CHEMICAL ENGINEERING TRANSACTIONS

VOL. 70, 2018

A publication of
AIDIC
The Italian Association
of Chemical Engineering
Online at www.aidic.it/cet

Guest Editors: Timothy G. Walmsley, Petar S. Varbanov, Rongxin Su, Jiří J. Klemeš

ISBN 978-88-95608-67-9; **ISSN** 2283-9216

DOI: 10.3303/CET1870026

Comparative Study on Heat Generation During Compressible Airflow Through Heatproof Nozzles

Krzysztof J. Wołosz*, Jacek Wernik

Warsaw University of Technology, Faculty of Civil Engineering, Mechanics and Petrochemistry, Łukasiewicza 17, 09-400 Płock, Poland
Krzysztof.Wolosz@pw.edu.pl

The study considers industrially applicable nozzles which are used as an auxiliary equipment in pneumatic pulsator systems for unclogging outlets of silos for loose materials. The aim is to determine the amount of heat (energy loss) which is generated during one work cycle of the system. Presented study compares energy loss during compressible (max. M = 1.7) and transient airflow through two types of nozzles: with and without a baffle. Investigation in this field has not been carried out so far and present-day designing process is significantly based on heuristic knowledge. The energy loss is calculated by using results of numerical simulation. The OpenFOAM CFD toolbox with the finite volume method has been used for that purpose with thermodynamically ideal gas model. The energy analysis and heat generation calculation have been based on the first principles of thermodynamics. For initial and boundary condition, authors' previous work, which considered rapid airflow from a vessel to the atmosphere, has been partially utilized. Instantaneous results of simulation indicate the high energy rates but they are available for a very short time frame. The values of the total heat generated during gas conversion within the nozzle show that both nozzle produces a small amount of energy loss which would cause a slight warm-up of them.

1. Introduction

Investigations of a flow of compressible medium through a nozzle usually leads to a determination of flow parameters and actually possibility of their intensification. Energy analysis of gas flow and possibility of its utilization in the nozzles is not commonly carried out. The most frequent analyses of energy efficiency concern systems of equipment which are mostly used in industry. In this article, the authors compare the amount of energy lost in the nozzle of a pneumatic pulsator system. In systems for clearing silo outlets, compressed air is used for to destroy the unfavourable structure in loose material bed such as archings, bridges and others. The air, which is pressurized in the pressure accumulator, is rapidly released and a pneumatic impact is produced. This impact is a force that counteracts cohesion forces which are specific for loose and bulk materials. Systems for clearing silo outlets are an important element of safety and permeability of transport systems in industrial plants. Visualization of a silo with pneumatic pulsator system placed on its walls can be observed in Figure 1. To make the investigated object easier to understand, the nozzle is zoomed.

In pneumatic pulsator systems, nozzles are used to direct the airflow and often to protect the pulsator itself against the heat generated inside the silo. This is the case, for example, in cement plants in silos which contains clinker from the lime kiln. Different types of nozzles are used which differing in size, shape and function. The article presents a comparative study of two straight nozzles. The difference between them is a presence of reinforcing baffles placed along air stream in one of them.

In the nozzles, due to the principal of operation of the industrial pulsator, there is a compressible, unsteady airflow. Accompanying phenomena are related to thermodynamic gas conversions, which in their description must take into account the formation of shockwaves. The article by Honma et al. (2003) is of high value in this field due to the methods used and the comparison of experimental results with numerical ones. The authors considered unsteady outflow of gas to the atmosphere. Ota et al. (2012), and the related article by Inage et al. (2013), presented also very similar works in which the authors primarily focused on measurement methods and imaging capabilities of the shockwave and related phenomena.

Please cite this article as: Wołosz K.J., Wernik J., 2018, Comparative study on heat generation during compressible airflow through heatproof nozzles , Chemical Engineering Transactions, 70, 151-156 DOI:10.3303/CET1870026

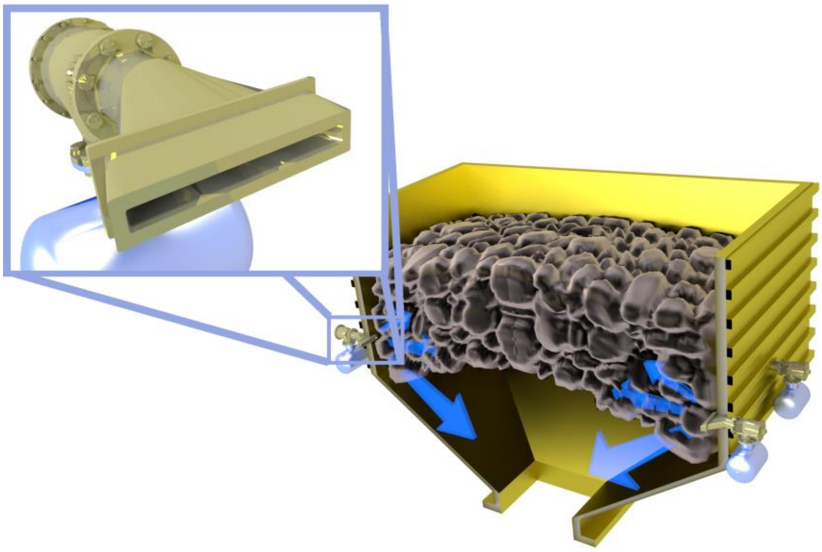

Figure 1: Pulsator system visualization on the background of a silo with the nozzle zoomed.

Investigations which consider nozzles and compressible flow mainly focus on the studies of Laval-type convergent-divergent nozzles. Due to the wide range of its applications, these studies most often focus on the effects of changes in geometry on the flow parameters and the studies consider specific industrial applications such as, for example, cooling systems (Lin et al., 2013), solar air conditioning systems (Hemidi et al., 2009) as well as improving the overall effectiveness of ejectors (Zhu et al., 2009).

Applications of research methods related to compressible flows in nozzles for heavy industry have been initiated by Wołosz and Wernik (2014) and then developed in a work focusing on the methodical aspects (Wołosz and Wernik, 2016b). The results of other studies related to pneumatic pulsators presented by Wołosz and Wernik (2016a) have also been used in this work, whereas the methods given by Wołosz (2018) to through-flow systems (with inlet and outlet) are developed.

The application of energy calculations is a novelty in systems in which the unsteady compressible gas flow is present. Literature review shows that researchers most often focus on nozzle parameters. However, they do not take into account the energy efficiency and its impact on a system in which these nozzles are located. The present study is an extension of the research to include these aspects. The article shows the magnitude of the energy losses associated with compressible transient airflow through a closed channel. It should also be emphasized that the energy efficiency is very rarely considered in this area of industrial issues.

2. Numerical Model

For the energy analysis, the geometric models of nozzles have been used. In both cases, the computational domain covers the inside of the nozzle with its symmetry plane taken into account. Each nozzle is mounted to a pulsator with a nominal diameter of 150 mm and the main dimensions are: length along the air stream 320 mm, width – 487 mm, and height – 325 mm. Figure 2 shows the visualization of nozzles cross-sections with the computational mesh applied. In both cases, the meshes exceeded 1,200,000 cells. Mesh generation as well as numerical calculation have been proceeded by using OpenFOAM. The OpenFOAM is an open source toolbox for Computational Fluid Dynamics (CFD) with finite volume method applied for solving governing equations of fluid flow.

Numerical simulation has been carried out by using formulas that are mathematical expression of basic conservation laws, as follows:

- mass conservation (continuity equation):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \tag{1}$$

- momentum conservation law:

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) - \nabla \cdot \mu \nabla \mathbf{u} = -\nabla p, \tag{2}$$

- energy conservation law:

$$\frac{\partial \rho e}{\partial t} + \nabla \cdot (\rho \mathbf{u} e) - \nabla \cdot \left(\frac{\lambda}{c_v}\right) \nabla e = p \nabla \cdot \mathbf{u}, \tag{3}$$

where ρ is density, **u** – velocity vector, μ – dynamic viscosity, p - pressure, t – time, e – internal energy of the gas, and λ denotes thermal conductivity.

a) b)

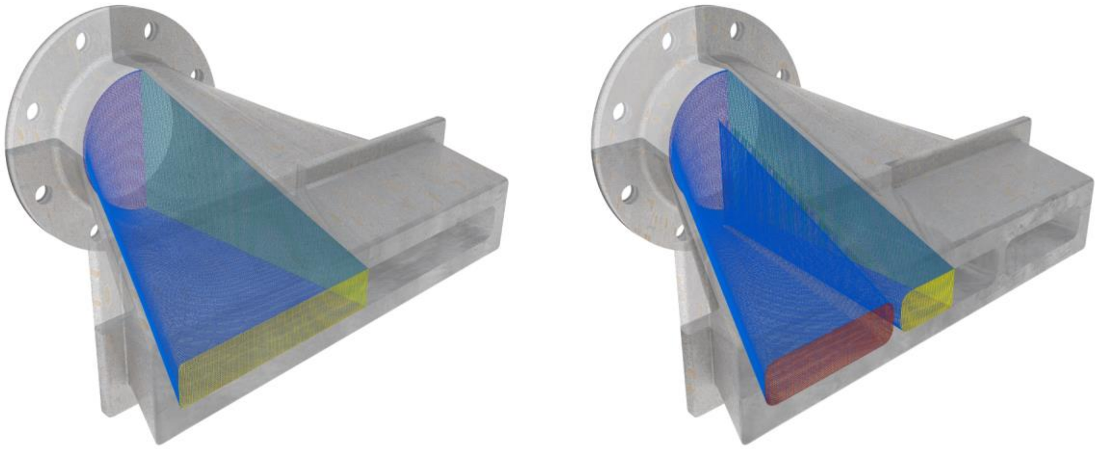

Figure 2: Nozzles with computational mesh applied: a) plain (without a baffle), b) with a baffle.

Eq(1) - Eq(3) are properly solved only for laminar flows. Air velocity reaches high values and the maximal Mach number is M = 1.7 in both investigated cases. Therefore, it is necessary to apply a model of turbulence to gain reliable results. For that purpose, standard k-ε model has been used (Launder and Jones, 1972) which is well validated in engineering aplications.
Air is a compressible medium which is commonly modelled as a gas which obey ideal gas equation:

$$R = c_p - c_v = \frac{p}{\rho T} = 287 \, \frac{\mathrm{J}}{\mathrm{kg} \cdot \mathrm{K}}, \tag{4}$$

where R – gas constant, c_p – specific heat at constant pressure, c_v - the one at constant volume, and T denotes temperature.
Internal energy of the gas is calculated by using the following expression:

$$e = c_v T, \tag{5}$$

and the entire aforementioned system of equations must be complemented by equation of heat conduction, which is Fourier's law:

$$\mathbf{q} = -\lambda \nabla T, \tag{6}$$

where **q** is heat flux density.
The system of equations Eq(1) - Eq(6) was numerically solved by using finite volume method. The equations were discretised by using an appropriate discretisation scheme. In this study for spatial discretisation, Gauss linear scheme was used, which is second-order accurate. Because the study considers transient airflow, precise, second-order accurate, Crank-Nicolson scheme for time discretisation was utilized.
Properly stated numerical problem can only be solved when boundary and initial conditions are applied. In the present study, the driving force of the flow is pressure difference on the both sides of the nozzles. The inlet pressure values come from the authors' previous study on emptying of the pressure accumulator (Wołosz and Wernik, 2016a). These results made it possible to calculate inlet values of pressure by using formula for calculating stagnation and static pressure ratio according to theory of shockwaves as follows:

$$p_p = \frac{p_0}{\left(1 + \frac{\gamma - 1}{2\gamma} \psi |\mathbf{u}|^2\right)^{\frac{\gamma}{\gamma - 1}}} \tag{7}$$

where p_p denotes patch (boundary) pressure, p_0 – stagnation pressure, and γ is adiabatic index. The value of pressure has a crucial influence on the value of energy of the gas at the inlet of the nozzle.
Other necessary boundary conditions were as follows:

- initial inlet and outlet temperature is 300 K (Dirichlet boundary condition),
- inlet pressure and velocity gradients equal zero (Neumann boundary condition),
- walls of the nozzle are assumed adiabatic and impermeable.

It is very important to proceed numerical model verification and solver validation during numerical simulations. For the solver validation, the benchmark model has been built in OpenFOAM on the basis of experimental setup reported by Giglmaier et al. (2013) and the results have been compared. The error estimation has been carried out by using procedure proposed by Freitas (2002).

3. Energy and Energy Loss

The amount of energy which is delivered to and received from the nozzle can be determined by using the First Law of Thermodynamics (FLT). For the purpose of this study, it is convenient to express this Law by its integral form, as follows:

$$\dot{E} = \frac{\partial}{\partial t}\int_V \rho\left(e + \frac{1}{2}\mathbf{u}\cdot\mathbf{u}\right)\mathrm{d}V + \oint_A \rho\left(e + \frac{1}{2}\mathbf{u}\cdot\mathbf{u}\right)\mathbf{u}\cdot\mathbf{n}\,\mathrm{d}A \tag{8}$$

The first integral in Eq(8) is of no value if there are neither chemical nor nuclear reaction, i.e. there are no internal heat sources. Airflow through the nozzle proceeds very fast, therefore, adiabatic gas conversion can be considered. With these assumptions taken into account, FLT looks as follows:

$$\dot{E} = \oint_A \rho\left(e + \frac{1}{2}\mathbf{u}\cdot\mathbf{u}\right)\mathbf{u}\cdot\mathbf{n}\,\mathrm{d}A \tag{9}$$

Because of impermeability of the nozzle walls, the boundary area A is limited to the inlet and the outlet of the nozzle. The values of density, temperature, pressure and velocity are changing along the considered area. Therefore, to predict total energy flux, the values have been area-weighted averaged, and the FLT is:

$$\dot{E} = \dot{m}\left(e + \frac{u_n^2}{2}\right) \tag{10}$$

where u_n is average velocity normal to the boundary area A.

4. Results and Discussion

The most important flow parameter regarding speed of the entire process is pressure, both inlet and outlet, and their changes in time. Averaged values of pressure for both types of investigated nozzle and their time relationship are shown in Figure 3. Furthermore, time dependence of stagnation pressure p_0, which for both types of nozzle has been the same, is shown in Figure 3a. It can be noticed that there is a difference between inlet pressure values after simulation time of 0.4 ms. From that time, inlet pressure is lower in the nozzle with the baffle then in the one without it. Inlet pressure is calculated by using Eq(7), so the baffle causes velocity increase by narrowing flow cross-section. This is the case where flow parameters influence the inlet boundary conditions. That is why it is very important to state the initial and boundary conditions properly.

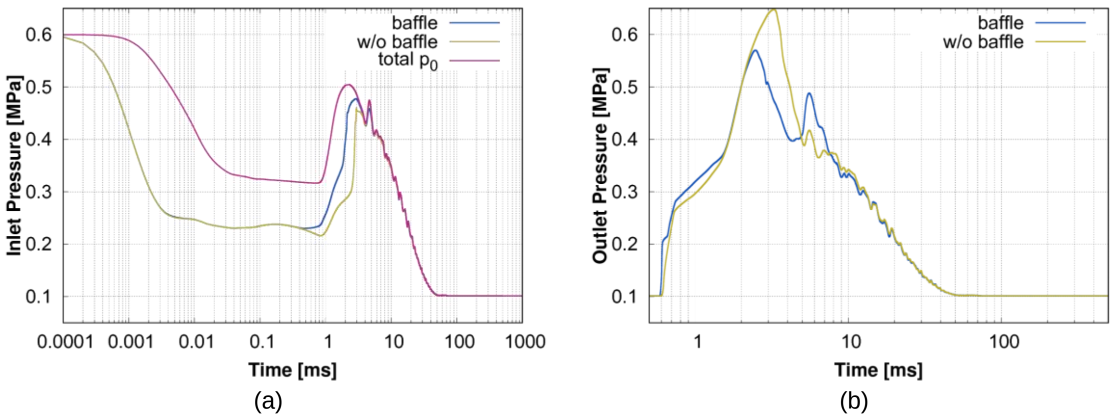

Figure 3: Pressure time distribution at: a) inlet, b) outlet of the nozzles.

The energy analysis is subsequently carried out based on the results of numerical simulation. The values of energy fluxes at inlets and outlets of each nozzle are obtained in this analysis. The plots of these fluxes and their time relationship can be seen in Figure 4. It should be noticed that the scale of the horizontal axis differs slightly for the inlet and outlet plots. Because the time needed to overcome the distance of the nozzle length is about 0.5 ms, the outlet values equal initial ones up to this time. Therefore, due to the increased readability of the plots, the scale of the horizontal axis is shortened in Figure 3 and 4.

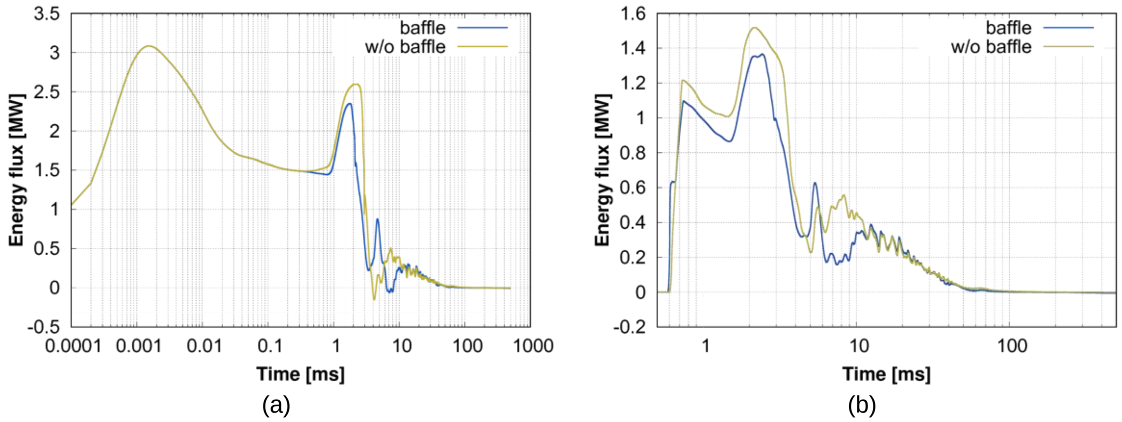

Figure 4: Time distribution of energy flux at inlet and outlet of the nozzle.

Inlet energy fluxes for both nozzle types start to differ since 0.4 ms of simulation time. It follows directly from the difference of inlet pressure what has been mentioned before. Therefore, it might be claimed that the presence of the baffle generates inlet gas energy decrease by causing inlet pressure decrease.
The highest recorded value of the inlet energy flux is above 3 MW. This value is more than twice less at the outlet, i.e. approx. 1.5 MW. The energy fluxes shown in the plots of Figure 4 do not give the answer what is the total energy of the process for both nozzles. Due to slightly different energy flux characteristics, it is difficult to directly estimate the energy utilization and compare it between both types of nozzle. Furthermore, the highest energy flux is available at very short time frame. For example, 3 MW are available only for period 0.001 ms. For that reason, the total energy must be determined for the whole process from 0 to 500 ms, i.e. the time after which the flow in the nozzles stops. Simulation time could be shorter, however, to restrict the calculation error and due to slow coincidence of temperature results, it was decided to extend simulation time to 500 ms.
Time derivative of energy gives energy flux, thus, to calculate total energy at inlets and outlets, the energy fluxes need to be integrated in time. Since numerical simulation gives discretised results, the integral can be converted into sum as follows:

$$E = \int_0^{t_1} \dot{E}\, \mathrm{d}t = \sum \dot{E} \cdot \Delta t \tag{11}$$

After this operation, the values of total energy can be determined for inlets and outlets of both types of nozzle and these values can compared in order to determine the energy loss according to the following relation:

$$\Delta H = E_{inlet} - E_{outlet} \tag{12}$$

The results of total energy at inlets and outlets and heat generated (energy loss) are presented in Table 1 below:

Table 1: Results of energy efficiency analysis

Nozzle type	Inlet energy [J]	Outlet energy [J]	Energy loss [J]	Energy loss [%]
w/ baffle	10,399	10,322	76.9	0.74
w/o baffle	14,059	13,828	230.7	1.64

There is a noticeable lower inlet energy for the nozzle with the baffle which is obvious since the presence of the baffle itself. However, the lower value of losses for the nozzle with the baffle is unexpected. The presence of the baffle should suggest an increase in losses. Percentage comparison shows that the nozzle without the baffle generates more energy loss. This might suggest that there is a chocked flow inside the nozzle, which generates additional losses.

5. Conclusions

During this study, a method has been developed for evaluating how much energy, which is carried with compressed air, is lost during the unsteady, compressible flow through the heatproof nozzles of an industrial pneumatic pulsator system. The amount of these losses, which must be converted into heat eventually, is regarded as small. Depending on the nozzle type, the amount of heat generated in the nozzle does not exceed 1 % for the nozzle with the baffle and 2 % for the nozzle without the baffle. These values are not large and should not affect on the quality of the nozzle functioning. What may affect the quality of their functioning are the conclusions resulting from the comparison of the values obtained for individual nozzles. In spite of expectations, the energy loss is greater when using the nozzle without the baffle. This will lead to further research on the use of other nozzle shapes. However, a simple conversion towards the de Laval nozzle cannot take place due to the unsteady nature of the pneumatic impact phenomenon, which is encountered in industrial pneumatic pulsators. Therefore, in future works, it is also expected to perform flow simulations established for both considered nozzles using the initial boundary conditions of this study and to compare the energy values for steady and unsteady flows.

References

Freitas C.J., 2002, The issue of numerical uncertainty, Applied Mathematical Modelling, 26, 237–248.

Giglmaier M., Quaatz J.F., Gawehn T., Gülhan A., Adams N. A., 2013, Numerical and experimental investigations of pseudo-shock systems in a planar nozzle: impact of bypass mass flow due to narrow gaps, Shock Waves, 24, 139–156.

Hemidi A., Henry F., Leclaire S., Seynhaeve J.M., Bartosiewicz Y., 2009, CFD analysis of a supersonic air ejector. Part I: Experimental validation of single-phase and two-phase operation, Applied Thermal Engineering, 29, 1523–1531.

Honma H., Ishihara M., Yoshimura T., Maeno K., Morioka T., 2003, Interferometric CT measurement of three-dimensional flow phenomena on shock waves and vortices, Shock Waves, 13, 179–190.

Inage T., Tsuchikura S., Ota M., Maeno K., 2013, Three-dimensional laser interferometric CT (LICT) measurement of shock wave interaction around a circular cylinder, Flow Measurement and Instrumentation, 31, 102–106.

Launder B.E., Jones W.P. , 1972, The prediction of laminarization with a two-equation model of turbulence, International Journal of Heat and Mass Transfer, 15, 301–314.

Lin C., Cai W., Li Y., Yan J., Hu Y., Giridharan K., 2013, Numerical investigation of geometry parameters for pressure recovery of an adjustable ejector in multi-evaporator refrigeration system, Applied Thermal Engineering, 61, 649–656.

OpenFOAM®, 2017, The open source CFD toolbox, Ver. 4.1, <www.openfoam.org> accessed 7.01.2017.

Ota M., Udagawa S., Inage T., Maeno K., 2012, Interferometric measurement in shock tube experiments, In I. Padron, Ed., Interferometry - Research and Applications in Science and Technology, 226–244.

Wołosz K.J., 2018, Exergy destruction in the pneumatic pulsator system during one working cycle, Energy, 146, 124–130.

Wołosz K.J., Wernik J., 2014, Heat generation calculation on the basis of numerical simulation results of supersonic airflow in a nozzle, Chemical Engineering Transactions, 39, 1363–1368.

Wołosz K.J., Wernik J., 2016a, Exergy analysis of the air evacuation process from the pressure accumulator of the pneumatic pulsator system, Chemical Engineering Transactions, 52, 19–24.

Wołosz K.J., Wernik J., 2016b, On the heat in the nozzle of the industrial pneumatic pulsator, Acta Mechanica, 227, 1111–1122.

Zhu Y., Cai W., Wen C., Li Y., 2009, Numerical investigation of geometry parameters for design of high performance ejectors, Applied Thermal Engineering, 29, 898–905.