TECHNICAL TRANSACTIONS | CZASOPISMO TECHNICZNE
MECHANICS | MECHANIKA

2-M/2015

PIOTR CYKLIS, PRZEMYSŁAW MŁYNARCZYK*

THE ROLE OF THE CFD MODELLING IN THE SHAPE OPTIMIZATION OF THE PRESSURE PULSATIONS DAMPERS

ROLA MODELOWANIA CFD W OPTYMALIZACJI KSZTAŁTU TŁUMIKÓW PULSACJI CIŚNIENIA

A b s t r a c t

Pressure pulsations in volumetric compressors systems have an important influence on the compression power and reliability. This problem occurs not only in huge compressor systems, as in the natural gas piping, in gas mines or national transport systems, but also in a small refrigeration compressors in domestic applications. Nowadays, it is necessary to develop pressure pulsation damping elements which will be effective for different mass flow rate values. One of the solutions are passive damping elements in the form of various types of nozzles. In addition to the decrease of pressure pulsations value, the result is also an increase in power consumption of the compression. Therefore, shape optimization is necessary. In the article the possibility of optimizing the shape of such elements using computational simulations is discussed.

Keywords: *CFD simulations, Pressure pulsations damping, nozzle gas flow*

S t r e s z c z e n i e

Pulsacje ciśnienia w instalacjach sprężarek wyporowych mają duży wpływ na ich moce sprężania oraz niezawodność. Problem ten pojawia się nie tylko w dużych systemach sprężarkowych, jak na przykład w sprężarkach gazu ziemnego w kopalniach i rurociągach transportowych, ale również w małych sprężarkach chłodniczych w zastosowaniach domowych. Aktualnie konieczne jest opracowanie elementów tłumiących pulsacje ciśnienia dla różnych wartości strumienia przepływającego czynnika. Jednym z rozwiązań są pasywne elementy tłumiące w postaci różnego rodzaju zwężek. Poza spadkiem wartości pulsacji ciśnienia efektem jest również wzrost poboru mocy sprężania. Wymagana jest więc optymalizacja kształtu takich elementów. Niniejszy artykuł rozważa możliwość przeprowadzania optymalizacji kształtu pasywnych tłumików pulsacji ciśnienia przy pomocy symulacji komputerowych.

Słowa kluczowe: *Symulacje CFD, Pulsacje ciśnienia, przepływ gazu w dyszy*

* Prof. Ph.D. D.Sc. Eng Piotr Cyklis, M.Sc. Eng. Przemysław Młynarczyk, Faculty of Mechanical Engineering, Cracow University of Technology.

Nomenclature

f_D	–	dimensionless Darcy's friction factor from Moody's diagram or Colebrook's equations
L	–	pipe length
D	–	pipe internal diameter
ρ	–	fluid density
v	–	average flow velocity
εr	–	relative roughness of the wall
e	–	roughness height
Re	–	Reynolds number
$\dot{M}$	–	mass flow rate
α	–	flow number
ε	–	expansion number
A_d	–	area of the nozzle cross-section
Δp	–	pressure difference (pressure drop)
v_1	–	specific volume before the nozzle

1. Introduction

Periodic working machines like volumetric compressors are among the most popular machines in industry. Pressure pulsations attenuation in volumetric compressor systems are constantly one of the most important problems in a compressed gas manifold. Most of the phenomena caused by pressure pulsation, like vibrations, acoustic noise, energy loss, and valves wear are harmful to the installation and its components. A wide range of compressors, from large, low-speed natural gas compressors to small, compact refrigeration compressors with variable speed makes it difficult to find a universal solution. In all cases pressure pulsation reduction is an important engineering task. The standard way to attenuate pressure pulsation is to apply the Helmholtz theory for the muffler design as it is described in [1, 2].There is still no practical solution to suppress different values of pressure pulsation with variable mass flow rate over time using one compact damping element. The CFD application for simulation of constant and impulse flows through the nozzle can give an answer how the nozzle affects the pulsating flow. Investigations on flow pulsations damping are also shown in literature nowadays [3–5]. Especially simulations of a nozzle flow are widely presented in the publications [6, 7]. The CFD simulation has been used by [8] for the simulation of a single pipe internal flow excited with a single disturbance. It has been shown that the response which is periodic with a constant frequency is characterized by a certain degree of damping.

The core of this article is the answer to the question whether the CFD simulations can be used to optimize the shape of pressure pulsations dampers.

2. Nozzle designs

For simulation and experimental investigation several nozzle designs have been prepared. The passive pressure pulsations damping was analysed with specially shaped hyperboloidal nozzles and normalized shaped nozzles like Venturi orifice and Venturi nozzle as it is shown in the Figure 1. Three different internal diameters were chosen: φ10, φ15 and φ20. After preliminary experimental investigations and CFD simulations the most promising shapes were tested in both directions flow and as a configuration of two coupled elements. The key element of this investigation is the assessment of the nozzle's influence on the pulsation attenuation and pressure drop on the basis of a computer simulation and the results compared with the outcomes of the experimental investigations. With normalized shapes computational results may be checked by standard, known analytical equations.

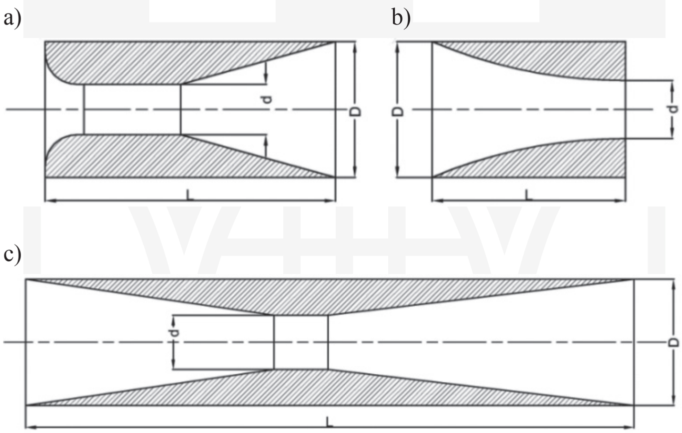

Fig. 1. Shape and main dimensions of the a) Venturi nozzle, b) Hiperboloidal nozzle, c) Venturi orifice

3. CFD results verification methods

The desired effects of the designed nozzle impact on pressure pulsations is a reduction in pressure pulsation to at least 20% with insignificant influence on the compressor's power with the same volumetric flow rate.

For this task the CFD simulation method could be selected as a tool for finding the best possible solution. To achieve this goal two different simulation results have to be computed: the unsteady simulation which shows the influence on the pressure pulsation and the steady state simulation which corresponds with the flow restriction affecting the compressor's power.

3.1. Experimental investigations

In the experimental investigations of pressure pulsations damping the nozzles were placed at the discharge of screw compressor DS-40 CompAir. The screw compressor was powered by a diesel engine with a variable speed control. The laboratory test stand is shown in Fig. 2.

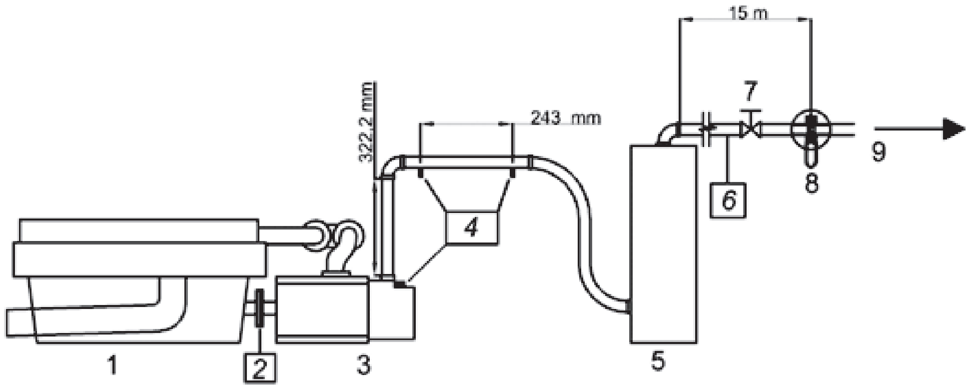

Fig. 2. Measuring system, where: 1 – engine, 2 – torque meter on the propeller shaft, 3 – screw compressor, 4 – pressure pulsations sensors, 5 – oil separator, 6 – static pressure transducer, 7 – throttle valve, 8 – metering orifice, 9 – outflow

The nozzle is located 17mm above the compressor discharge chamber. The ICP Dynamic Pressure Sensors made by PCB Piezotronics were used. The signal from sensors goes through the 4-channel ICP Sensor Line Power unit to the NI USB-6251 data acquisition module. The LabView data acquisition software was used to acquire the pressure pulsation data.

3.2. Analytic equations

The results of the steady state CFD flow simulation have been compared for validation with the standard known formulas for normalised shapes. The normalised, known methods for pressure loss in a pipe or astandard nozzle are shown below:

– the Darcy-Weisbach method for tube pressure loss under steady flow conditions [9]:

$$\Delta p = f_D \cdot \frac{L}{D} \cdot \frac{\rho V^2}{2} \tag{1}$$

Where the Darcy's friction factor could be taken from the Moody's diagram or calculated, for example, using the Haaland formula:

$$f_D = -1.8 \log_{10}\left[\left(\frac{\varepsilon r}{3.7}\right)^{1.11} + \frac{6.9}{\mathrm{Re}}\right] \tag{2}$$

or the Swamee-Jain equation:

$$f_D = 0.25\left[\log_{10}\left(\frac{e}{3.7D} + \frac{6.9}{\mathrm{Re}}\right)\right]^{-2} \tag{3}$$

– For Venturi orifice and nozzle ISO5167 a standard formula may be applied for pressure drop on the element:

$$\Delta p = \frac{\dot{M}^2 v_1}{2\alpha^2 \varepsilon^2 A_d^{\,2}} \tag{4}$$

4. CFD Procedures

For the CFD simulations the ANSYS/FLUENT software was used. For all cases a 2D axisymmetric, ideal gas isentropic flow model was used with the Reynolds-Stress turbulence model. Three to six boundary layers at the wall and two areas with a different element size were modelled. The same geometry was used for a steady and unsteady (impulse flow) simulation.

Boundary conditions for an unsteady impulse flow:

– At the inlet mass flow impulse excitation 0.1 [kg/s]. The impulse excitation means that its duration is equal to one time step ($2e^{-06}$ s),
– Pressure outlet where the pressure at the outlet was defined as the arithmetical average between the pressure outside the domain and the last cell inside the domain,
– Wall (also for closed end elements) where tangential stresses were included in the momentum conservation equation.

Next, the results were spatially averaged at the inlet and outlet to obtain one dimensional flow and pressure pulsations functions, which are the result of the impulse flow excitation. Damping coefficient ξ, free frequency ω, delay time Δτ, and amplification coefficient K were estimated for each free frequency. The oscillation response has the form shown in Fig. 3.

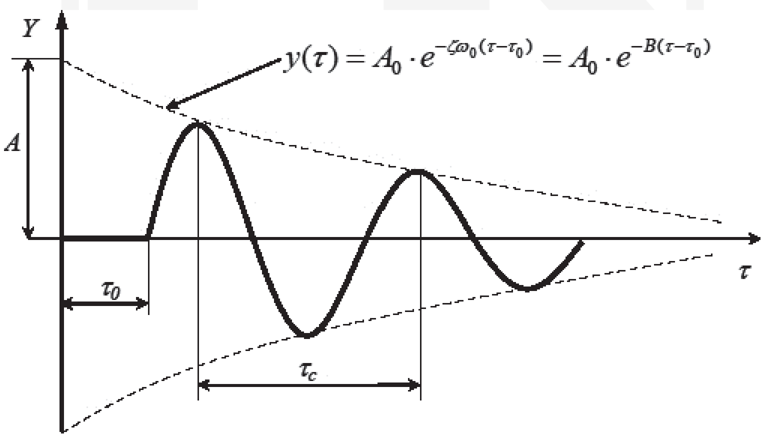

Fig. 3. Damped oscillating function and graphic presentation of its parameters

The analysis of the steady-state flow through the nozzle gives the pressure drop on the element which could be compared to the experimentally received mass flow to the power (V/N) coefficient's value. The pressure loss on the element has been related to the average pressure in the discharge compressor manifold and the pressure loss on a straight empty pipe (pipe0) using the following formula:

$$\beta = \frac{\Delta p_{nozzle} + p_{discharge}}{\Delta p_{pipe0} + p_{discharge}} \tag{5}$$

Convergent results were obtained for dense Cartesian mesh, using a very high number of elements with the so-called "metrics" above 0.95 for the orthogonal elements. The averaged value of "metrics" for elements above 0.85. The dense mesh caused no convergent results with lower "metrics". The mesh fragment for the Venturi nozzle is shown in the 4. The points: p_0 and p_1 are the pressure measuring points and the placement of these points is defined in the ISO 5167 standard formula.

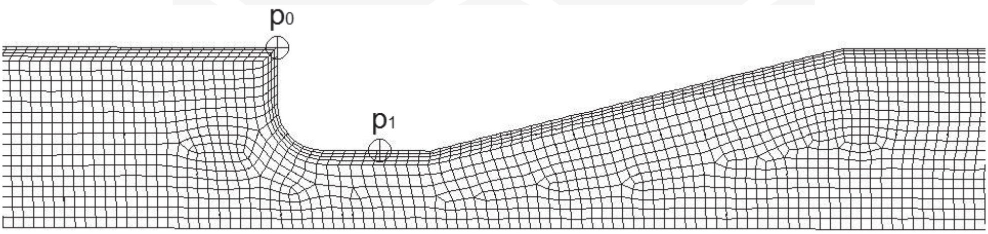

Fig. 4. Venturi nozzle mesh used for simulation

5. Results comparison

Results of pressure drop obtained in the steady-state flow for normalised shapes are compared to analytical calculations and are shown in Fig. 5.

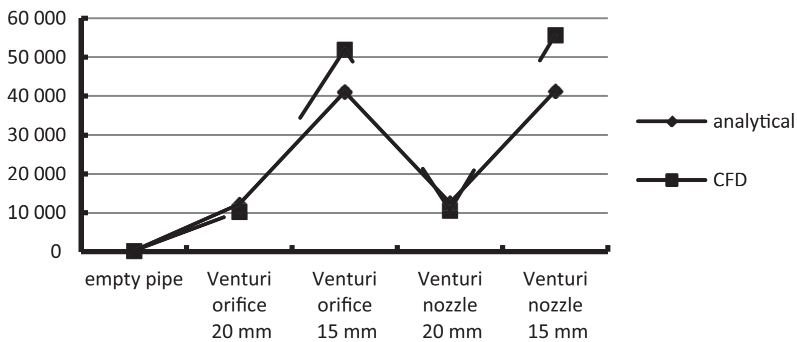

Fig. 5. Comparison of the CFD results with the analytical and standard flow measurement method results for standard shapes

The experimental investigations described in section 3.1 were performed for two different revolution speeds: 1615 rev/min and 2100 rev/min. In the article the results for the higher speed are described. It has been assumed that V/N is related to the steady flow pressure loss on the element, therefore the steady flow through the element was simulated and the pressure loss was compared with an empty pipe pressure loss during steady flow.

The presented results show that, especially for the elements of one shape group, results from the experiment and from the CFD simulations agree qualitatively. The results are shown in Fig. 6 – experiment and Fig. 7 – CFD.

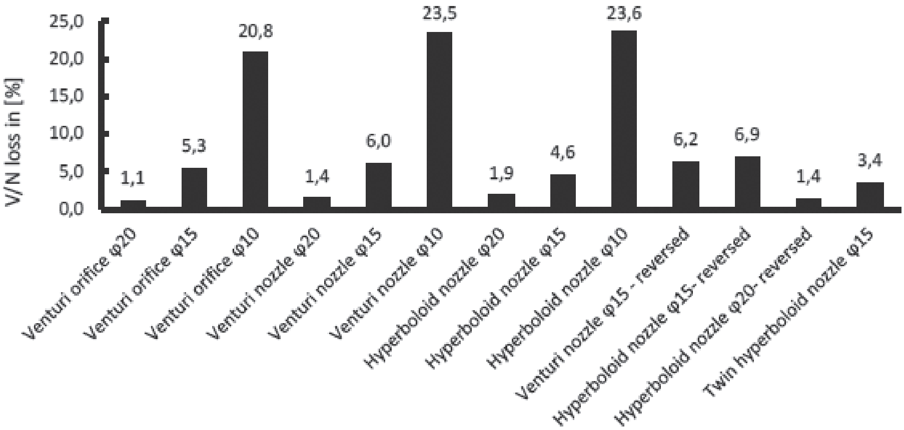

Fig. 6. V/N power loss in [%] compared to empty pipe for 2100 rev/min

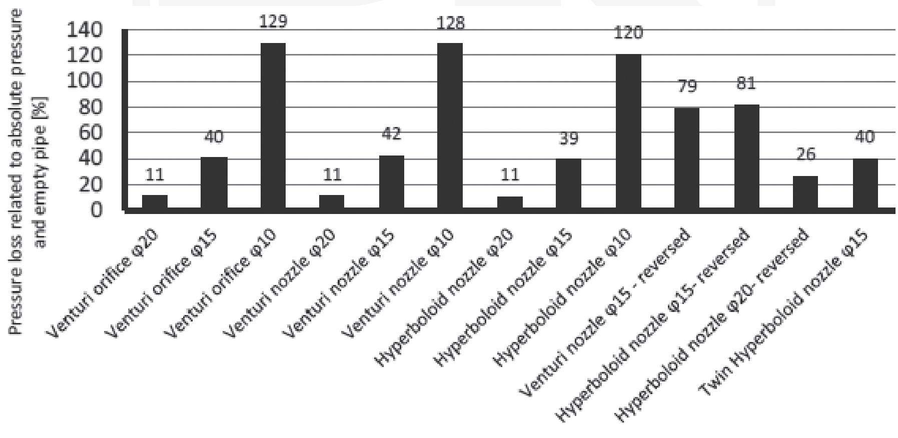

Fig. 7. Relative pressure loss β on damping element in comparison with the pressure loss on an empty straight pipe in [%]

6. Conclusions

The possibility to simulate pressure pulsations damping by passive elements is shown in the paper. The CFD simulation results were compared with experimental data and with results obtained in calculations of standard formulas. Due to the limited amount of space the article shows only the relationship between V/N and β loss. CFD results are not directly the same as experimentally determined parameters, because of the compressor system complex and the simplified CFD model. Referring to the growing number of compressors with a variable revolution speed, it is necessary to find the right shape of the damping element for a certain range of the mass flow rate, therefore the shape optimization is necessary. Experimental investigations provide accurate results but impose serious limitations. Standard formulas are limited to use only for typical shapes. It has been shown that the CFD simulation can be used as a tool to optimise the shape or dimension of the nozzle. According to the results, the conclusion is that the CFD's role in pressure pulsations dampers shape optimization is possible and necessary.

References

[1] Cyklis P., *Identyfikacja termodynamiczna elementów instalacji sprężarek wyporowych. Uogólnienie modelu Helmholtza*, Monography, nr 254, Wydawnictwo Politechniki Krakowskiej, Kraków 1999.

[2] Ma Y.-C., Min O.-K., *Pressure calculation in a compressor cylinder by a modified new Helmholtz modeling*, Journal of Sound and Vibration 243, 2001, 775-796.

[3] Lee, S-H., Ih, J-G., *Effect of non-uniform perforation in the long concentric resonator on transmission loss and back pressure*, Journal of Sound and Vibration 311, 2008, 280-296.

[4] Hermann J., Koreck J., Maess M., Gaul L., von Estroff O., *Frequency-dependent damping model for the hydroacoustic finite element analysis of fluid-filled pipes with diameter changes*, Mechanical Systems and Signal Processing 25, 2011, 981-990.

[5] Sadamoto A., Tsubakishita Y., Murakami Y., *Sound attenuation in circular duct using slit-like short expansion of eccentric and/or serialized configuration*, Journal of Sound and Vibration 277, 2004, 987-1003.

[6] Shademan Y., Barron M., Balachandar R., *CFD Study of effects of geometry variations on flow in a nozzle*, Engineering Applications of Computational Fluid Mechanics, Vol. 6, No. 3, 2012, 412-425.

[7] Torregosa A., Fajardo P., Gil A., Navarro R., *Development of non-reflecting boundary condition for application in 3D computational fluid dynamics*, Engineering Applications of Computational Fluid Mechanics, Vol. 6, No. 3, 2005, 447-460.

[8] Sekavcnik M., Ogorevc T., Skerget L., *CFD analysis of the dynamic behaviour of a pipe system*, Forsh Ingenieurwes 70, 2012, 139-144.

[9] *Engineering toolbox*, http://www.engineeringtoolbox.com/darcy-weisbach-equation-d_646.html (date of access: 2014-11-21).