Numerical and Experimental Investigations of Supersonic Jets from Sootblower Nozzle*

Masashi KASHITANI**, Yoshiaki MIYAZATO**,
Mitsuharu MASUDA** and Kazuyasu MATSUO**

The present paper describes the numerical as well as the experimental study on the axisymmetric supersonic jet associated with a sootblower nozzle for boiler cleaning. The Navier-Stokes equations are solved numerically, and the experimental work is also performed with the low-density wind tunnel. Calculations and experiments both cover the region from the nozzle exit to the downstream distance of 20 times the nozzle exit diameter. The effect of the nozzle divergence angle is discussed by comparing the experiments with calculations, and it is pointed out that, even when the expansion ratio across the nozzle is correct, the regular shock-cell structure appears in the jet for the nozzles with finite divergence angles. Also, the present numerical results on the impact pressures along the jet center-line agree favorably well with those reported by the other researchers.

Key Words: Compressible Flow, Supersonic Jets, Numerical Analysis, Sootblower Nozzle, Shock Cell Length

1. Introduction

The deposit accumulated on the fireside surface of boiler tube drastically reduces the heat-transfer efficiency and causes the high flue-gas temperature as well as the plugging of flue-gas passages. Therefore, this deposit must be removed for the continuous operation of the boiler with high thermal efficiency. While the boiler is being operated, the deposits can be removed by the jets from sootblower nozzles^{(1),(2)} installed in a lance tube as shown in Fig. 1. The lance tube moves back and forth with rotation around its axis, and the jets with high-pressure steam from the nozzles blast deposit away.

In general, a desirable shape for the sootblower nozzle is that the jet issuing from the nozzle does not contain shock waves, because shock waves decelerate the flow speed and decrease the jet power for removing deposit. The jet without shock waves is obtained

by the nozzle having the proper ratio of the throat to the exit area. In this nozzle, the static pressure at the nozzle exit is equal to the ambient pressure. In addition, the stream line of the jet must be parallel to its axis to avoid the generation of compression and expansion waves. Although this flow field is obtained by using the contoured nozzle, the sootblower nozzle is too small to manufacture with the desired shape, and the nozzle currently used has a simple conical shape from its throat to the exit. In addition to this

Fig. 1 Schematic diagram of sootblower jets

* Received 13th July, 1997

** Department of Energy Conversion Engineering, Interdisciplinary Graduate School of Engineering Sciences, Kyushu University, Kasuga, Fukuoka 816-8580, Japan

problem of the nozzle shape, the sootblower uses the far downstream portion of the jets where the process of decay and the regular shock-cell structure are not known.

To design the sootblower nozzles, the detailed structure of the jet must be known to predict its performance. Previous works^{(1),(2)}, however, has not considered the effect of the nozzle divergence angle on the jet structure, and no information has so far been available. The purpose of the present work is to clarify the detailed structure of the free jets issuing from a conically divergent supersonic nozzle. The TVD finite difference scheme is adopted for the numerical calculation of Navier-Stokes equations. The structures of correctly expanded supersonic free jets are calculated for the axisymmetric nozzles with and without divergence angles at the nozzle exit. The experiments are also performed using the low-density wind tunnel, and the impact pressure distributions in the jets are measured. Calculations and experiments both cover the region from the nozzle exit to the downstream distance of 20 times the nozzle exit diameter. The effect of the nozzle divergence angle is discussed by comparing the experiments with calculations.

Nomenclature

- C_p : specific heat at constant pressure
- De : nozzle exit diameter
- Dth : nozzle throat diameter
- e : internal energy per unit volume
- L_1 : first shock cell length
- Me : nozzle exit Mach number
- p : static pressure
- p_e : nozzle exit pressure
- p_b : test chamber pressure
- p_0 : stagnation pressure
- Pr : Prandtl number
- p_i : impact pressure
- r : vertical distance from jet center line
- Re : Reynolds number based on the throat radius and the throat condition
- t : time
- T : temperature
- u : axial velocity component
- v : radial velocity component
- x : axial distance from nozzle exit
- γ : specific heat ratio
- ρ : density
- ρ^* : nozzle throat density
- θ : nozzle divergence half angle
- μ : viscosity

2. Numerical Method

The unsteady, axisymmetric, compressible Navier-Stokes equations are solved by the high-resolution total variation diminishing (TVD) scheme^{(3),(4)} under the boundary conditions for the sootblower nozzle jet. This scheme has been proved valid from low subsonic to hypersonic flows. To compare the calculated results with the present experiments done by the low density wind tunnel, the flow is assumed laminar. The basic equations are written in the cylindrical coordinates as ;

$$\frac{\partial U}{\partial t} + \frac{\partial F_1}{\partial x} + \frac{\partial F_2}{\partial r} + \frac{\partial G_1}{\partial x} + \frac{\partial G_2}{\partial r} + H = 0 \quad (1)$$

where

$$U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ e \end{bmatrix}, \quad F_1 = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ (e+p)u \end{bmatrix}, \quad F_2 = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ (e+p)v \end{bmatrix},$$

$$G_1 = \frac{1}{Re} \begin{bmatrix} 0 \\ f_x \\ -\tau \\ uf_x - v\tau - g_x \end{bmatrix},$$

$$G_2 = \frac{1}{Re} \begin{bmatrix} 0 \\ -\tau \\ f_r \\ uf_r - v\tau - g_r \end{bmatrix},$$

$$H = \frac{1}{r} \begin{bmatrix} \rho v \\ \rho uv - \tau/Re \\ \rho v^2 - \alpha/Re \\ (e+p/\gamma)v + (vf_r - u\tau - g_r)/Re \end{bmatrix}$$

and

$$f_x = \frac{2}{3} \mu \left(\frac{\partial v}{\partial r} - 2 \frac{\partial u}{\partial x} + \frac{v}{r} \right),$$

$$f_r = \frac{2}{3} \mu \left(\frac{\partial u}{\partial x} - 2 \frac{\partial v}{\partial r} + \frac{v}{r} \right)$$

$$g_x = K \frac{\partial T}{\partial x}, \quad g_r = K \frac{\partial T}{\partial r}, \quad K = \mu C_p / Pr$$

$$\tau = \mu \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial r} \right), \quad \alpha = 2\mu \left(\frac{\partial v}{\partial r} - \frac{v}{r} \right)$$

The gas is assumed perfect with a constant specific heat ratio, and the viscosity is determined from the Sutherland law. Also, the Prandtl number Pr is taken as 0.72. The boundary conditions are the adiabatic no-slip on the nozzle wall and the flow symmetry along the jet center line. The downstream end of the computational domain was taken as an outflow surface.

To check the effect of the grid size on accuracy, the calculations are done with the two different grid size ; one has 150×600 points and the other 250×800 points. Under the present physical conditions for the analysis, the results show no significant difference for these two grid sets, and the calculated flow patterns

Fig. 2 Experimental setup

agree very well. By considering the computational time, the 150×600 grid scheme is used through out the present analysis. In this calculation, computational domain is $50De \times 60De$ by rectangular mesh.

The flow field inside the nozzle is not calculated. The effect of the nozzle divergence angle is taken into account by assuming the source type flow on the nozzle exit plane⁽⁵⁾, and parallel flow at the nozzle exit for the nozzle without divergence angle. As will be described in the later section, the structures of the jet are strongly affected by the nozzle divergence angle.

3. Experimental Setup

Figure 2 shows the schematic diagram of the low-density wind tunnel used in the present experiments. The gas in the high pressure reservoir flows into the plenum chamber through the rotameter, whereby the gas flow rate is adjusted by the valve. After the gas stagnates in the plenum chamber, it is accelerated by the test nozzle and then evacuated by a mechanical booster ($31.5 \text{ m}^3/\text{s}$) and a rotary ($5.0 \text{ m}^3/\text{s}$) pumps. The pressures in the plenum chamber and in the test section are carefully controlled by adjusting the flow rate and the gate valve, so that the pressure ratio across the nozzle is kept constant and also the pressure at the nozzle exit matches that in the test section. This assures the expansion of the gas from the nozzle being correct.

The jet flow field is measured by the impact pressure probe with 1 mm in outer diameter. This probe is set to the traversing device attached on the upper part of the test section.

Two axisymmetric nozzles are used in the experiments. These are shown in Fig. 3 with their typical dimensions and experimental conditions. In this figure, Me is the equivalent Mach number at the exit, D_{th} the throat diameter, De the exit diameter, p_0 the plenum chamber pressure, p_b the test section pressure and θ the divergence half angle at the nozzle exit. From Rott's compressible laminar boundary layer

Fig. 3 Test nozzles and experimental conditions

thickness estimation⁽⁶⁾, the exit Mach number of nozzle I in considering the effect of boundary layer thickness is 2.95. Hereafter, these nozzles are denoted by nozzle I and II, respectively. As shown in this figure, nozzle I has a 4 mm parallel passage upstream of the exit plane, whereas nozzle II has a simple conical form in the diverging section.

4. Results and Discussion

As described in the previous section, the experiments are done with the pressure ratios for the correct expansion of the gas across the nozzle. To be consistent with the experiments, the pressure ratio is also taken as correct in the numerical analyses. The calculated results are shown in Fig. 4. This is the contour of density ρ normalized by its value at the nozzle throat ρ^* . In this figure, x is the downstream distance from the nozzle throat, r the radial distance from the jet center line, De the nozzle exit diameter and θ the divergence angle of the nozzle. Figure 4(a) is the flow from the contoured nozzle where the stream line at the nozzle exit plane is parallel to the jet center line. As can be expected, the flow field shows simply decay by the effect of viscosity, and no compression or expansion waves are generated. On

Fig. 4 Density contours

the other hand, Fig. 4(b) shows the jet from the nozzle whose divergence angle at the exit is 20° . This corresponds to the flow from nozzle II of Fig. 3. Although the jet pressure at the exit plane is matched to the ambient pressure, the flow field shows the regular pattern of the compression and expansion waves. Since the stream line at the nozzle exit is not parallel to the jet axis but is directed away from the center line, the gas continues to expand after flowing out from the exit. This results in the jet pressure lower than the ambient pressure, and the compression waves are generated on the jet boundary to adjust the pressure difference. These compression waves then coalesce to form oblique shock waves that surround the jet core. The compression or oblique shock waves are reflected as expansion waves on the opposite jet boundary. In this way, the expansion-compression pattern is formed and repeated to the far downstream region. The light and dark regions in Fig. 4 correspond to these expansion and compression regions, respectively. In this paper, the region between the exit and the location where the oblique shock waves intersect is denoted as the first cell, and the cell next to this as the second cell, as shown in the figure. The length of these cells are compared with experiments in the later section of this paper.

The sootblower performance is conventionally presumed to be correlated with the jet's impact pressure⁽²⁾. The impact pressure distribution along the jet center line is shown in Fig. 5, where the impact pressure p_i is normalized by the plenum chamber pressure p_0 , and the circles and the line are the results of the present experiments and the numerical calculations, respectively. These are for the flow from nozzle I and the exit stream line is made parallel to the jet

Fig. 5 Center line impact pressure distribution (nozzle I)

Fig. 6 Center line impact pressure distribution (nozzle II)

axis. The experimental impact pressures, however, are not uniform near the exit, showing the existence of weak compression and expansion waves. As shown in Fig. 3, the wall of nozzle I has a inflection point at 4 mm upstream of the exit, so that the compression wave is probably generated at this point. Except this small fluctuation, the experiments support strongly the numerical results.

Similar results are obtained for nozzle II and are shown in Fig. 6. The A, B and C in this figure denote

Fig. 7 Radial impact pressure distribution (nozzle II)

the locations where the radial impact pressure profiles are measured (Fig. 7 below). As shown in Fig. 3, nozzle II has the divergence angle of 20° , at the exit. As described above, the expansion waves are generated in the flow field and the jet expands after the exit. This causes the increase of the Mach number resulting in the decrease of the impact pressure. The impact pressure then increases abruptly at the location where the oblique shock waves intersect. The fluctuation of the experimental impact pressure along the jet axis seems to be well explained by the present calculation. As shown here, the expansion-compression pattern (cell structure) extends periodically to the far downstream region. Tam calculated the shock cell length for the slightly underexpanded jet by the first order perturbation theory⁽⁷⁾. According to his results for the nozzle with zero divergence angle, the ratio of the first shock cell length L_1 to the nozzle exit diameter D_e is about 3.7. Whereas, in the present work, the experimental L_1/D_e is about 1.9 and the calculation gives $L_1/D_e=3.1$. These are slightly smaller than the Tam's results, and the discrepancy may be attributed to the divergence angle of the nozzle. The decay of the jet in the far downstream region is also shown in Fig. 6, and the experiments, again, seem to support the calculations. These results are compared with those in Fig. 5 for the nozzle without a divergence angle, and the decay rate does not seem to depend strongly on the nozzle divergence angle. This is probably due to the oblique shock waves in the jet from nozzle II being weak. The small difference in decay rate suggests that the nozzle divergence angle is not a critical problem in the sootblower nozzle.

The radial distributions of the impact pressure are shown in Fig. 7, where r is the radial distance from the jet center line and A, B and C denote the axial locations indicated in Fig. 6. The A is the location just upstream of the first intersection of the

oblique shock waves. At this location, the impact pressure on the center line drops due to the increase of Mach number. The location B is immediately behind the intersection, and the impact pressure on the center line is maximum and the weak oblique shock waves are observed on both sides of the center line at $r/D_e = \pm 0.2$. On the other hand, the location C corresponds to the region far downstream, and the impact pressure profile resembles that of the subsonic flow. As shown in Figs. 6 and 7, the expansion and/or compression waves are not discernible in this location.

5. Concluding Remarks

The numerical and experimental investigation is carried out for the axisymmetric supersonic jets used in a sootblower for boiler cleaning. The Navier-Stokes equations are solved numerically by the TVD method, and the jet flow field is measured by the impact pressure probe with the low-density wind tunnel. Two nozzles are tested, whose divergence half angle at the nozzle exit is 0 and 20° , respectively. The numerical and experimental results show that the flow field is affected significantly by the nozzle divergence angle, and it is found that the nozzle with finite divergence angle yields the periodic expansion and compression flow pattern even when the pressure at the nozzle exit matches to the ambient pressure. Also, in the far downstream of the nozzle exit, the decay rate of the jet is obtained by the impact pressure measurements, and it is found to compare favorably well with calculations. Unlike the flow pattern near the nozzle exit, the decay rate in the far downstream region does not seem to depend strongly on the nozzle divergence angle. This suggests that the nozzle divergence angle is not a critical problem in the sootblower nozzle.

Acknowledgments

The authors are grateful to Mr. H. Yano, Mr. N. Kondo and Mr. J. Tanaka for their assistance in the experimental works. This research is partly supported by JSPS Research Fellowships for Young Scientists.

References

- (1) Moskal, T.E., Barton, M.A. and Jordan, C.A., Results of Laboratory Testing and Field Trials of Improved Sootblower Nozzles, TAPPI 1993 Engr. Conf. Proc., (1993), p. 963.
- (2) Jameel, M.I., Cormack, D.E., Tran, H. and Moskal, T.E., Sootblower Optimization, Part I: Fundamental Hydrodynamics of a Sootblower Nozzle and Jet, Tappi J., Vol. 77, No. 5, (1994), p. 135.
- (3) Yee, H.C., Upwind and Symmetric Shock-Capturing Schemes, NASA TM, 89464, (1987).
- (4) Yee, H.C. and Harten, A., Implicit TVD Schemes

- for Hyperbolic Conservation Laws in Curvilinear Coordinates, AIAA J., Vol. 25, No. 3 (1987), p. 266.
- (5) Zucrow, M.J. and Hoffman, J.D., Gas Dynamics, Vol. 1, (1976), p. 232, John Wiley & Sons, Inc.
- (6) Rott, N. and Crabtree, L.F., Simplified Laminar Boundary-Layer Calculations for Bodies of Revolution and for Yawed Wings, J. Aeronautical Science, Vol. 19 (1952), p. 553.
- (7) Tam, C.K.W., The Shock-Cell Structures and Screech Tone Frequencies of Rectangular and Non-axisymmetric Supersonic Jets, J. Sound and Vibration, Vol. 121, No. 1 (1988), p. 135.
-