Int J Adv Manuf Technol (2014) 71:1087–1092
DOI 10.1007/s00170-013-5493-9

ORIGINAL ARTICLE

Numerical simulation ultrahigh waterjet (WJ) flow field with the high-frequency velocity vibration at the nozzle inlet

Rongguo Hou · Chuanzhen Huang · Hongtao Zhu

Received: 25 April 2013 / Accepted: 14 November 2013 / Published online: 21 December 2013

Abstract This paper presents an investigation of ultrahigh pressure waterjet (WJ) flow field with the high-frequency velocity vibration at the nozzle inlet by computational fluid dynamics method. The velocity field of the flow inside the WJ nozzle is obtained. The influence of vibration parameters, such as amplitude and frequency on the flow field are studied. The results of investigation indicate that the flow velocity at the WJ nozzle outlet has almost the same vibrating type with the inlet velocity vibration. During a vibration cycle, the velocity field of the flow is changing greatly. The serials value of the vibration frequency and amplitude are taken to test their influence on the flow field. The simulation results show that the flow field also changes greatly with the different frequency and amplitude. Based on the obtained results, the mechanism of system pressure vibration influence on the water jet flow field inside the WJ nozzle is obtained, and a new method is provided to optimize the machining process with the aim to improve the machine efficiency and surface quality of the work piece.

Keywords High pressure waterjet (WJ) · Computed fluid dynamic (CFD) · Flow field · Velocity vibration

R. Hou · C. Huang (✉) · H. Zhu
Key Laboratory of High-efficiency and Clean Mechanical Manufacture, Ministry of Education, Center for Advanced Jet Engineering Technologies (CAJET), School of Mechanical Engineering, Shandong University, Jinan 250061, China
e-mail: chuanzhenh@sdu.edu.cn

R. Hou
e-mail: hourongguo212@163.com

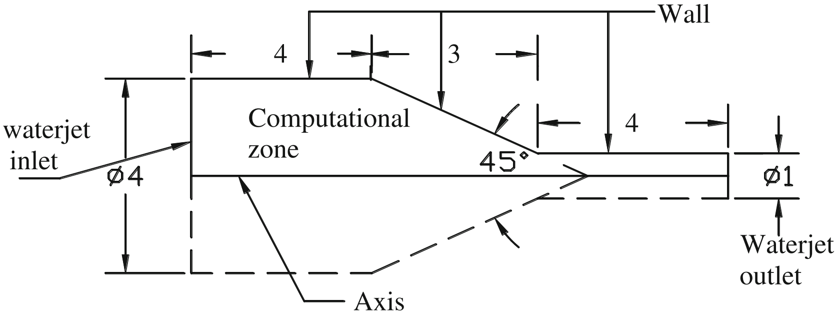
H. Zhu
e-mail: htzhu@sdu.edu.cn

R. Hou
School of Mechanical Engineering,
Shandong University of Technology, Zibo 255049, China

1 Introduction

Ultrahigh pressure waterjet (WJ) machining, as one of the most recent nontraditional method, is widely used in industry to machine materials. It is often used to cut a variety of soft/light materials including paper, food, fiberglass insulation, some plastics, and composites [1–3]. As for the ultrahigh pressure WJ machining technology, the cutting performance mainly depends on supersonic erosion process and its high impact force. However, this technology should be paid more concern because ultrahigh pressure WJ has its special characteristics including high velocity, and high turbulence; its many aspects are yet to be fully understood. In the aspect of theory research, some proposed theory and laws have not been fully accepted, and most of them are based on too many hypotheses such as time mean and special grain shape [4]; their application is limited in studying the complicated unsteady turbulence flow. When using experiment method to analyze the flow field, some measures cannot be used because of the flow's high velocity and high pressure. Recently, with the development of computer and computational mathematics, the computational fluid dynamics (CFD) software such as fluent Adina, CFX, etc. are widely used to analysis the fluid field. Many researchers have used the CFD software to solve various problems which cannot be solved by theoretical and experimental methods [5–8]. In the case of WJ technology, the simulation for the flow velocity field of the gas–liquid two-phase flow and the gas–liquid–solid three phase flow inside and outside the abrasive water jet nozzle has been carried out [9, 10]. Liu et al. [11] has used the FLUENT to simulate the jet dynamic characters for the two-phase and three-phase flow downstream from a very fine nozzle; his simulation result provide an insight into the jet characters. Gong et al. [12] have used the ALE and LY-DYNA

Fig. 1 The physic model of the WJ nozzle



softwares with an aim to simulation the abrasive water jet progress. Their simulation results have a good agreement with the experiment result. However, research on WJ machining is far to arrive at a comprehensive understanding, such as the system pressure is often vibrating before entering into the nozzle, which makes the velocity out of the nozzle changing and which causes the low machine efficient and poor surface quality reliability [13]. On the other hand, this velocity vibration at the outlet could be used to improve the machine process. This work presents a study of ultrahigh pressure WJ flow field with the high-frequency inlet velocity vibration by CFD method. It has an aim to investigate influence of the vibration parameters, such as amplitude and frequency on the flow field. Based on the above research, the results help to study the affection of system vibration on the water jet flow field inside the WJ nozzle and to find a new method to optimize the machining process.

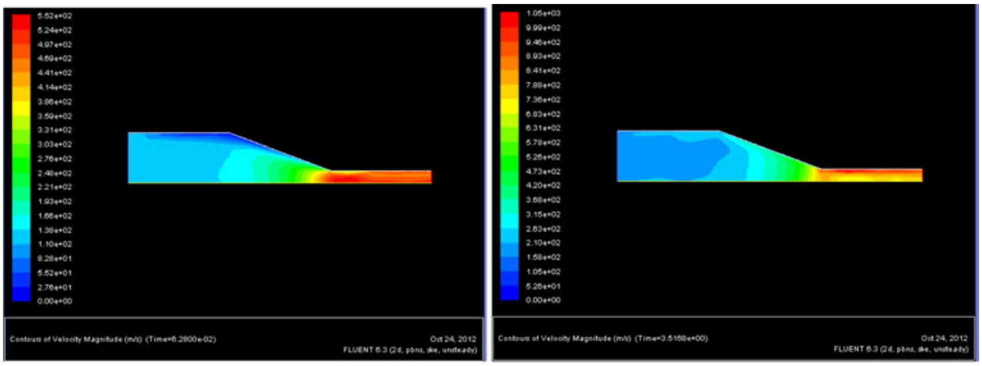
2 Governing equations

The mass, momentum, and energy equations are often used to analyze the structure and field of the turbulence flow. While as for the WJ flow, there is no heat deformation, so the energy equation is omitted.

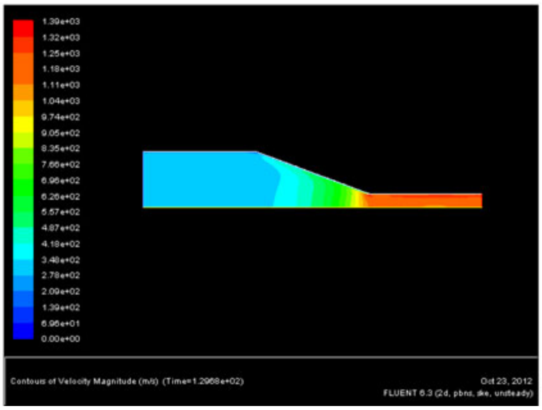
The continuum equation is

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho \nu_j)}{\partial x_i} = 0 \tag{1}$$

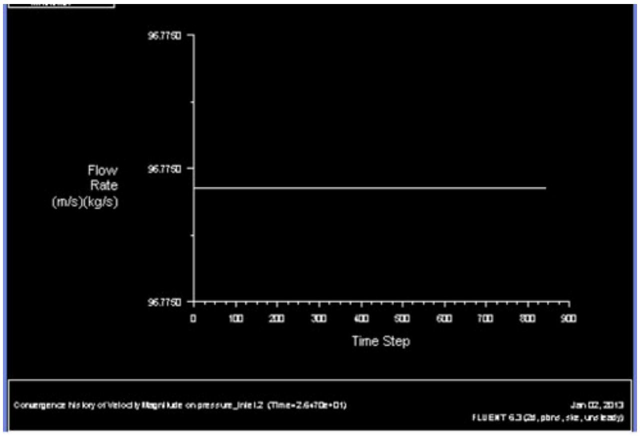
Fig. 2 The velocity field of the WJ inside and outside the nozzle (amplitude is 80 m/s, frequency is 1,000Hz)



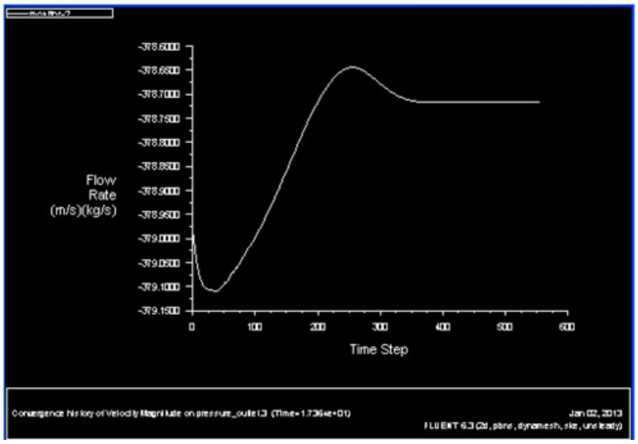
(a) Start time (b) Middle time



(c) End time



(a) The flow rate value at the inlet



(b) The flow rate value at the outlet

Fig. 3 The flow rate value at the inlet and outlet at the different time (no vibration)

The momentum equation is

$$\frac{\partial(\rho v_i)}{\partial t} + \text{div}\left(\rho v_i \vec{v}\right) = \rho\left(\frac{\partial v_i}{\partial t} + \vec{v}\cdot\text{grad}v_i\right) \tag{2}$$

Ultrahigh pressure WJ is a typical high turbulence flow. It has a character of unsteady and high pressure [14]. In fluent, the standard $K-\varepsilon$ turbulence equations are used as the government equations [15], and the transport equations for the turbulence energy k and dissipation rate ε are

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho v_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\frac{\mu_e \, \partial k}{\varepsilon_k \, \partial x_j}\right] + P - \rho\varepsilon \tag{3}$$

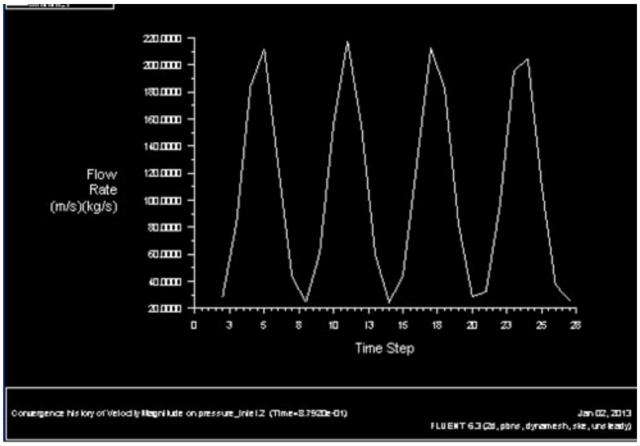
$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho v_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\frac{\mu_e \, \partial\varepsilon}{\varepsilon_e \, \partial x_j}\right] + \frac{\varepsilon}{k}\left(c_{1\varepsilon}P - \rho c_{2\varepsilon}\varepsilon\right) + c_{3\varepsilon}\frac{\varepsilon}{k}P \tag{4}$$

Where v is the vector of the velocity (m/s), P is the flow pressure (Pa), x_i is the component of i direction, k is the turbulence energy, ε is the dissipation rate, ρ is the density (kg/m^3), and μ_e is the viscosity. $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are the constants. $C_{1\varepsilon}=1.44$, $C_{2\varepsilon}=1.9$, $C_{3\varepsilon}=0.09$.

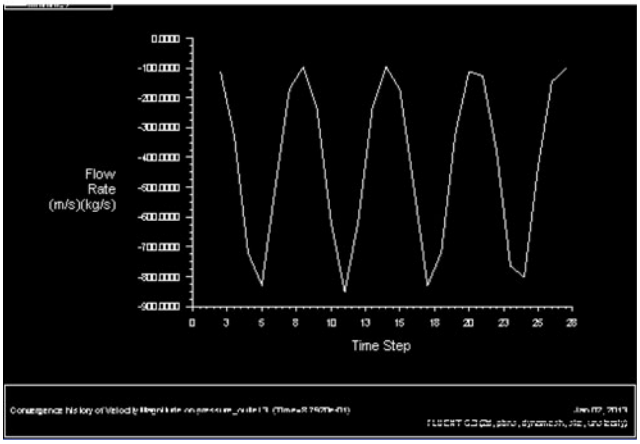
3 Physical models and solution setting

3.1 Physical model

The geometry of the computational domain with boundary conditions is shown in Fig. 1. Ultrahigh pressure water jet is considered as a two-dimensional flow. The unsteady turbulent flow enters the nozzle from the left side, and it enters the atmosphere as a free jet after passing through the WJ nozzle. The size of the geometry model is as the following: the diameter of WJ nozzle inlet is 4 mm, its length is 4 mm, and the inner core angle is 45° where the area of the nozzle section is narrowing. The diameter of nozzle at the outlet is 1 mm and its length is 4 mm.

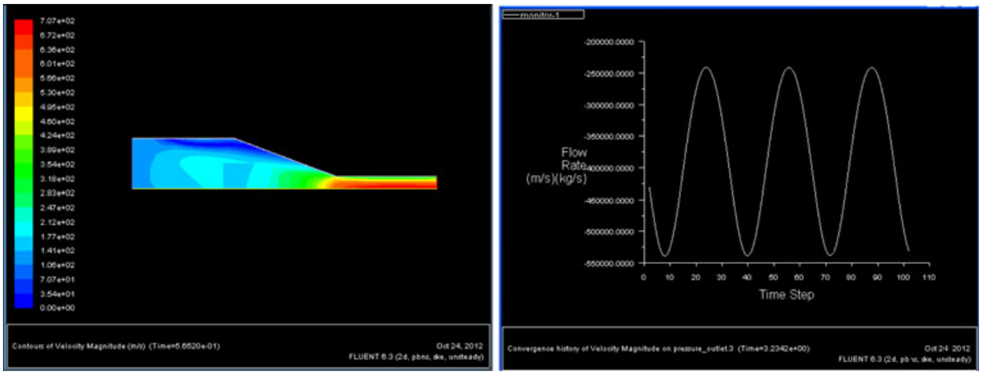


(a) The flow rate at the inlet



(b) The flow rate at the outlet

Fig. 4 The flow rate changing at the inlet and outlet (amplitude is 40 %, the frequency is 1,000 HZ)



(a) (b)

Fig. 5 The velocity field inside the nozzle and the flow rate changing at the outlet with 10 HZ frequency

3.2 Solution setting

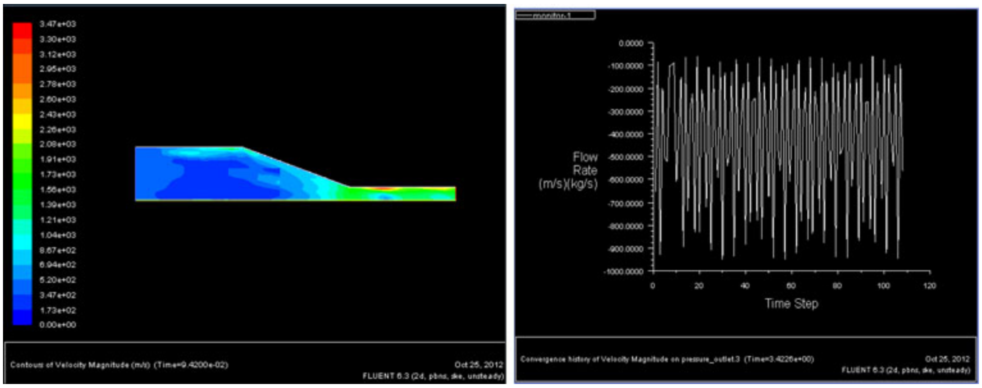
The boundary conditions are set as shown in Fig. 1. The left edge of the geometry is the velocity inlet of WJ where the water jet enters into the nozzle with an initial value of 200 m/s; its value is varying as the sin module shows in Eq. 5.

The velocity of the flow at the inlet is

$$\mathrm{Vt} = V_0 + A\sin(\omega \mathrm{t} + \varphi) \qquad (5)$$

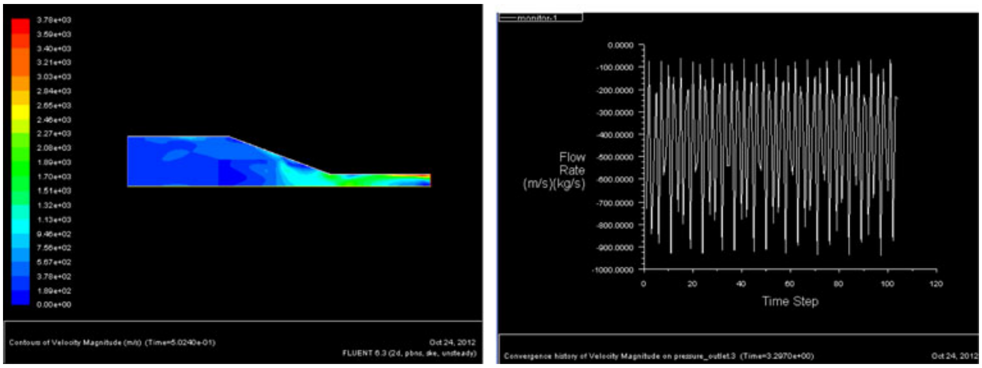
Where A is the amplitude (m/s), ω is the circular frequency of the inlet velocity, φ is the phase angle (rad), and V_0 is the constant (m/s).

The main vibration parameters such as the frequency and amplitude are as these: the frequency values are 1, 10, 100, 1,000, 10,000, and 100,000 HZ; they are varying from the low frequency to the high frequency, and its amplitude values are 40, 80, 120, 160, and 200 m/s. The right side is the pressure outlet with a value of 1.01325e+5Pa [9].



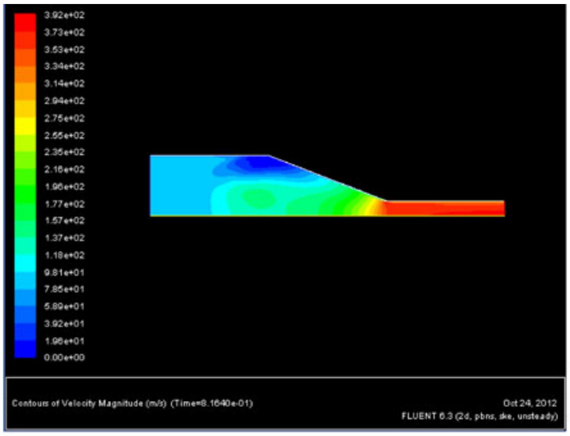
(a) (b)

Fig. 6 The velocity field inside the nozzle and the flow rate changing at the outlet with 1,000 HZ frequency

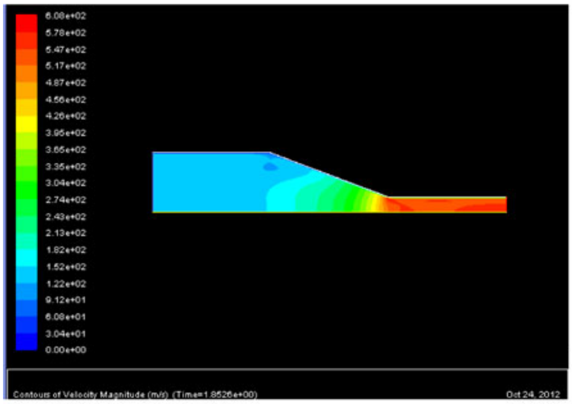


(a) (b)

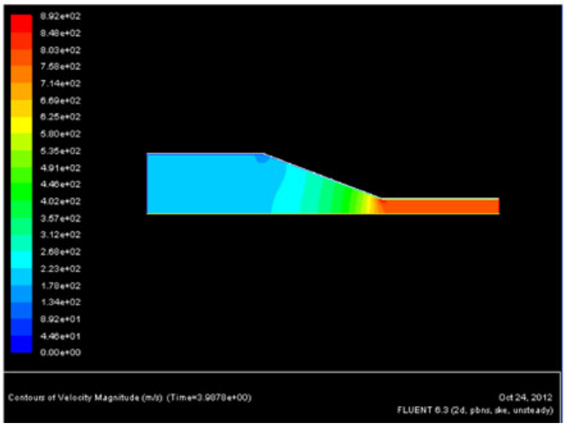
Fig. 7 The velocity field inside the nozzle and the flow rate changing at the outlet with 100,000 frequency



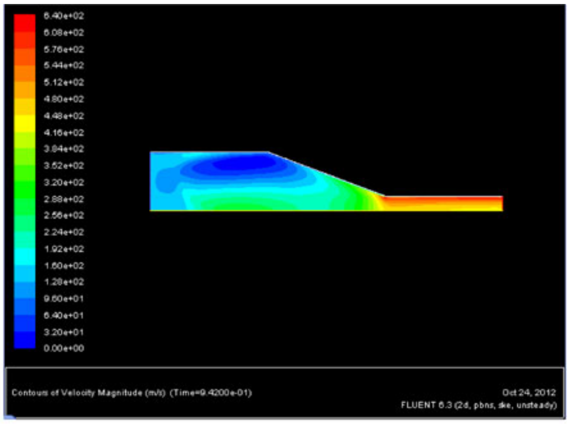
(a) Amplitude of the velocity is 40m/s



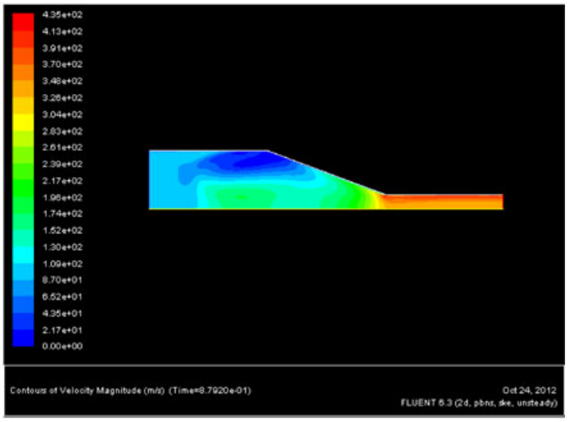
(b) Amplitude of the velocity is 80m/s



(c) Amplitude of the velocity is 120m/s



(d) Amplitude of the velocity is 160m/s



(e) Amplitude of the velocity is 200m/s a

Fig. 8 The velocity field at the outlet with different amplitude (frequency is 10 HZ)

As for the unsteady flow, the convergence is not easily obtained during the computer solution. To investigate the accuracy, stability, and convergence properties of the CFD model, a number of initial computational tests [10] were performed using the first upwind method and the second upwind method. As for the model divided by the triangle cell, the second upwind method has the more stability, rapid convergence, and accurate simulate result. Thus, the second upwind method is used to discretize the model. The time step size and number of time steps are the key factors to gain the rapidly convergence; their values are set as 0.0314 and 10 after several test. Other parameters are default. The relaxing number is selected according to the converging condition. When the iteration is converged, the simulated results are obtained [11].

4 Simulation result and its analysis

The simulation results of Fig. 2a–c are the flow field of the WJ inside the nozzle at the different time in one cycle, and they show that the flow field demonstrates periodic fluctuates. Figure 2a is the velocity field at the start time. The result shows that the velocity is increasing when the section area decreased and the velocity at the focus tube is the maximum value. Figure 2b is the velocity field at the middle time compared to Fig. 2a; the velocity becomes low, and the flow rate is reduced rapidly. Figure 2c is the velocity field at the end time; its velocity field is almost back to Fig. 2a.

The simulation results of Fig. 3a, b are the flow rate at the inlet and outlet with the condition of no vibration. It shows that the flow rate at the inlet is constant as Fig. 3a shows, and as shown in Fig. 3b, the flow rate at the outlet is waving firstly then it tends to be a constant.

The simulation results of Fig. 4a, b are the flow rate at the inlet and outlet with the condition of vibration, and the flow rate value at the inlet varies as the type of sine modular, while the flow rate at the outlet is also varying accordingly with the sine modular, and the high flow rates expressed complete the contrary at the inlet and outlet. The change of the flow rate at the outlet is amplified several times than at the inlet.

The simulation results of Figs. 5, 6, and 7 are the velocity field inside the nozzle and the velocity value at the outlet with difference frequency, and in Fig. 5 are the velocity field inside the nozzle and flow rate with the vibration frequency of 10 HZ at the inlet. Figure 6 results are the velocity field inside the nozzle and flow rate with the vibration frequency of 1,000 HZ at the inlet, and Fig. 7 results are the flow field inside the nozzle and flow rate with the vibration frequency of 100, 000 HZ at the inlet. Comparing these results, the flow rate at the outlet is vibrating with the corresponding vibration frequency, and the reason is that the flow rate is decreasing with the increase of the vibration frequency.

The simulation results of Fig. 8a–e are the velocity field inside the nozzle with deferent amplitudes. The vibration amplitudes are varying from 40 to 200 m/s. They show the different velocity fields. The flow rate tends to decease with the increasing of the vibration amplitude.

All of the above, the flow field expressed the significant changes at the different time in one cycle; the flow velocity at the nozzle outlet is almost the same vibrating with the inlet velocity vibration. The frequency and amplitude of the vibration influent the flow field greatly with the increasing of the frequency and amplitude of the vibration, the flow field changes, and the flow rate decreases greatly.

5 Conclusion

The study of high waterjet flow field with the condition of inlet velocity vibration is performed by CFD method. The flow field of ultrahigh pressure waterjet is obtained, and the vibration and waterjet parameters are considered their affection on the flow field. The results express that the flow velocity at the outlet of the nozzle is almost the same vibrating with the inlet velocity vibration, the frequency, and amplitude of the vibration effect the flow field greatly. With the increasing of the frequency and amplitude of the vibration, the flow field changed greatly and which results in the decrease of the flow rate. The flow field also indicates that the flow field changes greatly at the deferent time in one vibrating cycle. That is to say it is essential to remain system being steady, which will gain the stable velocity, and it is helpful to gain the high quality surface. However, the proper vibration velocity is also help to increase the machine efficiency. This research helps to study the system vibration on the water jet flow field and optimism the machining process.

Acknowledgments The work is financially supported by the National Natural Science Foundation of China (51075238 and 51175307), Natural Science Foundation of Shandong (ZR2009FQ026), and Shandong Provincial Scientific Research Foundation for Excellent Young Scientists (BS2009ZZ015).

References

1. Park W, Jang J, Chun H, Kim M (2005) Numerical flow and performance analysis of waterjet propulsion system. Ocean Eng 32:1740–1761
2. Hashish M (1989) Pressure effects in abrasive waterjet (AWJ) machining. J Eng Mater Technol ASME 111:221–228
3. Milena K, Michal R, Jan V, Sergej H, Milan K (2012) Determination of technologically optimal factors of modulated waterjet. Int J Adv Manuf Technol 60(1-4):173–179
4. Ni J (1991) The basic theory of solid-liquid two-phase flow and their new application. Technology
5. Mabrouki T (2000) Numerical simulation and experimental study of the interaction between a pure high-velocity water jet and targets: contribution to investigate the decoating process. Wear 239:260–273
6. Jia M (2002) The study and applicant of former mixing abrasive water jet technology. Master Degree, Shanghai University (In Chinese)
7. Li H (2009) Fluid flow analysis of a single stage centrifugal fan with a ported diffuser. Eng Appl Comput Fluid Mech 3(2):147–163
8. Holbeach JW, Davidson MR (2009) An eulerian–eulerian model for the dispersion of a suspension of microscopic particles injected into a quiescent liquid. Eng Appl Comput Fluid Mech 3(1):84–97
9. Hou RG, Huang CZ, Zhu HT, Zhao QZ (2010) Numerical simulation and experimental investigation of the gas-liquid-solid three-phase flow outside of the abrasive water jet nozzle. Key Eng Mater 431–432:90–93
10. Hou RG, Huang CZ, Wang J, Feng YX, Zhu HT (2006) Simulation of velocity field of two-phase flow for gas and liquid in the abrasive water jet–nozzle. Key Eng Mater 315–316:150–153
11. Liu H, Wang J, Kelson N, Brown RJ (2004) A study of abrasive waterjet characteristic by CFD simulation. J Mater Process Technol 153–154:488–493
12. Gong W, Wang J, Gao N (2011) Numerical simulation for abrasive water jet machining based on ALE algorithm. Int J Adv Manuf Technol 53:247–253
13. Nanduri M, David G, Thomas J (2002) The effects of system and geometric parameters on abrasive waterjet nozzle wear. Int J Mach Tool Manuf 42:615–623
14. Vinay S, Somnath C, Sergej H (2011) Multi response optimization of process parameters based on Taguchi-Fuzzy model for coal cutting by water jet technology. Int J Adv Manuf Technol 56(9-12):1019–1025
15. The Fluent Inc (2008) The user guide of the Fluent 6.3