Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 2, 2014, vol. LX
article No. 1984

Slawomir Blasiak*, Jakub E. Takosoglu, Pawel A. Laski*****

FLOW RATE ANALYSIS OF 3/2 DIRECTIONAL PNEUMATIC VALVE BY MEANS OF ANSYS CFX SOFTWARE

PRŮTOKOVÁ ANALÝZA 3/2 SMĚROVÉHO PNEUMATICKÉHO VENTILU, POMOCÍ SOFTWARU ANSYS CFX

Abstract

The main purpose of this paper was to develop a selection method of the pneumatic connectors for directional 3/2 valve. The method was established to minimize resistance and loss of pressure in the valve with mounted pneumatic connections for the selected pipe diameters. Directional valve was modeled in 3D CAD software SolidWorks while 3D models of the air connections have been downloaded from the website of one of the leading suppliers of pneumatic. Based on developed solid model the simulation of compressed air flow in the software for computational fluid dynamics Ansys CFX was conducted. The studies using CFD methods helped to determine which air connections best meet the assumptions. Performed numerical tests enable proper selection of items to the newly designed pneumatic systems for a particular group of valves. As a consequence, this translates into a reduction in energy consumption and improve the efficiency of the entire pneumatic complex system.

Abstrakt

Článek popisuje metodu pro vyhodnocení tlakových ztrát u 3/2 směrového pneumatického ventilu s pneumatickými spojkami pro vybrané průměry rozvodů tlakového vzduchu. 3D model ventilu byl vytvořen v CAD systému SolidWorks a modely pneumatických spojek byly staženy z webu významného dodavatele pneumatických komponent. Na těchto 3D modelech pak byla provedena průtoková simulace stlačeného vzduchu s využitím simulačního systému dynamiky tekutin Ansys CFX. Využití CFD metod umožňuje stanovit, které pneumatické zapojení ventilu nejlépe splňuje stanovené předpoklady. Provedené numerické výpočty umožňují volbu vhodných prvků při návrhu nové pneumatické ventilové sestavy. Dále pak vedou ke snížení spotřeby energie a zvýšení efektivity celého pneumatického systému.

Keywords

Control valve, turbulent flow, CFD simulation, numerical analysis.

* Ph.D. ing., Department of Mechanical Technology and Metrology, Faculty of Mechatronics and Mechanical Engineering, Kielce University of Technology, Aleja Tysiąclecia Państwa Polskiego 7, 25- 314 Kielce, tel. (+48) 413424756, sblasiak@tu.kielce.pl

** Ph.D., Department of Mechatronics Devices, Faculty of Mechatronics and Mechanical Engineering, Kielce University of Technology, Aleja Tysiąclecia Państwa Polskiego 7, 25- 314 Kielce, tel. (+48) 413424502, qba@tu.kielce.pl

*** Ph.D. ing., Department of Automation and Robotics, Faculty of Mechatronics and Mechanical Engineering, Kielce University of Technology, Aleja Tysiąclecia Państwa Polskiego 7, 25- 314, Kielce, tel. (+48) 413424489, pawell@tu.kielce.pl

1 INTRODUCTION

Due to the wide range of pneumatic systems applications to the topic in the proper selection of components, especially the elements of these systems, for example the air connections. The paper presents a numerical method to study the flow of compressed air through a directional 3/2 valve [1,2,3,4] with selected types of pneumatic connections.

Directional control valves [5,6,9,11] are pneumatic components whose task is to change the direction of the compressed air to individual routes of system and closing or opening the flow of air to the various circuits. Designing of pneumatic systems, in which are used a various pneumatic components is a task that requires a lot of experience from the constructor, and a vast knowledge of the entire process taking place on the production line, as well as the nature and the flow path of the working medium for each system routes. The use of mismatched connection and reduction elements or cables with wrong diameter may adversely affect the performance of the entire system.

Correct selection of the pneumatic components to the specific conditions of their work in the system is possible when characteristic parameters of these components under given conditions are known. Pneumatics calculations are based on approximate methods, and therefore getting the full compliance of established parameters with the actual parameters resulting from the nature of their work is extremely complex. For more detailed information about utility properties of designed components there is necessary to conduct laboratory tests using the finished product. The main aim of laboratory tests are: detailed analysis of the impact of various factors on the main parameters of the item and checking the correctness of the calculation assumptions, and to prepare the relevant characteristics. But not always it is possible to test the specific item in the laboratory. In such cases are often used simulation studies using specialized software. The use of flow simulation software with analysis and determination of flow characteristics significantly reduces the time to develop a project of the pneumatic system. The use of simulation studies allows for a thorough examination of the flow phenomenon of compressed air through the directional control valve with connectors. Comparing the calculated results with the experimental results (laboratory) allows full verification of the obtained data.

2 SOLID MODEL OF THE VALVE

The principle of operation of the valve-type 3/2 is presented in Fig. 1. In the initial position of the pilot piston (Fig. 1a), fluid flows between ways 1-2, while way 3 is cut-off. After the plunger is moved to the opposite extreme position, the medium flow followed by 2-3 at the cut-off road 1.

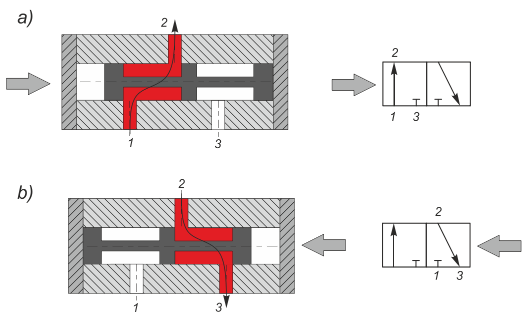

Fig. 1. Diagram of the 3/2 valve

The solid model of the directional valve with pneumatic connections to the selected conductor diameters is presented in Fig. 2.

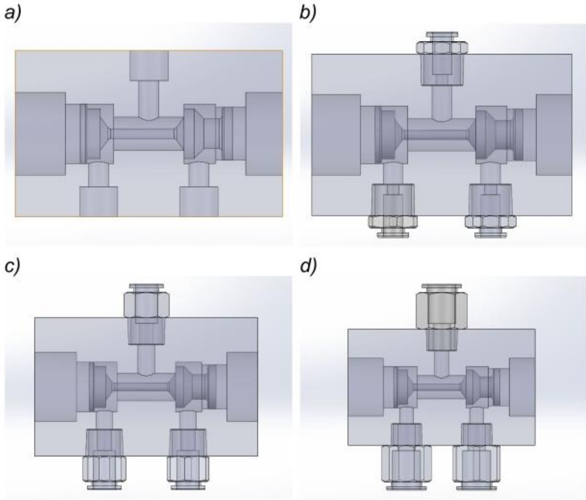

Fig. 2. Solid model of the 3/2 valve a) without pneumatic connections, b) with pneumatic connections ϕ 6 mm, c) ϕ 8 mm, d) ϕ 12 mm

Directional valves (3/2 type) are the most frequently used group of pneumatic valves in the propulsion systems and controls.

3 FLUID FLOW MATHEMATICAL MODEL

The calculation process included the law of motion. Developed the mathematical model of the control valve in the form of equations describing the physics of compressed air flow through the channels of the valve. The flow analysis, conducted with a computational fluid dynamics (CFD) program, involved transforming differential transport equations to obtain detailed information about the phenomena occurring in pneumatic control devices.

The turbulent flow of a viscous fluid is described with Reynolds equations (2)-(4), which, together with continuity equation (1) form a complete system of relationships able to determine the pressure and the flow rate area.

$$\frac{\partial \rho}{\partial t} + div\left(\rho \mathbf{U}\right) = 0 \tag{1}$$

Reynolds equations.

$$\frac{\partial\left(\rho U\right)}{\partial t} + div\left(\rho U \mathbf{U}\right) = -\frac{\partial P}{\partial x} + div\left(\mu\, grad\, U\right) + \left[-\frac{\partial\left(\rho \overline{u'^2}\right)}{\partial x} - \frac{\partial\left(\rho \overline{u'v'}\right)}{\partial y} - \frac{\partial\left(\rho \overline{u'w'}\right)}{\partial z}\right] \tag{2}$$

$$\frac{\partial\left(\rho V\right)}{\partial t} + div\left(\rho V \mathbf{U}\right) = -\frac{\partial P}{\partial y} + div\left(\mu\, grad\, V\right) + \left[-\frac{\partial\left(\rho \overline{u'v'}\right)}{\partial x} - \frac{\partial\left(\rho \overline{v'^2}\right)}{\partial y} - \frac{\partial\left(\rho \overline{v'w'}\right)}{\partial z}\right] \tag{3}$$

$$\frac{\partial\left(\rho W\right)}{\partial t} + div\left(\rho W \mathbf{U}\right) = -\frac{\partial P}{\partial z} + div\left(\mu\, grad\, W\right) + \left[-\frac{\partial\left(\rho \overline{u'w'}\right)}{\partial x} - \frac{\partial\left(\rho \overline{v'w'}\right)}{\partial y} - \frac{\partial\left(\rho \overline{w'^2}\right)}{\partial z}\right] \tag{4}$$

The *k-ε* model has become one of the most popular and definitely most commonly used models of turbulent flow. The two parameters (*k-ε*) require two additional transport equations, which can be written as:

$$\frac{\partial(\rho k)}{\partial t} + div(\rho k \mathbf{U}) = div\left(\frac{\mu_t}{\sigma_t}\, grad\, k\right) + \mu_t \phi - \rho\varepsilon \tag{5}$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + div(\rho \varepsilon \mathbf{U}) = div\left(\frac{\mu_t}{\sigma_t}\, grad\, \varepsilon\right) + C_1\, \mu_t\, \frac{\varepsilon}{k} \phi - C_2\, \rho \frac{\varepsilon^2}{k} \tag{6}$$

The parameter k denotes the turbulence kinetic energy, and ε is the turbulence kinetic energy dissipation rate. The above differential partial equations are implemented in the computational module of the Ansys CFX program [7,8,10]. To effectively solve the system of equations describing the turbulent flow of the fluid, it is necessary to use boundary conditions that guarantee the uniqueness of the solution and affect the computational process in the area analyzed.

4 NUMERICAL RESULTS

Comparative analysis of numerical results for the control valve with the selected pneumatic connectors are shown in Fig. 3.

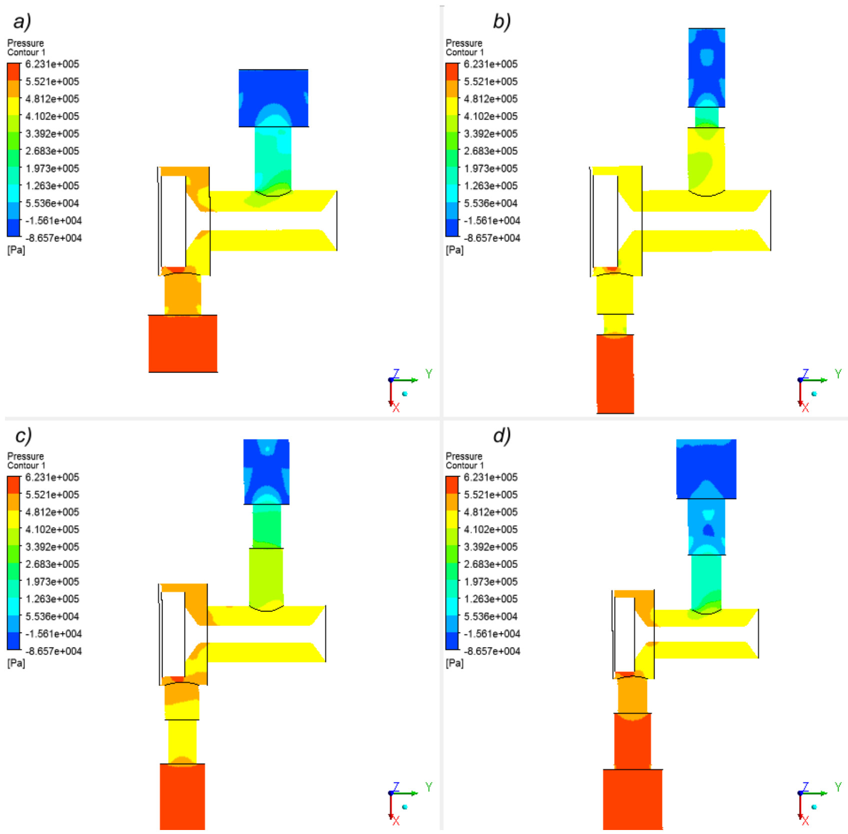

Fig. 1. Distributions of pressure a) without pneumatic connections, b) with pneumatic connections ϕ 6 mm, c) ϕ 8 mm, d) ϕ 12 mm

It may be noted that the pressure distribution in the symmetry axis of the valve in two extreme cases: without connector (Fig. 3a) and with pneumatic connector ϕ 12 mm (Fig. 3d) are very similar. It means that air connector for cable diameter of 12 mm was properly chosen and in the presented case there is no throttling of the air stream. In all studied cases the expansion of air is already in the inlet of valve chamber and reaches a value equal to the ambient pressure at the mouth to the atmosphere.

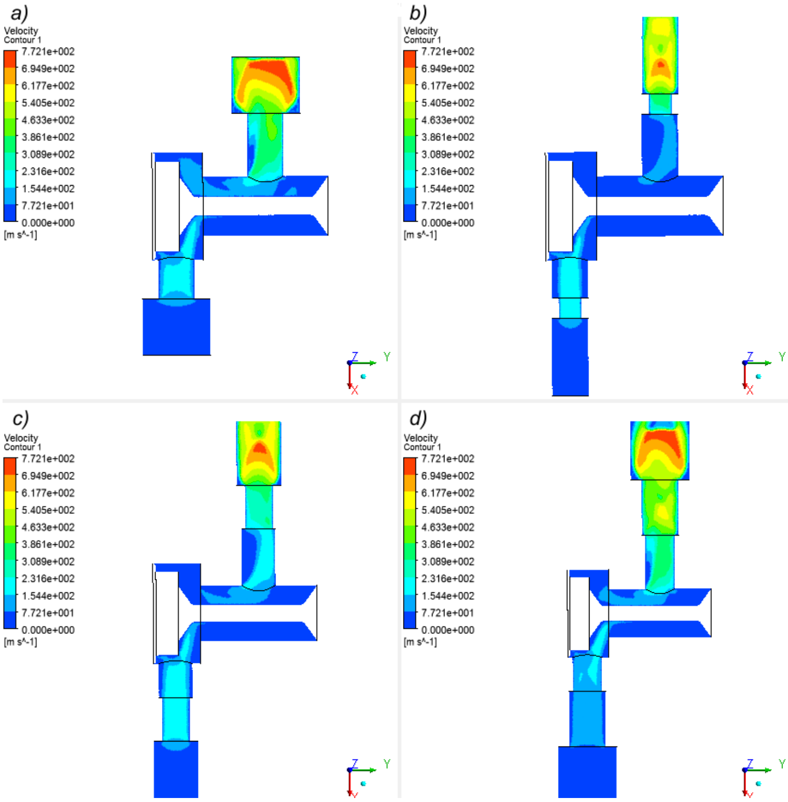

Fig. 2. Distributions of the flow rate, a) without pneumatic connections,

b) with pneumatic connections ϕ 6 mm, c) ϕ 8 mm, d) ϕ 12 mm

In the inlet and outlet channel can be clearly see that near the walls the air velocity is practically zero, and the highest velocity is achieved in the central part. The Fig. 4 also shows that the air velocity in the transition from the chamber to the outlet duct increases from close to zero to reach more than 720 $[m/s]$.

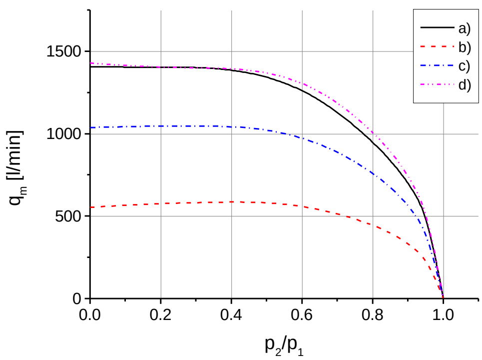

Fig. 3 The volumetric flow rate a) without pneumatic connections,

b) with pneumatic connections ϕ 6 mm, c) ϕ 8 mm, d) ϕ 12 mm

Flow characteristics are shown in Fig. 5 were obtained based on model tests. The studies showed the differences in the value of the volumetric flow rate for selected diameters of pneumatic connections of the directional valve.

5 CONCLUSIONS

Applied methods of computational fluid dynamics led to the development of numerical method for testing air flow through the valves applied to the pneumatic system and to determine the basic characteristics of the flow already in the initial design phase. Presented in the paper a method of selecting pneumatic components guarantees obtaining the desired parameters of executive components during the design of pneumatic systems.

REFERENCES

[1] J. Takosoglu & R. Dindorf, P. Laski. Rapid prototyping of fuzzy controller pneumatic servo-system. *Int J Adv Manuf Technol* 40 (2009) 349-361.

[2] J.E. Takosoglu & P.A. Laski & S. Blasiak. A fuzzy logic controller for the positioning control of an electro-pneumatic servo-drive. *Proceedings of the Institution of Mechanical Engineers. Part I: Journal of Systems and Control Engineering* 226 (2012) 1335–1343.

[3] M. Zhou & W. Gao & Z. Yang. Stiffness analysis of electromechanical transducer for nozzle flapper piezoelectric servo valve. *Przegląd Elektrotechniczny* 88 (2012) 196–199.

[4] H. Ohuchi & H. Akasaka & T. Osada. Pneumatic pulse flow conrtol valve driven by successive impulsive force of a piezoelectric multilayer actuator. *FLUCOME 2000, 6th Triennial International Symposium on Fluid Control Meausrement and Visualization*, Université de Sherbrooke, (2000) 54–57.

[5] A. Valdiero & C. Wang & L. Bortolaia & P. Andrighetto & Y. Jiang. Analysis of pneumatic directional proportional valve with CFX mesh motion technique. *COBEM 2007, 19th International Congress of Mechanical Engineering ABCM Symposium Series in Mechatronics.* 5th to 8th November, 2007, Brasilia - DF, Brazil ; (2007) 510–518.

[6] N. Okungbowa, CFD Analysis of Steady State Flow Reaction Forces in a Rim Spool Valve. Dysertacja, 2006.

[7] G. Palau-Salvador, P. González-Altozano, J. Arviza-Valverde. Three-Dimensional Modeling and Geometrical Influence on the Hydraulic Performance of a Control Valve. *J. Fluids Eng.* 130 (2008) 11102.

[8] W.M. Dempster, W. Elmayyah (Eds.). A computational fluid dynamics evaluation of a pneumatic safety relief valve.

[9] I. Krzysztofik, Z. Koruba. Mathematical Model of Movement of the Observation and Tracking Head of an Unmanned Aerial Vehicle Performing Ground Target Search and Tracking. *Journal of Applied Mathematics*. vol. 2014, Article ID 934250, 11 pages, 2014. doi:10.1155/2014/934250

[10] S. Adamczak, J. Bochnia, C. Kundera. Stress and strain measurements in static tensile tests *Metrology and Measurement Systems*, No. 3, Vol. XIX, pp. 531-540, 2012

[11] L. Licev, M. Babiuch, R. Farana. Software Analysis of Mining Images for Objects Detection *ACTA MONTANISTICA SLOVACA* 18(1): pp. 59–66, 2013