CFD Investigation on Major Factors to Jet Small Dots

Qiu Zhongquan

Abstract

Simulation investigation on major factors, which are affecting jet small dots, is conducted based on computational fluid dynamics (CFD) software. As the market requires high performance and reliable applications, in terms of small beads and fast jetting speed, it requires to eject various fluids, adhesives, and coating, encapsulants and fluxes materials. Under the development of hybrid assembly and surface mount technology (SMT), non–contact fluid jetting process is widely used in electronic components encapsulation instead of traditional contact dispensing application. Typically, a NexJet system which is driven by new precision software control in product assembly is capable to not only jet accurate and consistent bead size, also eliminate unexpected defects such as hammerhead, tails and filaments, which are very common seen during contact dispensing process.

Keywords: computational fluid dynamics (CFD), encapsulants, SMT, bead size

1. Introduction

In recent years, application demands for diversified glues especially in electronics assembly have significantly increased. Before 1990s, the product assembly was absolutely to utilize a lead frame, die bonding, wire bonding and plastic molding process. Printed circuit board (PCB) assembly used wave a soldering device and IC inserters. [1] Nowadays, fluids are becoming integral parts of the electronics package as evidenced by liquid crystal displays (LCDs), biotechnology labs on a chip, lenses and MEMS devices. Therefore, the technology for applying fluids is evolved from non-needle dispensing contact methods to needle jetting non-contact ways. At this point, it was recognized that the glue cartridge could be employed to melt adhesives and force molten liquids to feed into the jetting chamber, where the needle is driven to move back and forth in very high speed and frequency. Normally, the needle speed is increased to 2.0m/sec. Needle movement direction is simultaneously alternated in 300Hertz and above. Typically, the advanced piezoelectric jets in this configuration are capable to work at the cycles reaching to 20 KHz. In general, the pneumatic-mechanical jet device works in a unique manner, and its structure diagram is shown in Fig.1. Basically, filling adhesives are pressurized at 0.3Mpa or above with moderate viscosity material. The pneumatic-mechanical jet is led to lessen flowing out the nozzle under liquid pressure alone and eject dots in terms of nozzle orifice size, needle tip size and seat geometry. The compressed air driven actuator has many benefits to jet materials. Most important advantage is to eject robust dots in electronics assembly such as underfill, epoxy, flux and surface-mount adhesive.

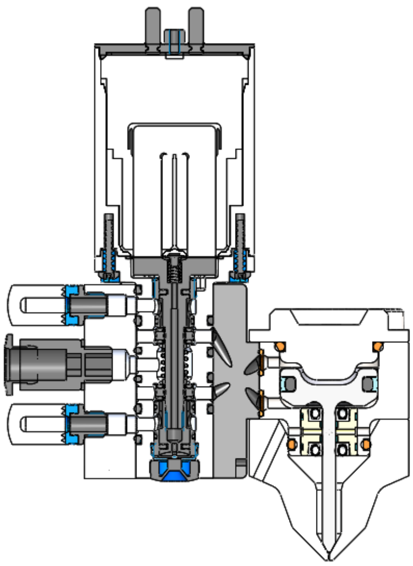


Figure 1: Pneumatic-mechanical diagram of jet device

And almost every kinds of fluid applied in electronics assembly was proven to eject in this technology. Adversely, the dot sizes are much bigger than piezoelectric or thermal inkjets. The average dot sizes are limited to 0.5mm, so the pneumatic-mechanical jetting device is able to obtain. New micro-electronics assembly requires 0.3mm or less dot sizes based on novel application process demands. [2]

In order to eject a small dot to feed up market needs, it is essential to investigate the fluid characteristics of a needle jetting device to determine the major factors, which affect the jetting dot size. The objectives of this research are explained as follows: (1) Perform to explore the fluid

*Corresponding Author: Qiu Zhongquan
(E-mail: peter.qiu@nordson.com)
[1] School of ShangHai jiaotong university Shanghai 200240, P.R. China
[2] Nordson(China) product design and manufacture company

characteristics of needle jetting device based on CFD software;(2)Identify the effects of the jetting dot size in terms of needle tip, nozzle orifice and seat geometry dimensions; (3)Comparison analysis is drawn according to simulation results.

2. Profile and Working Theory

Pneumatic air is applied to refill the needle-seat area. It drives the needle moving away from the seat to allow fluid to flow into the seat area. As the stem rises up, it makes the first chamber volume larger; consequently, fluids are forced to fill into the chamber through an adhesive cartridge supply. The jet nozzle orifice is so small, so fluid hydraulic pressure is great enough to not allow atmosphere air breathing through an orifice. Then the needle stem is moving down rapidly with a certain accelerative velocity to impact the seat. As the needle moves down, fluid is squeezed. Fluids in the vicinity of a needle tip displace with the stem, and, meanwhile, in the central place between the needle and inner wall of the chamber, flow back toward the supply port. This process is not ceased until the needle hits on the seat. In the premise of colliding with seat, needle tip and seat quickly form a chamber volume to trap an amount of fluids inside. Needle tip and seat are engaged after collision. And only orifice exit path is left for pressurized fluid to escape from. Because of it, fluid is ejecting throughout the orifice in a stream. However the source of additional fluid is already blocked, and final impact of needle and seat snaps the stream. Fluid exits from a nozzle orifice in stream prior to needle colliding with seat; it requires the needle, substrate and fluid to be interacting with each other at the same period. Adhesive is depositing on the substrate as soon as the fluid flees out a jet nozzle and impacts substrate surface. Usually the nozzle is 100μm orifice which has a length of 0.5mm and can be oriented 2mm away from a surface. Fluid deposition occurs at the same time as an needle moves down. In the meantime, the substrate also moves at a known line speed. Fluid exits out of the nozzle and travels to substrate surface after needle is stopped, and it is extracted away from that surface. During this one cyclic period, fluid continues to aggregate onto substrate although substrate is still moving, but its speed is relatively lower than fluid exit velocity. Thereof, amount of fluid accumulating on substrate is basically determined by fluid exit velocity and substrate line speed.

3. Mathematical Model Analysis

Dynamic mechanical analysis is applied to needle and nozzle impact model, in which the pneumatic actuator is used to drive needle. In following equations, the mass of needle is m kg; g is constant acceleration of gravity; inlet pneumatic pressure on needle top is p Mpa; needle cap pressure effect area is s m^2; needle stroke is designated as l; additional resistance force f_{total} is comprised of liquid friction resistance force, residual air resistance force in opposite air chamber and needle friction force with other components and seal. In terms of the third law of Newton, conservation of momentum, we can derive equations (2-1), (2-2) and (2-3) respectively.

$$m \cdot \frac{d^2x}{dt^2} = m \cdot g + p \cdot s - f_{total} \qquad (2\text{-}1)$$

$$\frac{1}{2} \cdot \frac{d^2x}{dt^2} \cdot t^2 = l \qquad (2\text{-}2)$$

$$t = \sqrt{\frac{2 \cdot m \cdot l}{m \cdot g + p \cdot s - f_{total}}} \qquad (2\text{-}3)$$

The mathematical model is derived to describe the needle kinetic characteristic. In this research, once an jetting device is designed, some of parameters will be defined accordingly. For example, the mass of needle is m , and needle cap pressure effect area is s. In order to get appropriate motion characteristic, needle geometry dimensions must be refined before freezing its structure. Considering on additional resistance force, this force is unexpected to see. Therefore, many methods are employed to mitigate this resistance force. Compared with pneumatic press force on needle top, this additional resistance force is relatively tiny so that it is ignored to take into account eventually. [3]

Three conservation laws are used to interpret jetting fluid dynamics process and may be defined in integral or differential form. Mathematical equations of those conservation laws may be applied to the concept of a compressed chamber. A compressed chamber is a volumetric change in space through which fluid is squeezed in and flown out. Integral formulations of the conservation laws consider the change in mass, momentum, or energy within the compressed chamber. Mass continuity (conservation of mass) is written in formulation (2-4). It proves that the amount change of fluid mass inside a compressed chamber must be equal to the net amount of fluid flow out the chamber. Physically, this statement requires that mass is neither increased nor destroyed in the compressed chamber and can be transformed into the integral form of the continuity equation. [4]

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho \upsilon)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \quad (2\text{-}4)$$

Where ρ is the fluid density. u, v, w are the flow velocity vectors, and t is time.

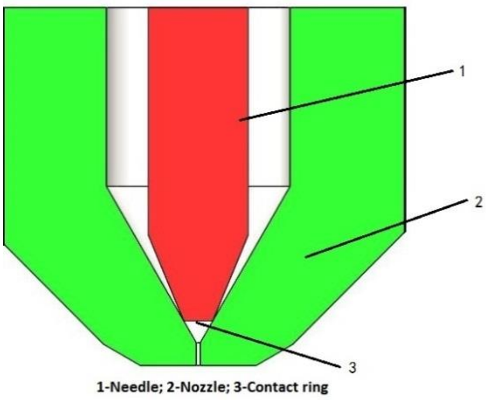
Navier-Stokes equations are useful to describe the physics of jetting fluid as a mathematical model. In theory, N-S equations are solved for a given flow problem by using the method of calculus. However, practically, these equations are too complex to solve analytically.

$$\begin{cases} \dfrac{\mathrm{d}u_x}{\mathrm{d}\tau}=f_x-\dfrac{1}{\rho}\dfrac{\partial p}{\partial x}+\nu(\dfrac{\partial^2 u_x}{\partial x^2}+\dfrac{\partial^2 u_x}{\partial y^2}+\dfrac{\partial^2 u_x}{\partial z^2}) \\ \dfrac{\mathrm{d}u_y}{\mathrm{d}\tau}=f_y-\dfrac{1}{\rho}\dfrac{\partial p}{\partial y}+\nu(\dfrac{\partial^2 u_y}{\partial x^2}+\dfrac{\partial^2 u_y}{\partial y^2}+\dfrac{\partial^2 u_y}{\partial z^2}) \\ \dfrac{\mathrm{d}u_z}{\mathrm{d}\tau}=f_z-\dfrac{1}{\rho}\dfrac{\partial p}{\partial z}+\nu(\dfrac{\partial^2 u_z}{\partial x^2}+\dfrac{\partial^2 u_z}{\partial y^2}+\dfrac{\partial^2 u_z}{\partial z^2}) \end{cases} \quad (2\text{-}5)$$

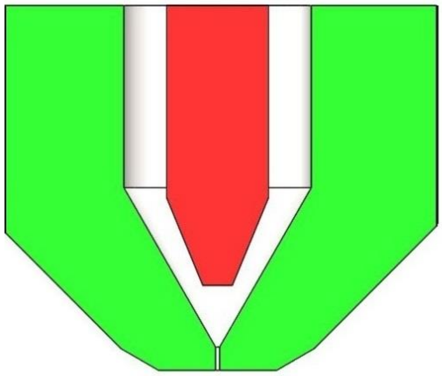
With the developing of numerical computation methodology, high speed computers are adopted to fit approximations to the formulation using various techniques such as finite difference, finite volume, and finite element. In this field computational fluid dynamics (CFD) is suitable to explore fluid properties to more match with real outcome.

4. Simulation Result Analysis

Based on current jetting device working principle, 2D model is created to conduct simulation study. In this research, jetting frequency is 167hertz, whose duty cycle is 3ms on and 3ms off. There are two stages in one cyclic period. In a closed stage, a needle is impacted with nozzle and stop moving forward to shut off fluid dispensing. Then less than 3ms off time, the needle reverses to move upwards rapidly until it hits on the cap to stop at upper point. Vice versa the device continues to eject fluid through a nozzle orifice.



A: Closed Stage



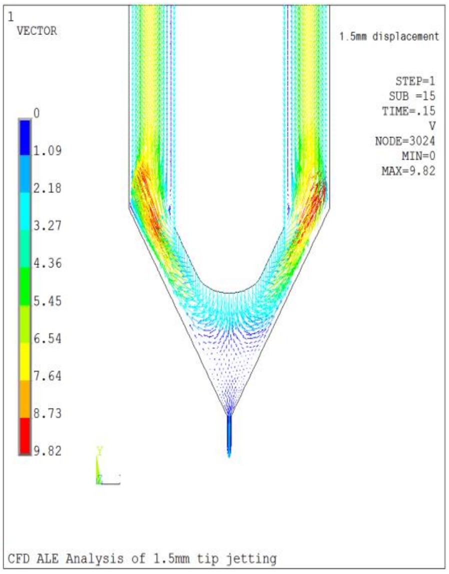
B: Open Stage

Figure 2: The jetting device working principle

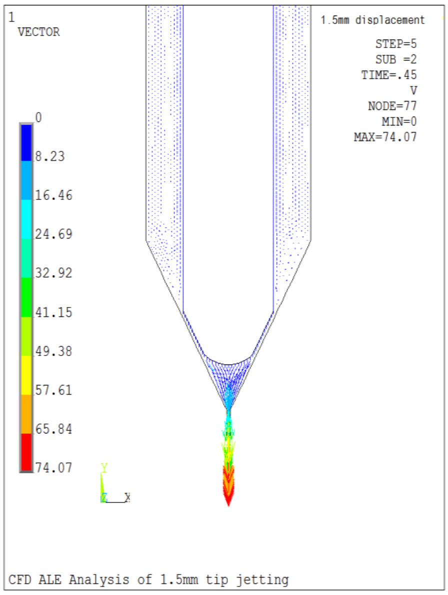
CFD is used to perform the simulation work. The boundary conditions are hypothesized to set as: the residual value is 10-5, the temperature is 300*K*, and the fluid media is purge glue. Kinetic viscosity is about 6,500*cp*; meanwhile fluid filling pressure and the nozzle outlet pressure are set to 0.3*Mpa* and standard atmosphere pressure, respectively.

4.1 Effects of Needle Stroke L on Hydrokinetic Characteristic

In order to research how needle stroke *L* affects fluid dynamics property, several CFD simulation analyses are applied in terms of a variety of needle stroke. Typically, it specifies different *L* values: 0.5mm, 1.0mm and 1.5mm, for running imitation. With the exception of various needle displacements, the simulations are conducted under same external and internal load conditions. The basic parameters are defined as: 1. Needle tip contact ring illustrated in Fig.2 remains the 1.5mm; 2. Pneumatic air pressure is set as 0.5*Mpa* to actuate needle motion. 3. Fluid hydraulic pressure is defined as 0.3Mpa to force fluid to fill into a compressed chamber inside the nozzle; 4. Although the needle stroke is varied, needle and nozzle geometric dimensions remain the same in three simulation scenarios. [5]



(a) Jetting at 0.15ms



(b) Jetting at 0.45ms

Figure 3: Contour plot of nozzle jetting velocity under 1.5mm needle stroke

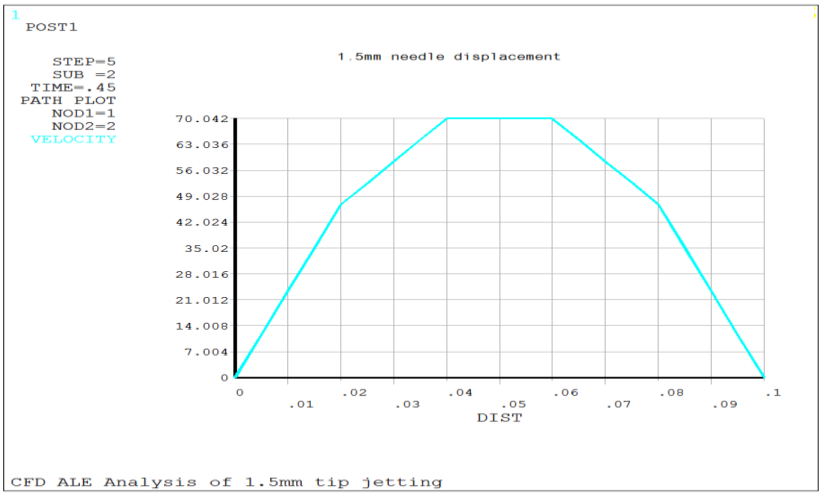
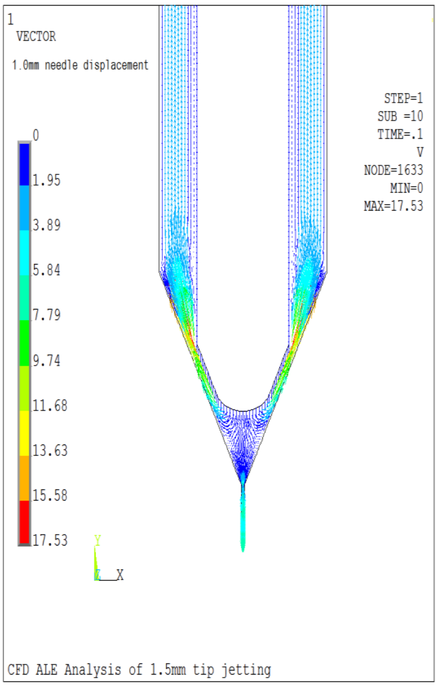
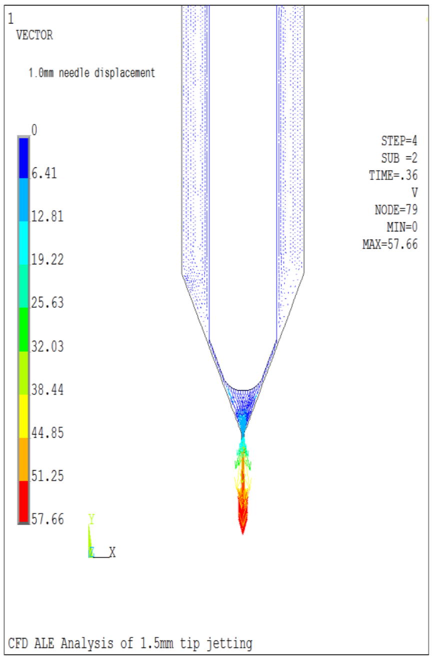


Figure4: Velocity curve of nodes in orifice under 1.5mm stroke

Needle travel time *t* is derived from taking needle stroke 1.5mm and other known values into formulation (2-3). In this case it takes needle to displace 0.45ms from open stage to hit nozzle in closed stage. After collision, the needle speed turns into zero in very short impact time. Fig.3 (a) is the contour plot of jetting velocity after 0.15ms needle movement. In the meanwhile, maximum Jetting velocity is reached in 0.45ms as soon as needle collision occurs, which is expressed in Fig.3 (b). Fluid exit vector plot displays in Fig.4, in which fluid exit velocity is distributed in cross section of nozzle.



(a) Jetting at 0.10ms



(b) Jetting at 0.36ms

Figure 5: Contours plot of nozzle jetting velocity under 1.0mm needle stroke

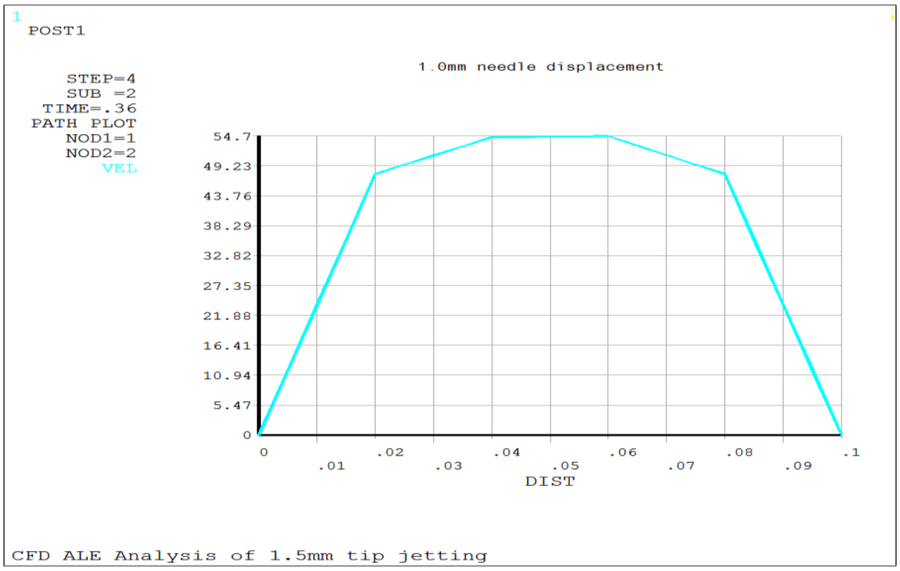
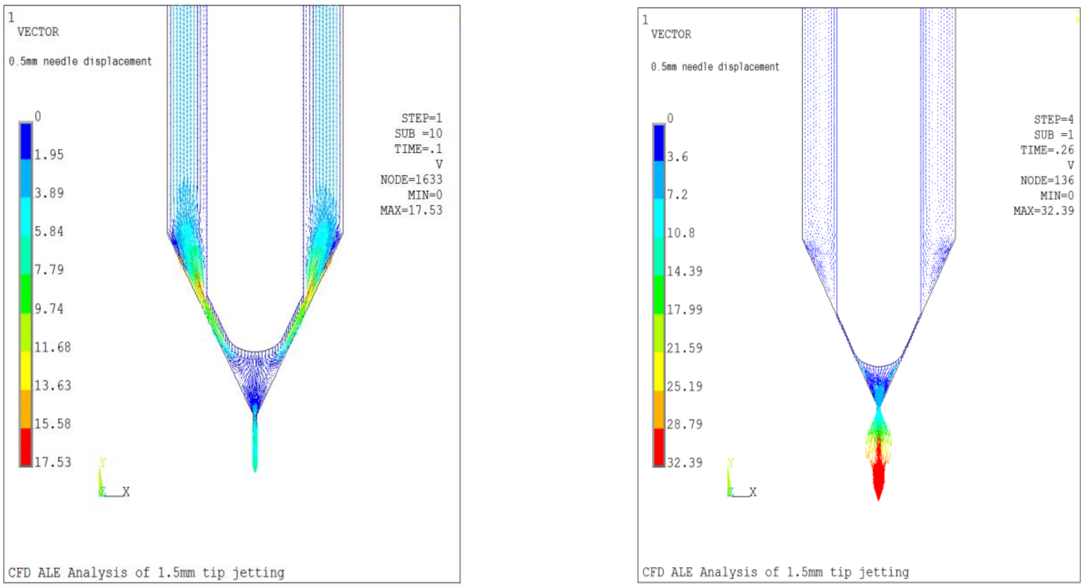


Figure 6: Velocity curve of nodes in orifice under 1.0mm stroke

Likewise, if the needle stroke is 1.0mm, needle travel time *t* is calculated to be 0.36ms accordingly. Thereof，Fig.5 (a) shows nozzle jetting velocity after 0.10ms needle movement, as well as the Fig.5 (b) displays the contours plot of nozzle jetting velocity in collision closed stage at 0.36ms. And fluid exit vector plot is displayed in Fig.6



(a) Jetting at 0.10ms **(b) Jetting at 0.36ms**

Figure 7: Contours plot of nozzle jetting velocity under 0.5mm needle stroke

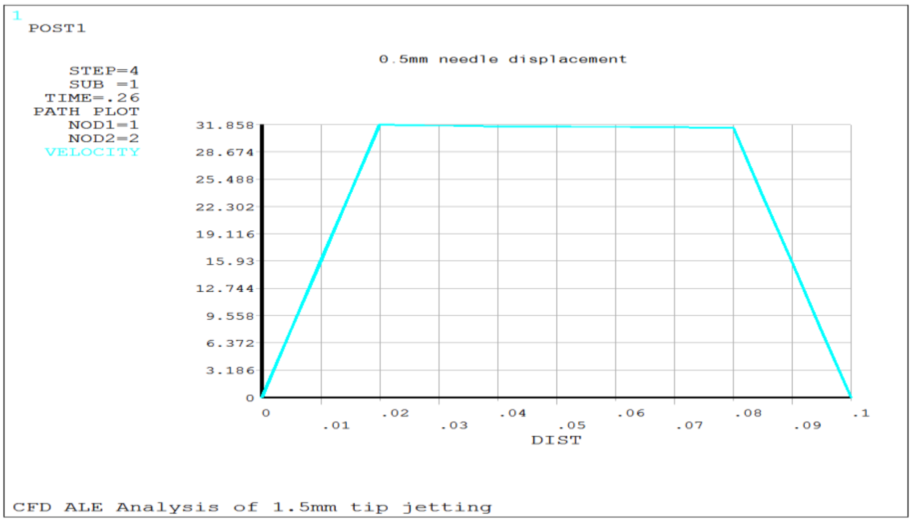


Figure 8: Velocity curve of nodes in orifice under 0.5mm stroke

Finally, if the needle stroke is 0.5mm, needle travel time *t* is calculated to be 0.26ms accordingly. Thereof，Fig.7 (a) shows nozzle jetting velocity after 0.10ms needle movement, as well as the Fig.7 (b) displays the contours plot of nozzle jetting velocity in collision closed stage at 0.26ms. And fluid exit vector plot displays in Fig.8

Table 1: Needle stroke effects

Scenario	L/mm	Vex (m/s)	Pin /Mpa
1	0.5	31.05	290.6
2	1.0	54.7	309.7
3	1.5	70.04	249.8

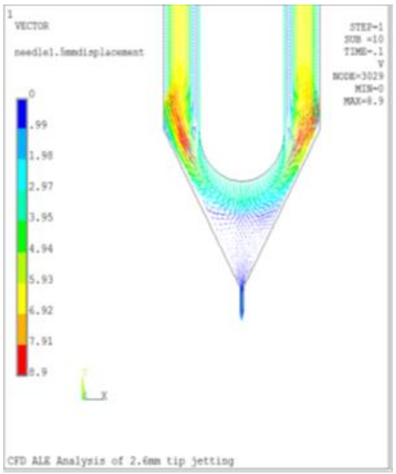
Note: V_{ex} –Fluid outlet max.velocity in nozzle

P_{in} – Fluid max.pressure inside compressed chamber

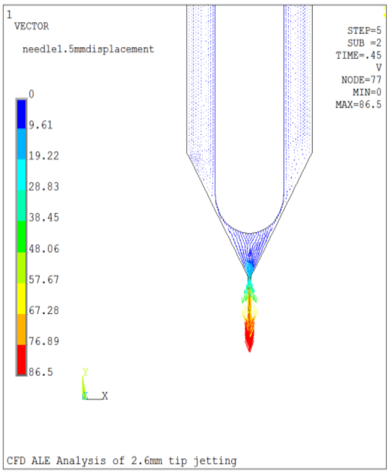
In terms of simulation results, it enables us to create Table1 to summarize simulation data for further assessment and concludes.

4.2 Effects of Needle Contact Ring Size on Hydrokinetic Characteristic

CFD simulation results indicate that the needle tip contact ring size is critical to influence fluid dynamics property. According to fluid jetting process, a needle compacts with nozzle and seats tightly inside. Needle tip round corner matches with nozzle inner wall, where the contact ring is formed referring to Fig.2. In this study, it separately specifies 2.6mm, 1.5mm and 1.0mm variable values for the needle tip size for simulation running. Besides, other parameters are defined as: 1.Needle stroke is kept as 1.5mm for three scenarios; 2.Pneumatic air pressure is 0.5*Mpa* to actuate needle motion; 3.Fluid hydraulic pressure is defined to 0.3Mpa to force fluid to fill into a compressed chamber inside nozzle; 4. Despite that the needle tip size is varied, needle and nozzle geometric dimensions remain the same in three simulation scenarios.



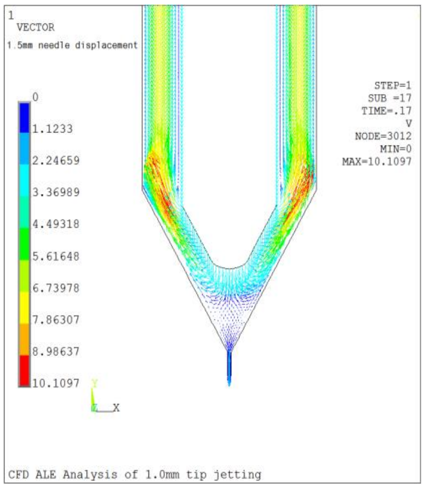
(a) Jetting at 0.10ms



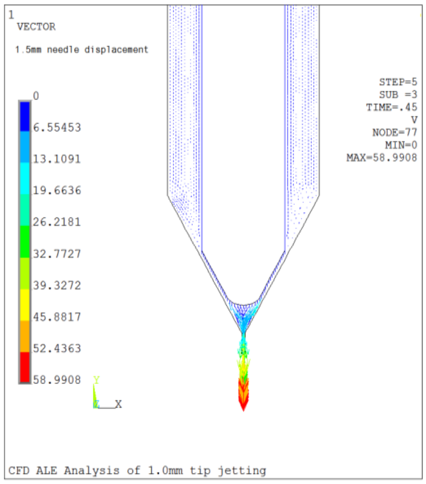
(b) Jetting at 0.45ms

Figure 9: Contours plot of nozzle jetting velocity for 2.6mm needle tip.

Fig.9 (a) is the contours plot of nozzle jetting velocity for 2.6mm needle tip after 0.10ms needle movement. Similarly, Fig.9 (b) exhibits the contour plot of nozzle jetting velocity for 2.6mm needle tip at 0.45ms as a needle hits the nozzle.



(a) Jetting at 0.17ms



(b) Jetting at 0.45ms

Figure 10: Contours plot of nozzle jetting velocity for 1.0mm needle tip

Fig.10 (a) is the contours of nozzle jetting velocity of 1.0mm needle tip after 0.17ms needle movement. And Fig.10 (b) exhibits the contours of nozzle jetting velocity of 1.0mm needle tip at 0.45ms as a needle hits the nozzle.

Based on simulation results and output data, respectively, for 2.6mm, 1.5mm and 1.0mm needle tip size, the summarized Table.2 chart is generated for further analysis and conclusion made.

Table 2: Needle tip size effects

Scenario	*Tip/mm*	$V_{ex}\,(m/s)$	$P_{in}\,/Mpa$
1	2.6	84.74	462.2
2	1.5	70.04	249.8
3	1.0	55.96	218.6

Note: V_{ex} –Fluid outlet max.velocity in nozzle

P_{in} – Fluid max.pressure inside compressed chamber

4.3 Effects of Nozzle Flare Angle on Hydrokinetic Characteristic

Fig.11 exhibits a 60°flare angle that is drilled inside the nozzle body. In order to evaluate the effects of flare angle on fluid dynamics property, a variety of angles are assigned the φ values : 60°, 90°and 120°, respectively. And the boundary conditions preference setting is kept as the same as the previous one.

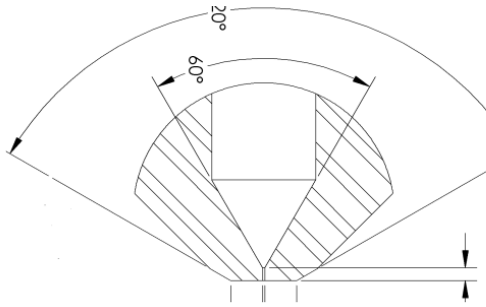


Figure 11: Nozzle flare angle legend

In addition, we hypothesize all of proposed angles are adopting of the same needle tip size β_{tip} mm. As stated needle moves down toward and hits onto nozzle in reference to Fig2. Therefore the compressed chamber is reiterating to generate during working cycles. We assume the cross section area of compressed chamber to be S mm^2. In regards to this specific compressed chamber volume, it is equivalent to section area S. As long as the flare angle is defined, the volume can be determined in calculation formulation.

$$S_{60^\circ} = \frac{1}{2} \times L \times H = \frac{\sqrt{3}}{4} \times {\beta_{tip}}^2 mm^2$$
$$S_{90^\circ} = \frac{1}{2} \times L \times H = \frac{1}{4} \times {\beta_{tip}}^2 mm^2 \quad (3\text{-}1)$$
$$S_{120^\circ} = \frac{1}{2} \times L \times H = \frac{\sqrt{3}}{12} \times {\beta_{tip}}^2 mm^2$$

Apparently, the smaller flare angle, the bigger volume size of compressed chamber obtained. We assume that the boundary conditions and loads are exactly same configured, except to the flare angle. Overall fluid amount flowing into compressed chamber is theoretically identical before a needle touches the nozzle, despite that the flare angle is varied. A needle pushes fluid into a compressed chamber, in which inner pressure is generated.

Considering to chamber space, this pressure is correlated with section area S. They are inversely ratio related. Meanwhile, high inner pressure forces fluid to jet through a nozzle orifice. According to Bernoulli equation (3-2) for ideal liquid, fluid exit velocity u_2 is primarily driven by needle ultimate speed u_1 at the moment when compressed chamber volume is built. Whatever the flare angle is designated, needle final impact speed is remained constant as long as needle stroke and its geometry are all the same. In the other hand, (Z_1-Z_2) is the relative height Z axis distance, which is the distance between needle tip and a nozzle orifice when compressed chamber is generated. In this analysis, the flare angle of 120° has lowest Z height so that the exit velocity would be smallest although the needle speed is commensurate with others. Therefore, small size of flare angle results in high jetting velocity. On the contrary, more open flare angles are beneficial for squeezing fluid to escape from a compressed chamber. This is obviously negative to build an inner pressure inside chamber.

$$\frac{p_1}{\rho \cdot g} + z_1 + \frac{u_1}{2 \cdot g} = \frac{p_2}{\rho \cdot g} + z_2 + \frac{u_2}{2 \cdot g} \quad (3\text{-}2)$$

5. Conclusions

Simulations regarding the effects of major components structure geometry and feature on the fluid dynamics characteristics of jetting device are implemented while the purge glue material is used as the working media fluid. Research results demonstrate that structural parameters and features influence the fluid exit velocity, of which the jetting dot size is affected finally. The relevant conclusions can be obtained.

1). Needle stroke is one of important factors to determine fluid jetting velocity. In order to eject a small dot size, as smaller stroke should be designated as possible. In this research the best appropriate stroke is 0.5mm.

2). Needle tip size is the second important factor to influence fluid jetting velocity. In order to eject small dot size, 0.5mm needle tip seems to be the suitable choice, but a tapered needle trends to decrease mechanical strength incurring to part failure. It needs to compromise this potential risk.

3). Lastly, the nozzle flare angle had better takes a big value. We do prefer to 120° angle as the best solution. However the correlated components such as needle must be associated with this specific geometry accordingly.

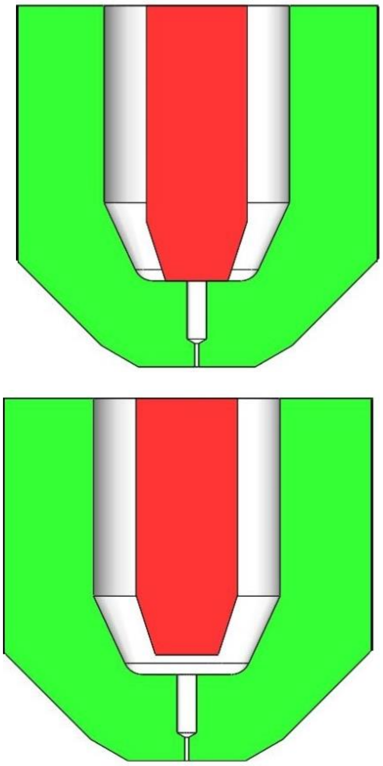


Figure 12: Proposed jetting structure

In a nutshell, Fig.12 shows the best proposal design of jetting actuator structure for the intent on jetting small dot size and working reliability.

References

[1]. B. G. Yacobi ,S. Martin , K. Davis etc, Adhesive bonding in microelectronics and photonics. Journal of Applied Physics 91, 6227 (2002):10-20

[2]. Babiarz, Alec J，Jetting Small Dots of High Viscosity Fluids for Packaging Applications. Semiconductor International, 2006, 29(9): SP2-SP8

[3]. Li Ruping, Nie Songlin, Yi Menglin etc, Simulation investigation on fluid characteristic of jet pipe water hydraulic servo valve based on CFD. J Shanghai Univ (Engl Ed), 2011, 15(3):201-206

[4]. Xu Xin, Jet dispensing process simulations and experimental research [D], School of electromechanical engineering Guangdong University of technology, 2014

[5]. Xie Longhan, Zhao Xinyu, Zhang jiongming, Ansys CFX fluid simulation and analysis [M], Electronic Industry Press, 2012

My name is Zhong-Quan Qiu, I was born in YanCheng city JiangSu province China in 1977. I received B.S. degree in Mechanical Engineering from NanJing University of aeronautic and astronautic in 2000. In addition I obtained M.S. degree in Mechanical Engineering from ShangHai JiaoTong University in 2015. Now I am working at Nordson China. As an engineering manager, I do charge of new products research responsibility for developing melter, hose and applicator, which are widely used in Nonwoven, packaging and product assembly business fields.