Journal of Physics: Conference Series

Verification of the calculation modeling methods of the atomizing of a gas and gas-liquid stream from a pneumatic nozzle

To cite this article: Viktor Kuznetsov *et al* 2019 *J. Phys.: Conf. Ser.* **1261** 012019

Verification of the calculation modeling methods of the atomizing of a gas and gas-liquid stream from a pneumatic nozzle

Viktor Kuznetsov [a,b*], **Aleksandr Shebelev**[a], **Igor Anufriev** [b], **Evgeniy Shadrin** [b], **Andrey Minakov**[a,b]

[a]Siberian Federal University, 79 Svobodny, Krasnoyarsk, 660041, Russia
[b]Kutateladze Institute of Thermophysics, Novosibirsk, 630090, Russia

*victor_partner@mail.ru

Abstract. The paper presents a verification of calculation modeling methods of atomizing of a gas and gas-liquid stream from a pneumatic nozzle based on the experimental data obtained in the IT SB RAS. Turbulent supersonic flow of compressible gas is considered here. Mathematical model includes description of gas phase motion based on the RANS (Reynolds Averaged Navier-Stokes) and URANS (Unsteady RANS) approach using two-parameter turbulence model k-ω SST and Reynolds Stress Model (RSM). The Lagrange method was used to model flow of water droplets. Dispersed phase is solved by tracking a large number of droplets through calculated flow field. Dispersed phase can exchange momentum, mass, and energy with fluid phase. Comparative analysis showed an acceptable qualitative and quantitative agreement between calculation and experiment, both for subsonic and supersonic flow.

1. Introduction

Currently, the world is actively exploring prospective slurry alternative fuels for energy production, which consist of water, crushed coal or fuel waste of its processing, and other additives. Such fuels are called [1-3] organic coal-water (OCWF).

Efficiency of the pneumatic nozzle for spraying organic coal-water fuel (OCWF) is determined by organization of gas flows. Experimental and computational studies are needed to control atomization process of liquid fuels, obtain dependencies of angle of flame divergence and the dispersed composition of fuel on geometric characteristics of the nozzle. However, up to present time, systematic experimental and numerical studies of hydro-gasdynamic processes accompanying operation of the nozzle have not been carried out.

The group of researchers at the Institute of Thermophysics of the Russian Academy of Sciences has proposed a pneumatic nozzle [4, 5] based on using properties of wall and cumulative jets and the Coanda effect. For further widespread industrial use of this type of nozzle, it is necessary to determine characteristics of atomization and combustion of suspension fuel. Determining role in efficiency of pneumatic nozzle is played by organization of gas flows. To study the structure of gas-liquid flows of proposed pneumatic nozzle, it is necessary to develop an efficient and reliable numerical modeling methods technique for describing atomization of liquid fuels in perspective burners.

The purpose of this paper is a verification of calculation modeling methods of atomizing of a gas and gas-liquid stream from a pneumatic nozzle based on the experimental data based obtained by PIV (particle image velocimetry) method by staff of the IT SB RAS.

2. Problem statement and research methods

To verify calculation modeling methods of atomizing of a gas and gas-liquid stream from a pneumatic nozzle, following task was considered. The geometry of the nozzle are shown in Fig. 1. Air is fed through annular channel. Width of annular clearance is 0.8 mm. In experiments, studied regimes with initial air pressure in annular gas chamber of the nozzle in range of 1-3 bar.

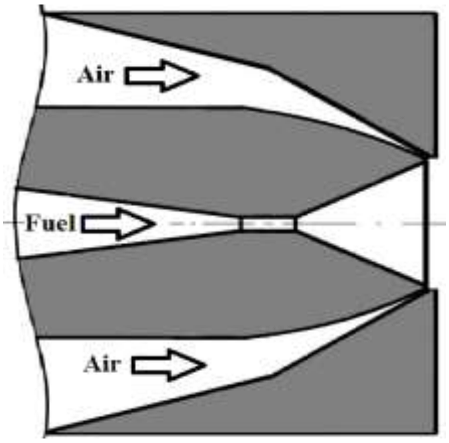


Figure 1. Scheme of the pneumatic nozzle.

A turbulent supersonic flow of a compressible gas is considered here. For the RANS RSM [6, 7] and the k-ω SST models [8], problem was solved in a two-dimensional axisymmetric stationary formulation. Fragment of calculated region and grid is shown in Fig. 2. For calculation, we used structured computational grids with local closeness near diffuser nozzle and region of jet formation. General detailing of this computational grid in axisymmetric case was 620600 nodes. In three-dimensional case, computational grid for the URANS k-w SST method contained about 9,6 million cells, in cross-section, structure of computational grid is similar to grid shown in Fig. 2, but coarser. Conditions of a solid wall were set on outer and inner surfaces of the nozzle. Output conditions with a fixed pressure were set at all outer boundaries of computational domain. The Lagrange method was used to model flow of water droplets. Dispersed phase is solved by tracking a large number of droplets through calculated flow field. Dispersed phase can exchange momentum, mass, and energy with the fluid phase [9].

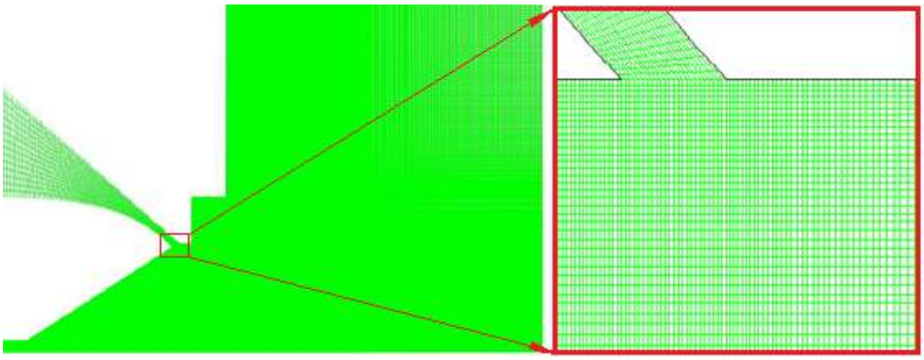
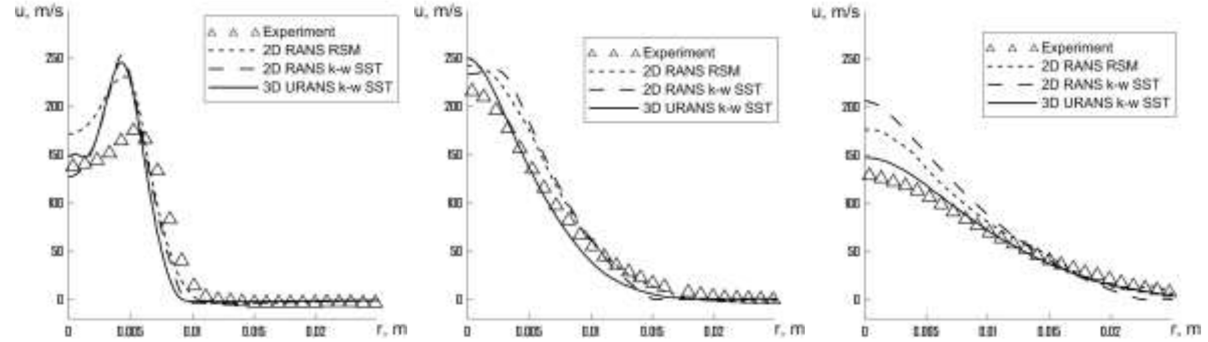


Figure 2. Computational grid, 620600 knots.

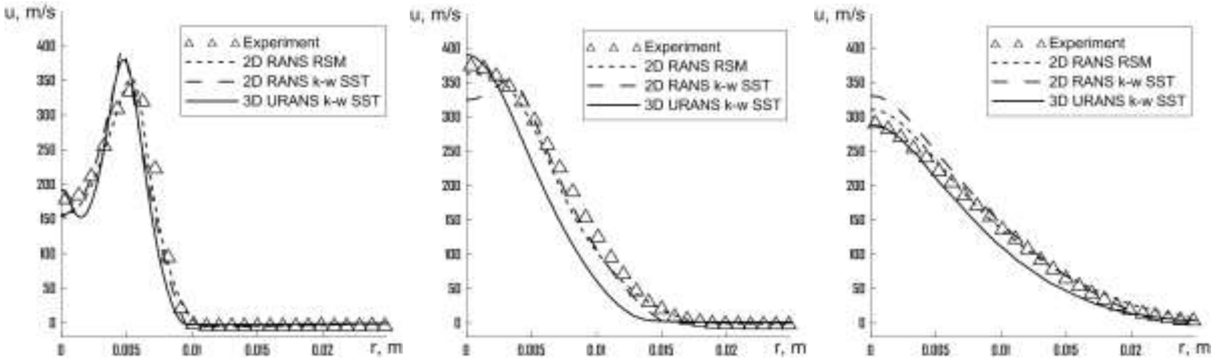
3. Results and discussion

Figs. 3-4 show velocity distribution at an overpressure in the nozzle P_0 = 1 bar and 3 bar for a gas task without addition of water. Comparison of calculation results with experiment showed that calculation have a good agreement with the experimental measurements. Comparing axial velocity profiles shows that the RANS RSM model better describes the experimental data than RANS k-w SST. Use of unsteady problem statement (URANS k-w SST) allows to increase accuracy of calculations, since it describes pulsating components of velocity more correctly.



z = 5 mm z = 40 mm z = 80 mm

Figure 3. Profiles of axial velocity at different distances from the nozzle (P_0 = 1 bar).



z = 5 mm z = 40 mm z = 80 mm

Figure 4. Profiles of axial velocity at different distances from the nozzle (P_0 = 3 bar).

Qualitative comparison of calculated results of atomization with experimental photos for a gas-liquid jet at a pressure of 1-3 bar is shown in Figs. 5-6. Experimental images were obtained by feeding water (flow rate 200 kg/h). The calculated images are visualized by concentration of water droplets. From comparison, on whole, a good qualitative agreement between calculation and experiment is observed. The calculation correctly describes opening angle of jet for different pressures.

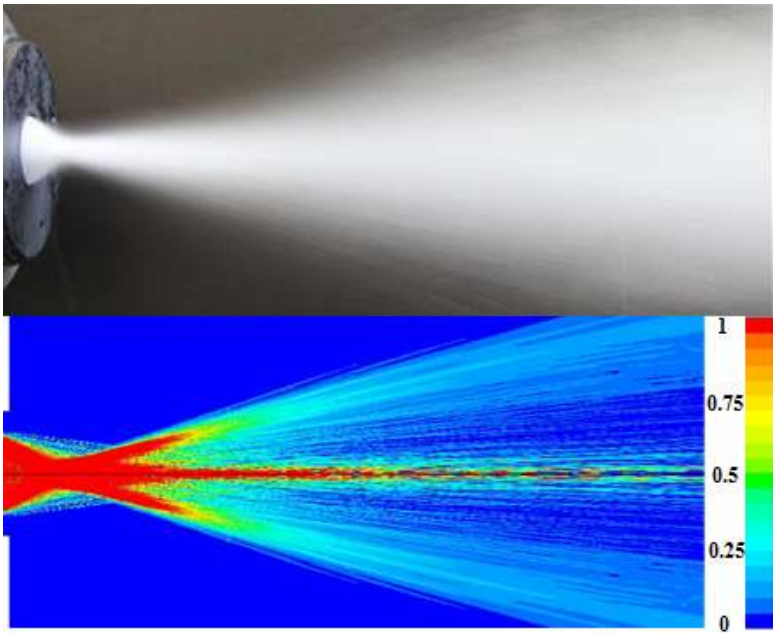


Figure 5. Gas-liquid jet, P_0 = 1: From above - Experiment; Below – Calculation (2D RANS RSM) (water concentration, kg/m^3).

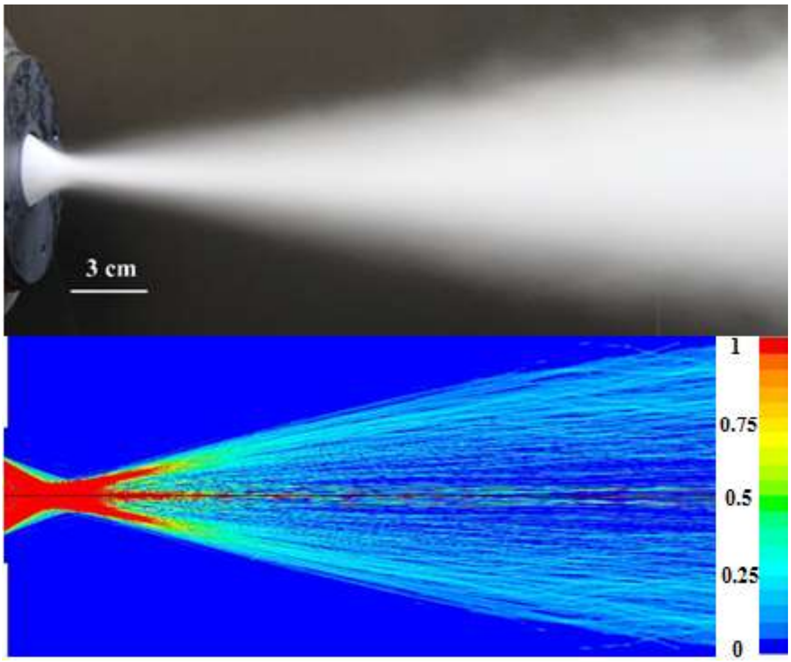


Figure 6. Gas-liquid jet, P_0 = 3: From above - Experiment; Below – Calculation (2D RANS RSM) (water concentration, kg/m^3).

4. Conclusion

We have performed verification of calculation modeling methods of atomizing of a gas and gas-liquid stream from a pneumatic nozzle. Comparative analysis showed an acceptable qualitative and quantitative agreement between calculation and experiment on main characteristics of atomization and may be useful in future for estimates, since it allows qualitatively correct determination of divergence angle of two-phase stream. Use of unsteady problem statement allows to increase accuracy of calculations, since it describes pulsating components of velocity more correctly.

The reported study was funded by RFBR according to the research project № 18-38-00594. The experimental part of the reported study was funded by RFBR according to the research project № 18-38-00153.

References

[1] Gorlov E.G., Seregin A.I., Khodakov G.S. Conditions of utilization of coal mining and processing sludges as slurry fuel // Solid Fuel Chemistry. 2007. T. 41. № 6. p. 364-369.
[2] Patrakov Y.F., Fedorova N.I., Efremov A.I. Composite water-containing fuel from low-grade coals of Kuzbass // Vestnik of Kuzbass State Technical University journal. 2006. № 3. P. 81–83.
[3] Arkhipov V. A., Sidor A. M., Surkov V. G. Examination of the physics-chemical and energy performances of organic water-coal fuils // Technical Thermophysics and Industrial Heat and Power Engineering. 2013. № 5. P. 39–47.
[4] Russian Federation Patent No.2346756 (2007), Pneumatic injector.
[5] Sergey Alekseenko, Igor Kravchenko, Leonid Maltsev.Combustion Technology for Water-fuel Mixtures // Proceedings of the 2012 International Conference on Thermal Treatment Technologies & Hazardous Waste Combustors, October 22-24, 2012, in New Orleans, LA. Paper No. 25. – 19 p.
[6] Gibson M.M., Launder B.E. Ground Effects on Pressure Fluctuations in the Atmospheric Boundary Layer // J. Fluid Mech. 1978. Vol.86. P.491–511.
[7] B. E. Launder, G. J. Reece, and W. Rodi. "Progress in the Development of a Reynolds-Stress Turbulence Closure". J. Fluid Mech. 68(3). 537–566. April 1975.
[8] Smirnov P.E., Menter F.R. Sensitization of the SST turbulence model to rotation and curvature by applying the Spalart-Shur correction term // Journal of Turbomachinery. 2009. V. 131. I. 4. P. 1-8.
[9] ANSYS Fluent 12.0 User's Guide, 2009