Effect of Nozzle Diameter on Particle Velocity in Fine Particle Peening Processes

Koichiro NAMBU[*1], Yusuke SAEKI[2] and Masahiro OKUMIYA

[1] *Department of Mechanical Engineering, Osaka Sangyo University, 3-1-1, Nakagaito, Daito-city, Osaka, Japan*
[2] *Formerly of Toyota Technological Institute, 2-12-1, Hisakata, Tenpaku-ku, Nagoya-city, Aichi, Japan*
[3] *Toyota Technological Institute, 2-12-1, Hisakata, Tenpaku-ku, Nagoya-city, Aichi, Japan*

Fine Particle peening is a method to obtain surface modification effects, such as fatigue strength improvement, by bombarding the work material with particles at high velocity. However, there are many factors that affect the surface modification effect, making it difficult to select the optimum conditions. The particle velocity and particle flight behavior have not been clarified due to the large number of flying particles in addition to the extremely high particle velocity. Therefore, in this study, in addition to air flow analysis inside and outside the nozzle, particle velocity analysis using the particle method was conducted. ANSYS was used for the airflow analysis, and Particle Works was used for the particle method. The nozzle diameter and nozzle-to-work distance were varied. The nozzle diameter was varied from 3 to 10 mm. The nozzle-to-work distance was 50, 100, and 150 mm. The pressure at the nozzle entrance was set to 0.2 MPa, and air flow analysis was performed under incompressible fluid conditions. The particle method used iron-based particles with a particle diameter of 100 μm as a model for analysis. The results of the airflow analysis showed that the potential core area increases as the nozzle diameter increases. This was attributed to the shear layer caused by the wall resistance inside the nozzle. Next, particle velocity analysis showed that particle velocity tended to increase with increasing nozzle diameter. In addition, it was found that the particle velocity increased with increasing nozzle-to-work distance. Next, the particle flight behavior was analyzed, and it was found that the particles accelerated most at the parallel part of the nozzle and continued to accelerate after the nozzle exit. Finally, to verify the validity of the analysis, the particle velocities were compared with those measured by a high-speed camera. Although the geometry of the nozzle was slightly different, the measured and calculated velocities showed similar trends, suggesting that the present method is valid.

Keywords: *Fine Particle Peening, FEM, Particle method analysis*

1. Introduction

Fine particle peening treatment, one of the surface modification treatments, is widely used for automotive gears. One method of flying these particles is the air injection method. In this method, shots are ejected from a nozzle by a stream of air. As the particles are pushed out by the air stream, the air stream affects the velocity of the particles. Outside the nozzle, the spread of the airflow causes a spread in the trajectory

Our group has succeeded in simulating particle flight behavior by combining fluid analysis and the discrete element method in order to clarify particle flight behavior. Based on these results, this study examines the effects of particle flight behavior and nozzle diameter.

2. Experimental Procedure

A three-dimensional finite element method (FEM) steady-state analysis was used to analyze the airflow in and out of a shot peening nozzle. ANSYS Fluent was used for the FEM analysis of airflow. The analytical model used in the airflow analysis was created using CAD software. Figure 1 shows a cross-sectional view of the analytical model. The dimensions of the base material were 50 mm wide, 50 mm long, and 5 mm thick. The center axis of the nozzle was set as the y-axis, and y=0 was set at the nozzle exit.

Air flow was assumed to be air (ideal gas). The turbulent viscosity model used was the k-ω SST model, which can correctly capture turbulent flow near the nozzle wall and turbulent flow away from the wall. The process conditions were analyzed with a nozzle inlet pressure of 2000 hPa and a nozzle outlet to substrate distance of 50, 100, and 150 mm. The air temperature at the nozzle inlet and near the nozzle outlet were set to 300 K, and the atmospheric pressure was 1013 hPa. The effect of gravity was neglected in this analysis. A three-dimensional mesh was created using ANSYS Fluent polyhedral meshing method with a maximum mesh length of 0.001 mm. The minimum and maximum surface mesh lengths were 0.05 mm and 0.77 mm, respectively.

Next, we simulated the flying particles using Granuleworks, a powder analysis software. The simulation conditions were the same as those described above. The airflow was introduced into Granuleworks by entering the x, y, and z coordinates of the airflow region obtained from the airflow analysis described above and the velocities in the x, y, and z directions at those coordinates.

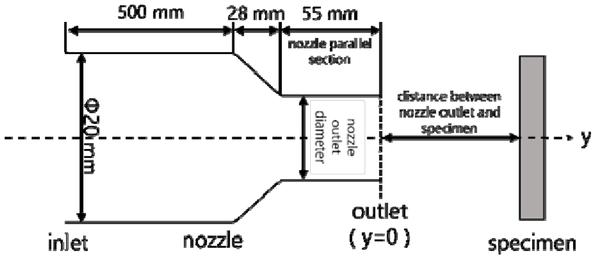

Fig. 1 Nozzle and specimen dimensions

3. Results and Discussion

3.1 Air velocity analysis at the center of the nozzle

Fig.2 shows the change in air velocity in the y-direction at the nozzle center axis (y-axis) when the distance between the nozzle outlet and the substrate is 150 mm. The figure shows that the airflow rapidly accelerates to a velocity of approximately 250 m/s just before the nozzle parallel ($y \le -55$), continues to accelerate at the nozzle parallel ($-55 \le y \le 0$), and reaches approximately 330 m/s (almost the speed of sound) near the nozzle exit (y=0). After the nozzle exit, the airflow decelerates and reaches 0 m/s just before the substrate. The airflow is considered to slow down rapidly just before the substrate not only due to air resistance, but also due to the bounce of the airflow from the substrate. The airflow velocity increases with increasing nozzle exit diameter, especially outside the nozzle (y≥0). Generally, it is expected that the air velocity increases as the nozzle exit diameter decreases. However, in this study, the opposite is true. The pressure ratio of the nozzle outlet pressure to the nozzle inlet pressure is 1013/2000 = 0.5065. This is less than the critical pressure ratio (ratio of critical pressure to nozzle inlet stagnation point pressure: 0.52810) and is considered to be choked at any nozzle exit diameter. Therefore, it is considered that almost equal distributions of air velocity were obtained at the nozzle parallel (-55 ≤ y ≤ 0). Further upstream from the nozzle parallel (y<-55), the air velocity increases as the nozzle exit diameter increases. This is because the boundary layer has a larger effect on the air velocity as the nozzle exit diameter decreases, and the total pressure loss in the nozzle flow becomes larger, resulting in a decrease in the air velocity.

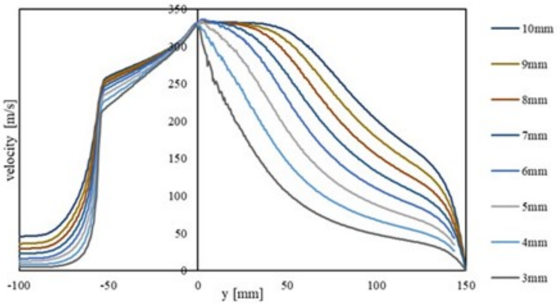

Fig.2 Air velocity analysis results at the center of the nozzle (The nozzle at a distance between nozzle outlet and specimen of 150 mm)

3.2 Particle Velocity Analysis

Based on the results of the airflow analysis, a discrete element analysis was performed to investigate the maximum velocity of the flying particles. For each condition, the velocity of 100 randomly selected particles was measured from inside the nozzle to just before impact with the substrate, and the average of the maximum velocity for each particle was taken. Fig.3 shows the average maximum velocity of the particles in the y direction at a distance of 150 mm between the nozzle exit and the substrate. The error bars in the figure show the maximum to minimum maximum velocity of the particles under each condition. The results show that the average maximum velocity of the particles increases as the nozzle exit diameter increases. This is thought to be due to the increase in air velocity in and out of the nozzle as the nozzle exit diameter increases.

3.3 Relation between air velocity and particle velocity

Fig. 4 shows the velocity variation of particles in the y direction at a distance of 150 mm between the nozzle outlet and the substrate. The plots in red are the points of maximum particle velocity for each condition. Particles accelerate rapidly at the nozzle parallel (-55 ≤ y ≤ 0), which has the greatest effect on particle velocity, and tend to accelerate after the nozzle exit (y = 0).

The red-filled plots in Fig.4 show that as the nozzle exit diameter increases, the particles reach their maximum velocity at a point closer to the substrate. This is because an increase in the nozzle exit diameter allows the airflow with higher velocity to reach a point further away from the nozzle exit, and thus the particles are accelerated to a point further away from the nozzle exit.

Under the conditions of this simulation, the particles reach their maximum velocity when the distance between the nozzle exit and the substrate is between 100 and 150 mm, so the optimum condition for the distance between the nozzle exit and the substrate is between 100 and 150 mm.

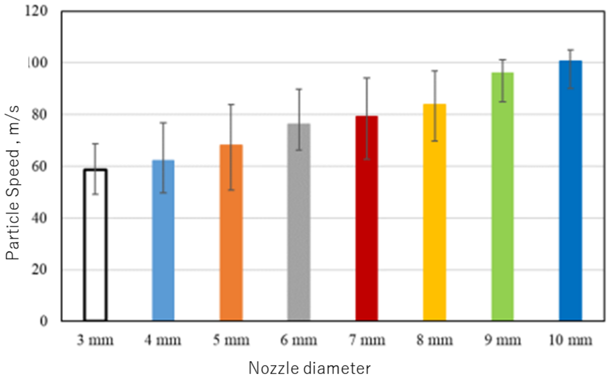

Fig.3 Maximum velocity of particles at each nozzle diameter

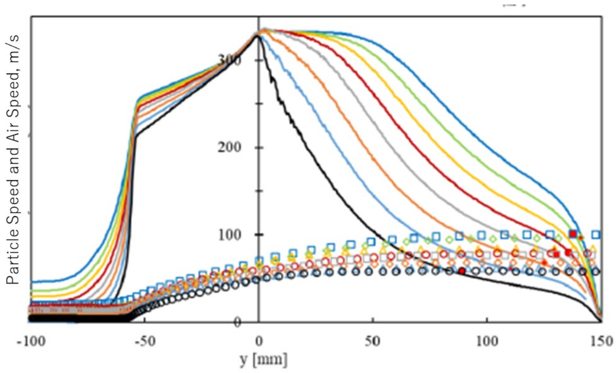

Fig.4 Airflow and particle velocity at nozzle center (150mm)

4. Conclusions

1. The airflow accelerates rapidly at the tip of the nozzle, and then accelerates slowly in the parallel section of the nozzle, reaching its maximum velocity (approximately 330 m/s) near the nozzle outlet.
2. Particles accelerate most rapidly at the nozzle parallel and continue to accelerate after the nozzle exit. 5.
3. As the nozzle exit diameter increases, the particles accelerate to a point farther from the nozzle exit because the airflow maintains a velocity of 300 m/s or higher.