20th International Conference
ENGINEERING MECHANICS 2014
Svratka, Czech Republic, 12 – 15 May 2014

OPTIMIZING THE FLOW RATE IN A PNEUMATIC DIRECTIONAL CONTROL VALVE

S. Blasiak[*], J. E. Takosoglu[**], P. A. Laski[***]

Abstract: *The aim of the study was to optimize the flow of compressed air through a pneumatic directional control valve by optimizing the valve design. The 3D model of the valve was constructed in a CAD SolidWorks program. The assumptions concerning the performance included low friction, a small pressure loss, and accordingly high flow parameters, i.e. sonic conductance and critical pressure ratio. Ansys CFX, a computational fluid dynamics program, was used to simulate and optimize the flow of compressed air through the valve, and examine the basic flow characteristics. By optimizing the fluid flow at an early design stage, it will be possible to produce directional control valves with improved flow parameters.*

Keywords: Poppet valve, Control valve, Turbulent flow, CFD simulation, Numerical analysis.

1. Introduction

This work discusses a method for optimizing the flow of compressed air through a directional control valve, using the example of a poppet valve. Pneumatic directional control valves allow compressed air to flow into different paths by opening and closing ports to different pneumatic circuits. Developing a new valve or modifying an existing one is a demanding task; it requires great experience and knowledge on the character of the fluid flowing along the valve ways (Takosoglu et al., 2009). Modifications to the design need to be verified via testing. It is also vital to determine the flow parameters, i.e. the critical pressure ratio and sonic conductance, in accordance with the appropriate standard. Design modifications made to individual valves prior to their production would be extremely costly and time-consuming. If suitable software is employed to analyze the fluid flow and determine the flow characteristics, the design modification time will be shortened and the costs reduced. Simulations can be used to rigorously analyze the flow of compressed air through a valve, with the findings being applicable to the entire group of directional control valves. To perform a full verification, the simulation results need to be compared with the experimental data.

2. Solid Model of the Valve

The principle of operation of the 3/2 valve is illustrated in Fig. 1. In the initial position of the pilot piston (Fig. 1a), the fluid flows between ways 1 and 2, while way 3 is cut-off. After the plunger is moved to the opposite position (Fig. 1b), the fluid passes between ways 2 and 3, with way 1 being cut-off.

The solid model of the poppet valve is presented in Fig. 2. 3/2 directional control valves are the most common valves used in pneumatic drive and control systems.

3. Mathematical Model of the Flow

The flow analysis, conducted with a computational fluid dynamics (CFD) program, involved transforming differential transport equations to obtain detailed information about the phenomena

[*] Sławomir Błasiak, PhD.: Department of Mechanical Engineering and Metrology, Faculty of Mechatronics and Machine Design, Aleja Tysiaclecia Panstwa Polskiego 7; 25-314 Kielce; Poland, sblasiak@tu.kielce.pl

[**] Jakub E. Takosoglu, PhD.: Department of Mechatronic Devices, Faculty of Mechatronics and Machine Design, Aleja Tysiaclecia Panstwa Polskiego 7; 25-314 Kielce; Poland, qba@tu.kielce.pl

[***] Paweł A. Łaski, PhD.: Department of Automation and Robotics, Faculty of Mechatronics and Machine Design, Aleja Tysiaclecia Panstwa Polskiego 7; 25-314 Kielce; Poland, pawell@tu.kielce.pl

occurring in pneumatic control devices. The calculation process uses the laws of motion; the mathematical model of the directional control valve is developed in the form of equations describing the flow physics (Takosoglu et al., 2012). 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.

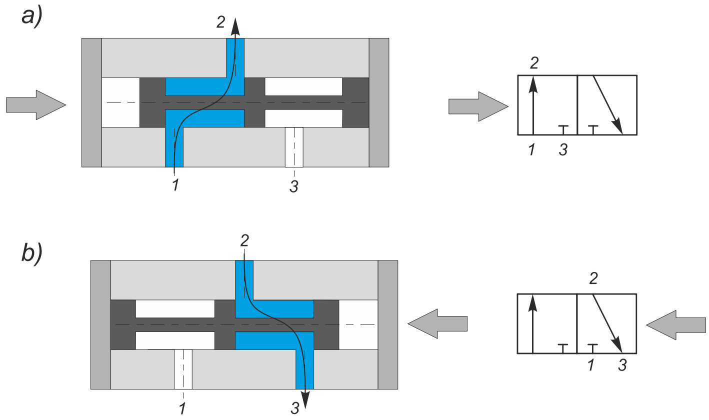

Fig. 1: Diagram of the 3/2 valve.

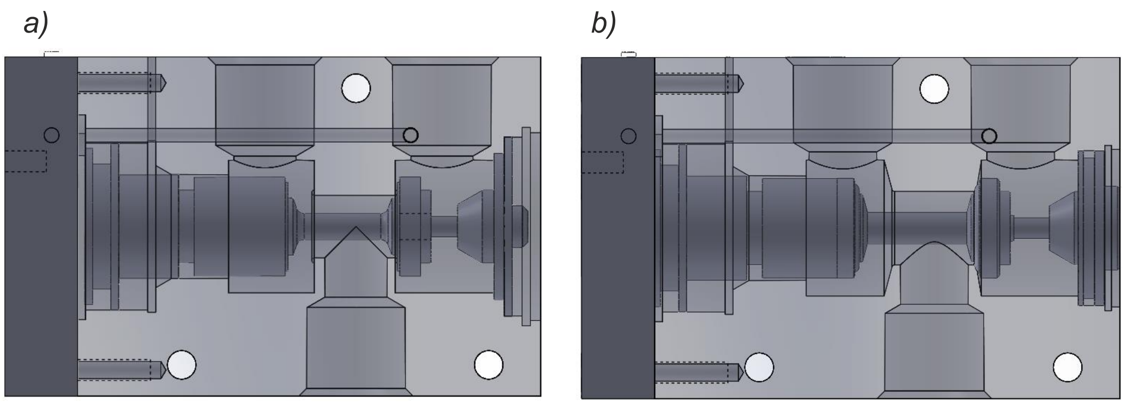

Fig. 2: Solid model of the 3/2 valve a) before modification, b) after modification.

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

The Reynolds equations are:

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

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

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

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. 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 Analysis

This section discusses the results of a numerical analysis for a poppet valve to determine its performance before and after modifications to the design.

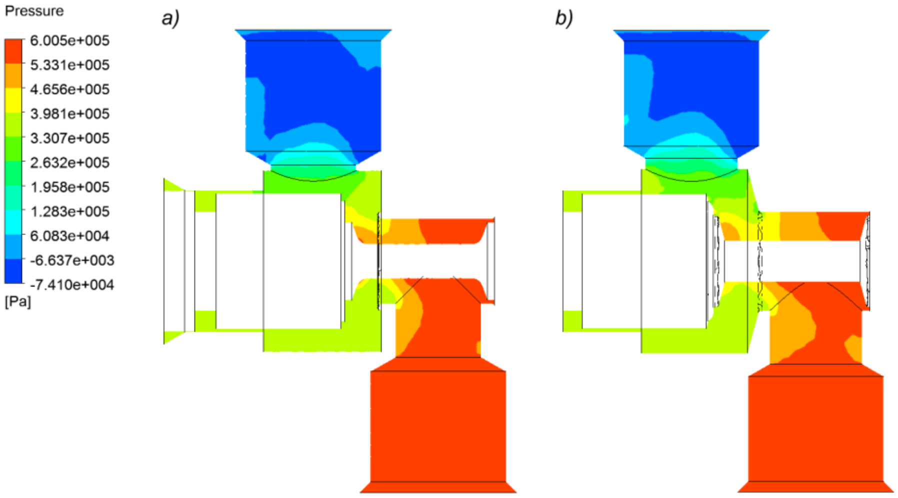

Fig. 3: Distributions of pressure.

As can be seen, the pressure distributions along the axis of symmetry are different for the two cases, i.e. before and after modifications (Figs. 1a and 1b, respectively). After the modification, the compression of the air takes place as early as in the inlet chamber; air reaches a pressure of approximately $4\cdot10^5$ [Pa]. The expansion occurs in the outlet port at the exit; the pressure of the air is nearly equal to the pressure of the environment.

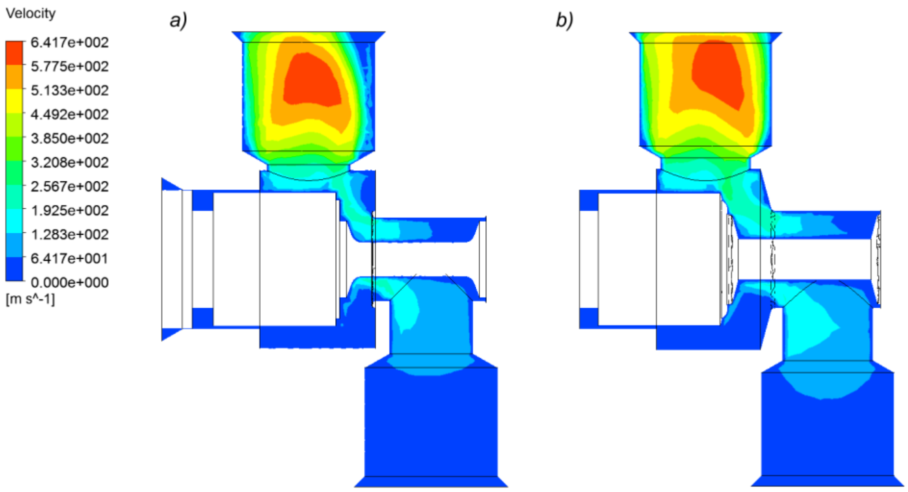

Fig. 4: Distributions of the flow rate.

It can be seen that, in the inlet and outlet ports, the rate of the air flow close to the walls is practically equal to zero and that the rate is the highest in the central part. As shown in Fig. 4, the air flow rate increases from a value close to zero to more than 640 $[m/s]$ when the air passes from the second chamber to the outlet port.

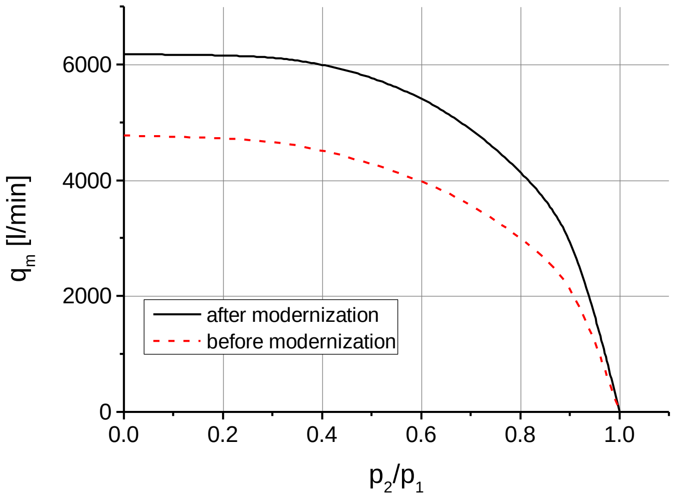

Fig. 5: Volumetric flow rate.

The curves in Fig. 5, based on the model analysis, illustrate the differences in the volumetric flow rate for the poppet valve before and after modifications to its design.

5. Conclusion

A numerical approach to fluid mechanics was used to optimize the flow of compressed air through the valve and determine the basic flow characteristics at the early design stage. This optimization method will enable us to produce pneumatic valves with improved operating parameters and functionality.

Acknowledgement

The numerical analysis was performed using software purchased for the LABIN Project (POPW.01.03.00-26-016/09) co-funded by the EU.

References

Takosoglu, J. E., Dindorf, R. F. & Laski, P. A. (2009) Rapid prototyping of fuzzy controller pneumatic servo-system. International Journal of Advanced Manufacturing Technology Vol. 40, No. (3-4), pp. 349-361.

Takosoglu, J. E., Laski, P. A. & Blasiak, S. (2012) A fuzzy 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, Vol. 226, No. 10, pp. 1335-1343.