can draw the conclusion that the test data and the simulation results have a good consistency in most aspects, such as peak distribution, corresponding frequency and amplitude variation. Obviously, the numerical simulation technique adopted in this work has a high feasibility, which can efficiently analyze the flow field characteristics of the ejector and accurately reflect the complex variation of the aerodynamics load.

The test results of the vibration acceleration of the nozzle support board are processed and described in Fig.15. It can be seen that the peak value is about $0.096g$, appearing near 125 Hz, and the sub-peak value is about $0.043g$, corresponding to frequency of about 80 Hz. In addition, the acceleration value is about $0.032g$ while the frequency is about 30 Hz. Comparing the test data with the simulation results, the maximum error is calculated, which is about 27%.

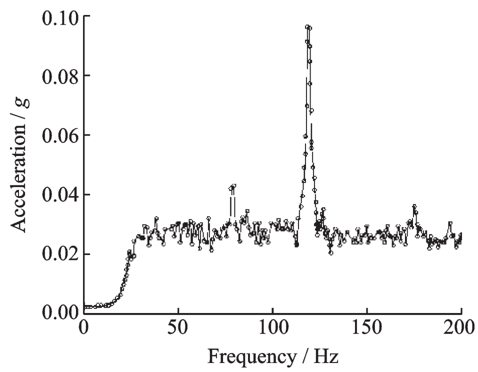


Fig.15 Acceleration test results of support plate

Through summarizing the analysis results of the numerical simulation and the vibration tests, it can be found that the pulsating pressure reaches the largest value at the frequency of about 30 Hz and the corresponding vibration acceleration is about $0.032g$. On the other hand, the pulsating pressure is the second largest at the frequency of about 120 Hz, but the vibration acceleration is the largest, about $0.1g$. The main reason is because the aero load frequency is very close to the modal frequency of the suction chamber structure. That is, the vibration of the support plates is the modal vibration of the structure induced by the pulsating load of the airflow.

5 Conclusions

Based on the methods of computational structural dynamics, computational fluid dynamics and fluid-solid coupling algorithm, several simulation techniques are employed to accomplish the structural dynamics analysis of a two-stage ejector, the flow simulation of inner flow field, and the calculation of structural vibration response under airflow excitation. Meanwhile, ejector vibration tests are performed and studied to verify the simulation results. According to the research achievements, some conclusions can be drawn as follows:

(1) The CFD model established in this work is accurate and efficient, which can comprehensively describe the complex unsteady flow phenomena in the inner flow field of the ejector, such as shock waves and eddy currents, and also can figure out the pulsating pressure of the airflow that is well consistent with the test data. Consequently, the external loading environment of the ejector structure is clarified through these studies.

(2) The precise mapping of pressure load from fluid domain to structural domain is realized in the fluid-solid coupling model. As a result, the frequency spectrum of vibration acceleration response obtained through the coupling simulation well matches the test data, which can reveal the flow-induced vibration characteristics of the ejector structure.

(3) Both numerical simulations and vibration tests demonstrate that larger structural vibration will happen as long as the frequency of the pulsating airflow is close to the modal frequency of the structure. In order to avoid this case, it is necessary to optimize the structure mode of the ejector when designing.

In summary, the numerical simulation methods adopted in this paper can be used to explore the internal mechanism and external characteristics of the flow-induced vibration of the ejector. Moreover, the corresponding simulation results can be provided as a significant reference for the structure optimization and vibration control of the ejector. Thus, these simulation methods will be tenable in engineering application.

References

[1] ZHAO J Y, SUN H J. Basic principles and applications of ejector [J]. Journal of Beijing Institute of Civil Engineering and Arc, 2000, 16(4): 12-15.

[2] KNOWLES K, SADDINGTON A J. A review of jet mixing enhancement for aircraft propulsion application [J]. Journal of Aerospace Engineering, 2006, 220: 103-127.

[3] LIAO D X, CHEN J M, YU Y S. Experimental study on the performance of mixing-enhancing nozzles in ejectors [J]. Journal of Experiments in Fluid Mechanics, 2007, 21(3): 24-29.

[4] LIAO D X, REN Z B, YU Y S, et al. Design and experiment of constant-pressure mixing ejector [J]. High Power Laser and Particle Beams, 2006, 18(5): 728-732.

[5] BLEVINS R D. Flow-induced vibration[M]. New York: Van Nostrand Reinhold Co., 1977.

[6] GOYDER H G D. Flow-induced vibration in heat exchangers [J]. Chemical Engineering Research and Design, 2002, 80(3): 226-232.

[7] ZDRAVKOVICH M M. The effects of interference between circular cylinders in cross flow [J]. Journal of Fluids and Structures, 1987, 1(2): 239-261.

[8] SUMNER D. Two circular cylinders in cross-flow: A review [J]. Journal of Fluids and Structures, 2010, 26(6): 849-899.

[9] LI T, ZHANG J Y, LI Z J, et al. Co-simulation on fluid-structure interaction of high-speed train based on Fluent and Simpack [J]. Chinese Journal of Computational Mechanics, 2012, 29(5): 675-680.

[10] SHI Y Q, WANG Q Q, YANG Q Z. Analysis of fluid-structure coupling characteristics in fan/ compressor rotor [J]. Computer Simulation, 2011, 28(7): 115-119.

[11] YU Y, GUO X M, JIN T J, et al. Fluid-structure interaction simulation of half-sheet structure under wind load based on S-A turbulence model [J]. Chinese Journal of Computational Mechanics, 2013, 30: 155-158.

[12] WEI S H, ZHANG L J, YAN Y Z. Analysis of draft tube vibration induced by tailrace eddy based on a fluid-structure coupling numerical method [J]. Water Power, 2013, 25(5): 90-92.

[13] XIAO R F, WANG Z W, LUO Y Y. Stress analysis of Francis turbine runners based on FSI [J]. Journal of Hydroelectric Engineering, 2007, 26(3): 120-123.

[14] FARHAT C, LESOINNE M, MAMAN N. Mixed explicit/implicit time integration of coupled aeroelastic problems: Three-field formulation, geometric conservation and distributed solution [J]. International Journal for Numerical Methods in Fluids, 1995, 21(10): 807-835.

[15] SANG W M, LI F W. Numerical analysis of flows around 3-D high-lift system by adaptive Cartesian grid method [J]. Acta Mechanica Sinica, 2005, 37(1): 80-86.

Acknowledgements This work was supported in part by the National Natural Science Foundation of China (Nos.

51806234, 51805530).

Authors Associate Prof. **NIE Xutao** received the B.S. and Ph.D. degrees in mechanical engineering from National University of Defense Technology, Changsha, China, in 2000 and 2008, respectively. He joined in China Aerodynamics Research & Development Center (CARDC) in April 2008, where he is an associate professor of Facility Design and Instrumentation Institute (FDII). In 2014, he worked as a visiting scholar in School of Engineering, Cranfield University, UK. His research has focused on numerical simulation, fluid-solid coupling analysis and structural vibration test.

Researcher **LIU Zongzheng** received the B.S. and Ph.D. degrees in mechanical engineering from Wuhan University of Technology and Tsinghua University, in 1991 and 2012, respectively. He joined CARDC in August 1991 and worked as a researcher from 2012. His research is focused on wind tunnel mechanical design, flow-induced vibration analysis and structural vibration control technology.

Dr. **QIN Chaojin** received the B.S. and Ph.D. degrees in aerospace engineering from Beijing University of Aeronautics and Astronautics, in 2005 and 2012, respectively. He joined Chengdu Star-Union Ltd. as CTO in 2016. His research is focused on flow-induced vibration simulation, hypersonic aircraft and relevant fields.

Mr. **ZHANG Wei** received the B.S. and Master degrees in refrigeration and cryogenic engineering from Xi'an Jiaotong University and Zhejiang University, in 2011 and 2014, respectively. He joined CARDC as an engineer in 2014 and his research is focused on unsteady flow numerical simulation and cryogenic phase change analysis.

Author contributions Associate Prof. **NIE Xutao** carried out the research work and wrote the manuscript. Researcher **LIU Zongzheng** designed the study. Dr. **QIN Chaojin** was responsible for the flow-induced vibration numerical simulation. Mr. **ZHANG Wei** assisted Dr. Qin to complete the CFD simulations of the ejector. All authors commented on the manuscript draft and approved the submission.

Competing interests The authors declare no competing interests.

(Production Editor: Sun Jing)