Analysis of the possibility to optimize the passive pressure pulsations dampers using CFD

MSc eng. Przemysław Młynarczyk

Supervisor: Prof. dr hab. inż. Piotr Cyklis

Abstract

Vibration and noise caused by pressure pulsations in volumetric compressor manifolds are one of the most important problem in compressor operation. Passive damping of those pulsations is possible using specially shaped nozzle placed in place of the straight tube. Experimental analysis of the pressure pulsations damping caused by a nozzle is possible however only some nozzle shapes can be investigated. In this case the CFD simulations could be helpful. Some parameters obtained in the experimental investigations in the compressor manifold can be related with the parameters obtained in the steady state flow nozzle simulation. In this paper the experimental and numerical results comparison is showed and, thus, the analysis of the possibility to use CFD simulations to optimize the nozzle shape is described.

Keywords

Compressors, Pressure pulsations, CFD

1. Problem analysis

Periodic work flow of the volumetric compressors causes the pressure and mass flow pulsations. Pressure pulsations in volumetric compressor manifolds have a high impact on compression power requirement and the reliability of manifold operation. They induce vibrations, noise and in some cases even mechanical failure of piping or compressor valves. Nowadays there is a lot of publications that describes pressure pulsations as a one of the major problem in the compressor manifold operation [1][2]. Widely introduced variable revolution speed compressors makes it difficult to find a universal solution that will attenuate pressure pulsations in a wide range of compressor revolution speed so other possibilities of pressure pulsations damping than the standard way are really wanted [3]. In some cases there is no need to supress the pressure pulsations in high rate but attenuation in the 20%-30% is desirable [4]. The point is to suppress the pulsations generated at various operating parameters of the compressor. There are many articles described that the CFD application for simulation of impulse and constant flows through the nozzle can give estimation of the nozzle effect in advance [4][5][6]. In this article the comparison of some parameters obtained in the experimental results and numerical simulations results are presented.

2. Investigated geometries

For simulation and experimental investigation several nozzle designs have been prepared. The shapes are two Venturi nozzles and two Venturi orifices, which are normalized shapes described in the ISO standards, and two Hyperboloidal nozzles. All of the nozzles have the outter diameter equal to 35 milimeters and the only difference between two the same nozzles is inner diameter which is 20 or 15 mm. The nozzle shapes are shown in the fig. 1. There are also investigated combination of this shapes, like twin hyperboloid nozzles in a different combinations, and the nozzles reversed shapes.

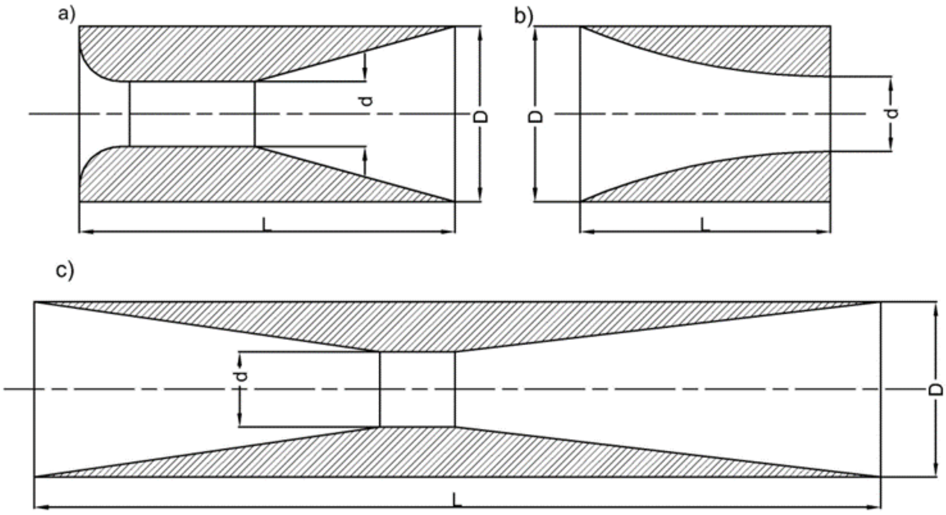

Fig. 1. *Shape and main dimensions of the a) Venturi nozzle, b) Hiperboloidal nozzle, c) Venturi orifice*

3. Experimental investigations

In the experimental investigations the DEMAG screw compressor powered by diesel engine with the variable speed control was used. The unit consist of 3 cylinders, 2.2 liters engine, with nominal revolution speed equal to 3000 revolutes per minute and screw compressor which nominal efficiency is 4.2 cubic meters per minute at 7 bars discharge pressure. In the fig. 2. The test stand is shown. In the experimental investigations of pressure pulsations damping the nozzles were placed 17 mm above the screw compressor discharge chamber.

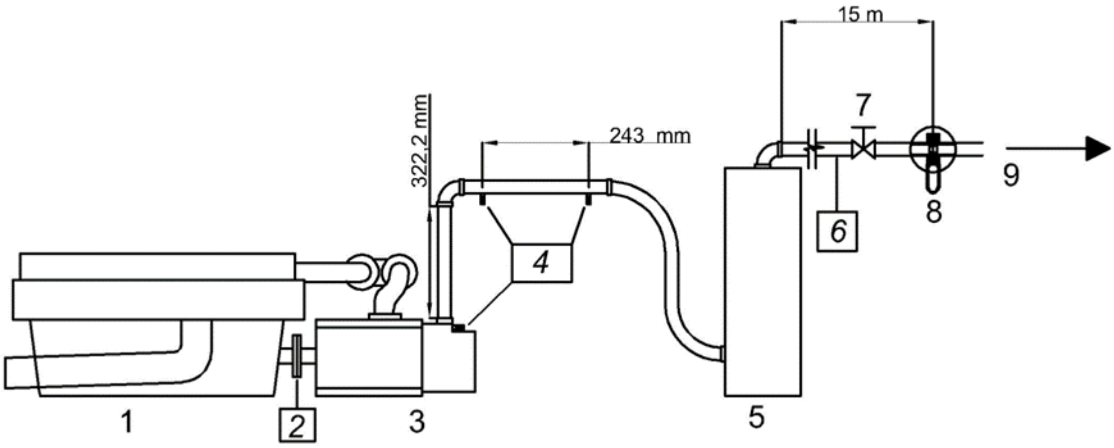

Fig. 2. Measuring system, where: 1- Engine, 2- Torque meter on the propeller shaft, 3- Screw compressor, 4- Pressure pulsations sensors, 5- Oil Separator, 6- Static pressure transducer, 7- Throttle valve, 8- Metering orifice, 9- Outflow

The 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. The measurements data were recorded as soon as the compressor and manifold reached steady operation.

4. Simulation procedure and benchmark test

For the numerical calculations the Ansys Fluent software was used. Simulation procedure for all investigated shapes was determined as an ideal gas isentropic flow in the two-dimensional axisymmetric model. Geometry for all cases was designed as a combination of the straight pipe and passive damper shape. The mesh model contains two areas with different element size. In the area of nozzle geometry smaller elements were defined than in the straight pipe geometry. The Raynolds stress model was chosen as a most accuracy turbulence model. Mesh detail are shown in the fig.3.

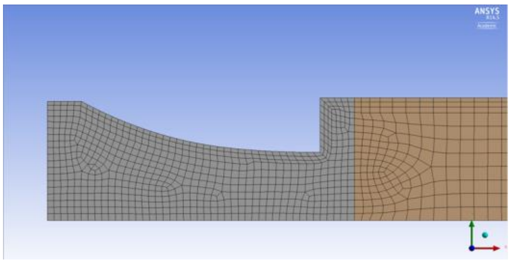

Fig. 3. *Hyperboloidal nozzle φ20 mesh detail*

Two different numerical calculations were performed for each element. In this paper the steady state flow simulations results are described. At the pipe inlet the mass flow equal to 0.05 [kg/s] has been applied. Pressure drop obtained in the steady state flow simulations can be related to the pressure drop in the installation during experimental investigations or to the other corresponding parameter. To check the prepared flow codes simply benchmark test was conducted. Numerical simulations results of the pressure drop for standard nozzles shapes were compared to the analytical results calculated according to the normalised, known methods for pressure loss in a pipe or standard nozzle. This methods are shown below:

- Darcy-Weisbach method for tube pressure loss under steady flow conditions [7]:

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

where Darcy's coefficient is taken from Moody's diagram or Colebrook's equations:

- according to Haaland:

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

- or Swamee-Jain:

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

- For Venturi orifice and Venturi nozzle ISO 5167 standard formula may be applied, for pressure drop on the nozzle :

$$\Delta p = \frac{\dot{m}^2 v_1}{2\alpha^2 \varepsilon^2 A^2} \tag{4}$$

In fact, this comparison is not exactly the same as a real nozzle with the inlet and outlet straight pipe required, therefore some differences from the results of standardised flow measurement method can be noticed. The results is shown in the table 1.

Table 1. *– Benchmark test results*

Shape	***Analytical [Pa]***	***CFD [Pa]***
Empty pipe	**194**	**219**
Venturi orifice φ20mm	**12201**	**11166**
Venturi orifice φ15mm	**41098**	**36825**
Venturi nozzle φ20mm	**11634**	**10631**
Venturi nozzle φ15mm	**41345**	**39848**

To conduct the preliminary simulations the author considered that the results with a maximum error about 10% are satisfactory.

5. Results comparison

The main assumption of this paper is to analyse the possibility to optimize the passive pressure pulsation damper using CFD. Therefore there is a need to assess the nozzle influence on the fluid flow. In the numerical calculations such a parameter can be a pressure drop on the element. In the experimental investigations the important parameter for the assessment of nozzle influence is the increase of power consumption due to flow restriction by the nozzle. To make this assessment the volumetric flow rate to compressor shaft power ratio was chosen. This parameter is increasing with lower energy consumption and decreasing with lower volume flow rate. Results of the experimental obtained V/N coefficient is shown in the fig. 4. for compressor revolution speed equal to 1615 rev/min and in the fig. 5. for 2100 rev/min. All the results are percentage increase or loss compared to an empty pipe V/N ratio value.

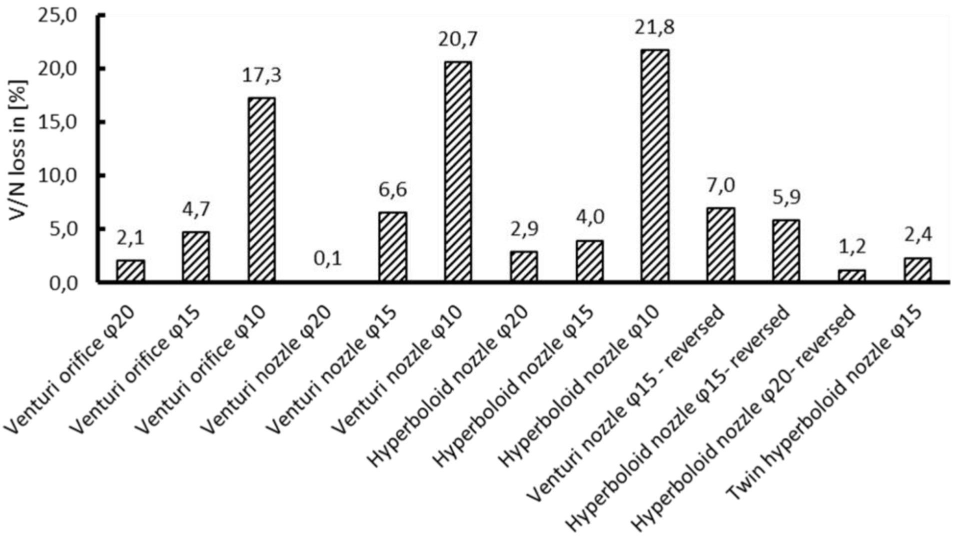

Fig. 4. *V/N power loss in [%] compared to empty pipe for 1615 rev/min*

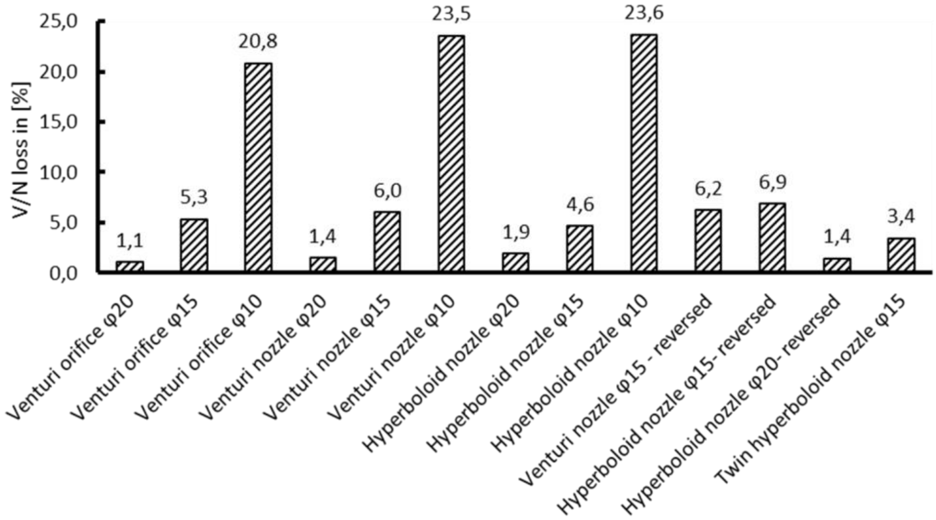

Fig. 5. *V/N power loss in [%] compared to empty pipe for 2100 rev/min*

In terms of values V/N parameter increase indicates compressor's better overall performance. 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. To have a better comparison to the simulation results the pressure drop 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:

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

Coefficient β is shown in Fig. 6. This figure shall be compared with the experimental results shown in Figs 4 and 5. The exact match here with the figures cannot be expected since this is not the same parameter, but the relations between the results for each shape and dimension are similar to the experimental results.

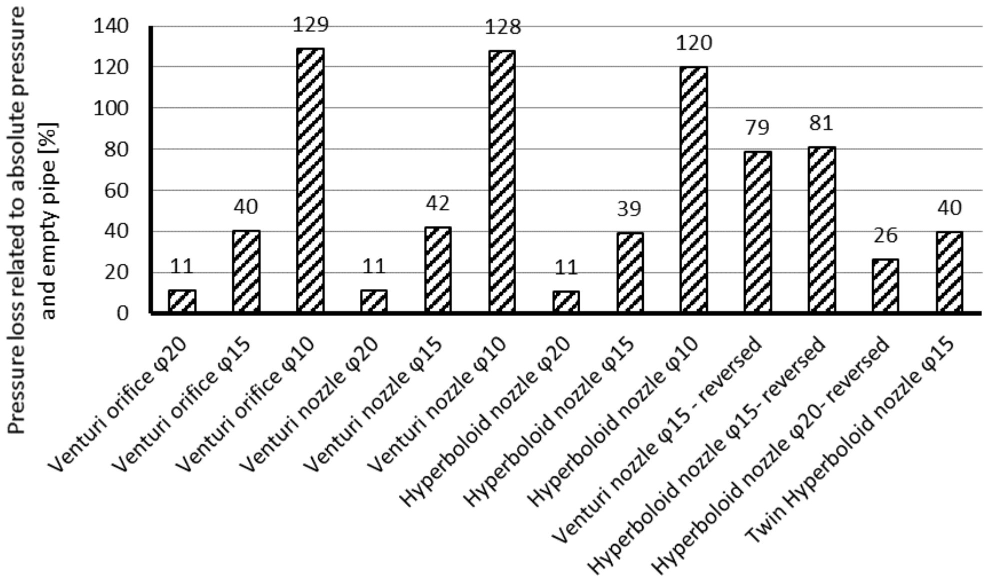

Fig. 6. *Relative pressure loss β on damping element in comparison with the pressure loss on an empty straight pipe in [%]*

6. Conclusions

In the paper comparison between parameters obtained in the experimental investigations and numerical simulations is analysed. The comparison shows that there is a possibility to exact some features from both, experimental and numerical results, which correspond to each other. It has been shown that the simulation of steady flow through the nozzle is a basis for compressor power increase determination. Obviously the CFD results are not directly the same as experimental determined parameters, because using CFD method only a short part of pipe with nozzle is simulated, while in the experimental investigation there is a lot of elements that have an influence on the number of parameters. In conclusion, a possibility of a good shape optimization using CFD is possible, but the interesting parameters have to be chosen reasonably and managed at a high level.

Symbols

Δp	pressure drop	(Pa)
f_D	dimensionless Darcy's coefficient	(1)
L	pipe length	(m)
D	pipe internal diameter	(m)
V	average flow velocity	(m/s)
εr	relative roughness of the wall	(1)
Re	Reynolds number	(1)
e	roughness height	(m)
$\dot{m}$	mass flow rate	(kg/s)
v_1	specific volume before the nozzle	(m^3/kg)
α	flow number	(1)
ε	expansion number	(1)
A	area of the nozzle cross-section	(m^2)
β	power loss coefficient	(kg/Ws)

References

1. He S., Jackson J., 2009. An experimental study of pulsating turbulent flow in a pipe. European Journal of Mechanics B/Fluids 28, pp.309-320.
2. Dowling, J. & Peat, K., 2003. An algoritm for the efficient acoustic analysis of silencers of any general geometry. Applied acoustics 65, pp. 211-227.
3. Lokitsangtong, M., Murakami, S. & Iijima, T., 2006. Acoustical characteristics of single-resonator-type silencers. Proceedings of the School of Engineering, Tokai University, Series E 31, pp. 39-44.
4. Hermann, J. et al., 2011. Frequency-dependent damping model for the hydroacoustic finite element analysis of fluid-filled pipes with diameter changes. Mechanical systems and signal processing 25, pp. 981-990.
5. Shademan, Y., Barron, M. & Balachandar, R., 2012. CFD Study of effects of geometry variations on flow in a nozzle. Engineering Applications of Computational Fluid Mechanics Vol.6, No. 3, pp. 412-425..
6. Lee, S.-H. & Ih, J.-G., 2008. Effect of non-uniform perforation in the long concentric resonator on transmission loss and back pressure. Journal of sound and vibration, pp. 280-296.
7. EngineeringToolbox, 2014. EngineeringToolbox. [Online]
Available at: http://www.engineeringtoolbox.com/darcy-weisbach-equation-d_646.html
[Accessed 07 04 2015].