CFD Simulation of Pneumatic Valve for Energy Efficiency Studies

Evelyn Melissa Serino Monteiro[1], Luciana Lima Monteiro[2], José Ângelo Peixoto da Costa[3]

[1]Department of Mechanical Engineering, Instituto Federal de Educação, Ciência e Tecnologia de Pernambuco, Brazil
Email: *emsm1[at]discente.ifpe.edu.br*

[2]Department of Mechanical Engineering, Instituto Federal de Educação, Ciência e Tecnologia de Pernambuco, Brazil
Email: *lucianamonteiro[at]recife.ifpe.edu.br*

[3]Department of Mechanical Engineering, Instituto Federal de Educação, Ciência e Tecnologia de Pernambuco, Brazil
Email: *angelocosta[at]recife.ifpe.edu.br*

Abstract: ***Pneumatics is a technology that uses air to perform work through actuators, which perform a specific task. These systems have been increasingly and widely used in the industrial environment for process automation, as they are economically viable, robust, safe and simple. Despite the various benefits, pneumatic systems have reduced efficiency, when it comes to energy consumption, therefore, increasing the costs, because of this, this work is aimed to reduce energy losses in pneumatic circuits. The methodology used was analytical, through computational simulation of fluid dynamics, following the steps of modeling, mesh and configuration, a conventional 5/2-way directional valve geometry was used as a reference for modeling and the simulation of satisfactory results of 25% in the increased mass flow.***

Keywords: CFD, efficiency, pneumatic, valve

1. Introduction

The technological advance, characteristic of the modern world, increases the need to optimize processes that previously required human interference. Pneumatic systems are known for their simplicity of design, economic viability, the robustness of components, and the various applications that can be inserted, according to [2], these characteristics make these systems widely used in the automation of machines present in the industrial environment. Among its different uses, we can mention all the packaging technology, machine tools, packaging of volumes and transport of parts.

Pneumatic technology uses compressed air as an energy source, from capture in the atmosphere, through storage and distribution in the circuit, to the generation of mechanical work, with movements that can be linear or rotary in nature. The generation group is composed of compressors, for compressed air treatment, energy conservation units with filters, lubricants and pressure regulators are used, and for storage, reservoirs are used. In addition, there are the control groups, composed of valves that direct, block, or regulate the airflow and actuators, among which are cylinders, engines, and suction cups.

Despite the advantages, pneumatics is characterized by the waste of energy within the mechanism, causing it not to be fully utilized. This occurs for several reasons, among them are air leaks due to wear, load losses in the piping, poor sizing of systems, and the lack of reuse of compressed exhaust air. Therefore, increasing energy efficiency is necessary, since reducing production costs is a determining factor for the survival of companies in a competitive market [3]. At the same time that academic productions in this area evolve, companies manufacturing pneumatic components are also launching solutions aimed at energy efficiency, due to the demands of Industry 4.0 [6].

As put by [5], there are different strategies for improving energy efficiency, one of which focuses on reducing consumption by reducing dead volume and optimizing dimensional parameters, [7] states that the recent development and popularization of CFD makes it easier to achieve an understanding of the flow solution through complex valve channels. Taking this as a basis, the present work aimed to improve the energy performance of a solenoid-driven 5/2-way directional valve, which was verified and had the structural parameters optimized, through a tool called Computational Fluid Dynamics Simulation (CFD), which enabled the analysis of the phenomenon of transport and flow of air through the interior of the component.

2. The Model of the Valve

The first step of a CFD simulation consists of the geometric modeling of the component to which the physics will be applied. In the study of the phenomenon, a 5/2-way valve was used, the geometric construction was based on the FESTO® valve MEH-3-0.9 DC24W, taken as a reference for the design of the porticos, spool, internal cavities and external part. The following characteristics are present: five-way, two positions, activation by solenoid and spring return.

The modeling was performed in SOLIDWORKS® software adopting the measurements of the reference valve and, subsequently, an assembly was made to fit the sealing bushings and spool in the internal structure, due to the dynamics of the process, the drives were disregarded since the relevance to the study is not in the movement of the spool itself, but in the behavior of the air passing through the porticos of the valve, in the drive position 1-4 for mass flow analysis Fig. 1.

In the spool actuated position, pressure line 1 is where the air passes reaching port 4 for use in the first chamber of a cylinder, while the air that is in the second chamber passes through port 2 and returns to the atmosphere through the exhaust 5. While in the deactivated position, the air returns from the first chamber straight to exhaust 3 and the air passing to port 2 keeps the actuator back. The flow rate during the passage of air from the inlet to the use line can indicate a good or bad performance by the valve in the circuit regarding energy efficiency.

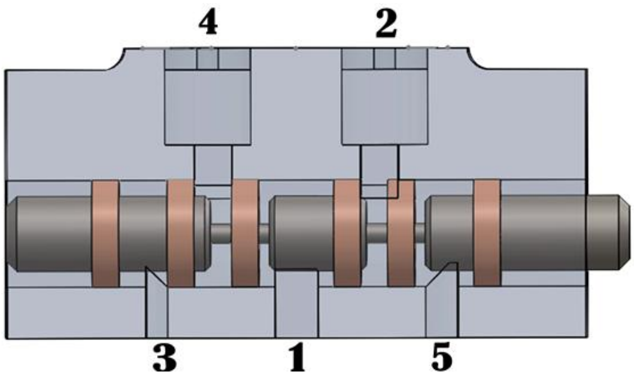

Figure 1: Valve modeled and assembled with bushings and spool

The geometry was extracted for simplification in the computational analysis Fig. 2, taking into account that the focus is on the air mass flow that the valve is able to transport according to its sizing, the study region was delimited. The preparation was done in the Design Modeler space of ANSYS® and only fluid domain regions were considered. The mesh generation Fig. 3 is part of the preprocessing and is done as a way to map surfaces for solving the equations using the finite element method, the mesh was generated with quadratic elements and limited to 454388 elements and 639755 nodes.

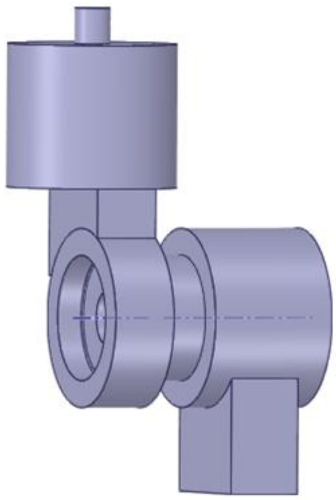

Figure 2: Extracted volume

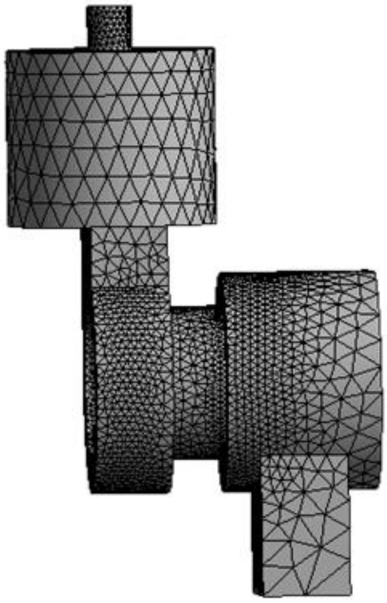

Figure 3: Computational mesh

2.1 Changes in internal geometry

With the proposal of increasing the energy efficiency of the valve in question, some modifications were made in the airflow path, in order to obtain an increase in the mass flow rate and consequently in the amount of air transported to the cylinder in the same period.
Internal structural changes were also made in the SOLIDWORKS® software to perform a new simulation, the following changes were made throughout the process and together at the end:

1) The volume reduction in air recirculation zones, caused by the spaces between the sealing bushings taking the spool as a reference.
2) Rounding on the inner edges of the spool.
3) Increase in diameter at the air outlet.

3. Mathematical Model

The mathematical models are defined in the component's processing environment, also called Setup, where all the physics is defined. The material condition was limited to Ideal Gas at a temperature of 25°C, which is part of CNTP (normal conditions of temperature and pressure), and the heat transfer model was isothermal, this choice of air in the domain activates the effects of compressibility since it uses the Ideal Gas Law or Clapeyron's Equation. At the air inlet, the pressure was set as 4 bar and at the outlet as 3.75 bar, an experimentally measured pressure drop, in the laboratory.

The Navier Stokes Equations were used to study the fluid flow inside the valve, including the Continuity Equation (1) which uses the relationship between the flow velocity and the available area, the Energy Equation (2) describing the behavior of the fluid inside the tubes, and the Three-Dimensional Momentum Conservation Equation:

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(pw)}{\partial z} = 0 \tag{1}$$

$$\rho c_p \left(u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = \beta T \left(u \frac{\partial P}{\partial x} + v \frac{\partial P}{\partial y} + w \frac{\partial P}{\partial z} \right) + \underset{\nabla}{\rightarrow} \cdot \left(k \underset{\nabla}{\rightarrow} T \right) + \Phi \tag{2}$$

These equations are used in conjunction with the k-ε turbulence model in which two transport equations are used to describe the turbulent flow, focusing on the processes that affect kinetic energy. This turbulence model is one of the most common ones used today for turbulent flows. 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 (BLASIAK et al., 2014). The boundary conditions for fluid inlet and outlet relative to the domain were defined, below are the two transport equations used Equation. (3) and (4):

$$\frac{\partial(\rho k)}{\partial x} + div(\rho k \mathbf{U}) = div\left(\frac{\mu_t}{\sigma_t} grad\ k\right) + \mu_t \Phi - \rho \varepsilon \tag{3}$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + div(\rho \varepsilon \mathbf{U}) = div\left(\frac{\mu_t}{\sigma_t} grad\ \varepsilon\right) + C_1 \mu_t \frac{\varepsilon}{k} \Phi - C_2 \rho \frac{\varepsilon^2}{k} \tag{4}$$

4. Results

The numerical results provided an overall parameter about the airflow inside the valve. Comparative analysis can be done on both cases that were simulated, where the values for velocity, pressure behavior and mass flow rate are presented, the applied contour is located near the central region of the valve. There were limitations on the mesh and interactions for convergence in the CFX Solver, the ANSYS license used was the student version, 200 interactions were defined. For ease of understanding, the reference valve will be treated as Case 1 and the modified valve as Case 2. The contours were created practically in the center of the valve, in the YZ plane, and vectors were used for velocity.

The velocity in Case 1 Fig. 4 presented subsonic values and it can be seen that the highest peak was reached at the final exit orifice with a value of 209 m/s, during the route the velocity reached the value of 0 m/s and there was air recirculation in some central regions, near the exit the air was more dispersed in its flow. The pressure remained within the parameters entered for pressure drop Fig. 5, this pressure difference was low and there was not much energy loss in the process.

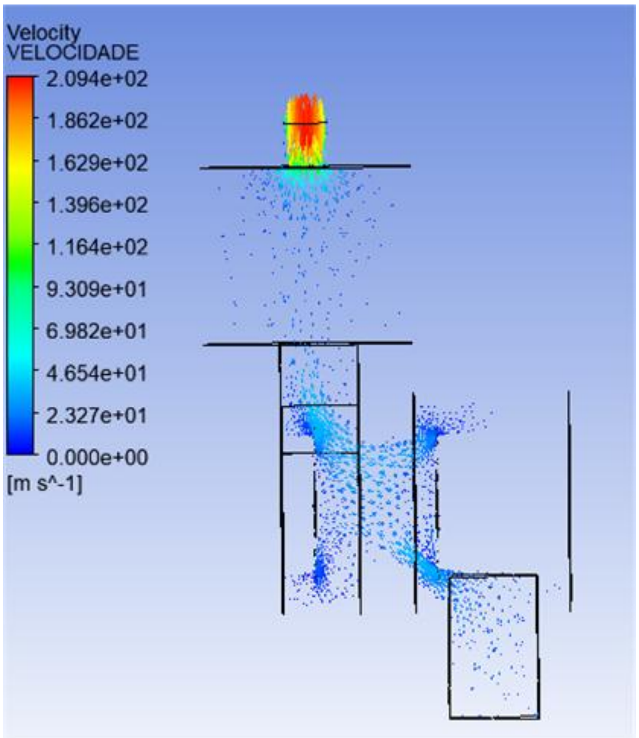

Figure 4: Velocity contour, case 1

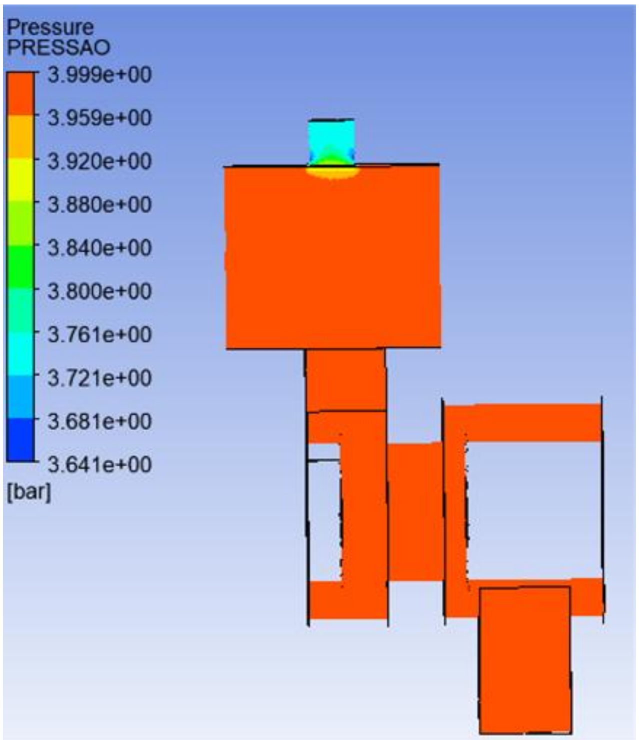

Figure 5: Pressure Contour, case 1

For Case 2, when the geometry was changed, an increase in airflow inside the valve was visible, the increase in the spool bevels and the decrease in volume allowed for part of the air passage to be unobstructed Fig. 6, the velocity had its value, in general, greatly decreased and the air exit presented the maximum velocity of 66 m/s. In both situations, the velocity did not change much during the route. The pressure drop was a little lower and was within the predefined parameter Fig. 7.

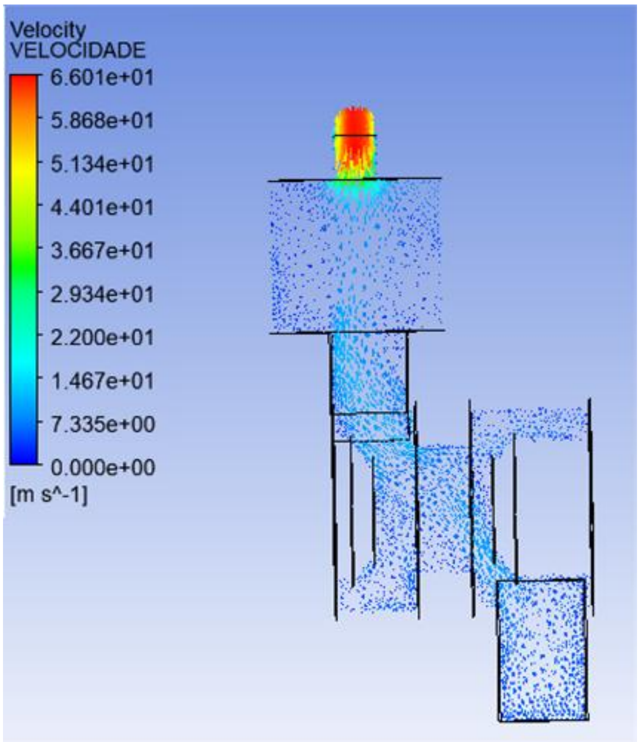

Figure 6: Velocity contour, case 2

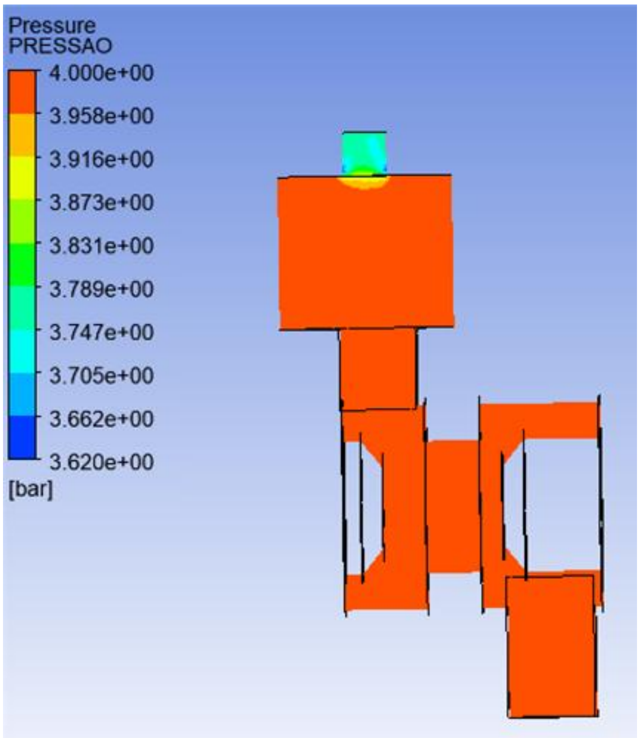

Figure 7: Pressure contour, case 2

The mass flow rate data is extremely relevant, since according to Whitehead et al. [1] and [4] changing aspects of the flow in the measurements made by it has an impact on the valve operation. The following mass flow rate values, predominant in the simulation results, were obtained, they are shown in the table 1 below:

Table 1: Table caption above the table

	Mass flow rate (e-06)
Case 1	4.227 kg/s
Case 2	5.280 kg/s

5. Conclusion

Computational fluid dynamics is a useful tool for analyzing pneumatic valves and making geometry modifications to improve efficiency because it gives a numerical overview in the early stages. Although the reduction in pressure drop was not significant, the results showed a 25% increase in mass flow along with improved airflow in the internal structure. Regarding the concreteness of the values, it is known that a more refined mesh can present values closer to the real one, besides that, an experiment with both cases can give a good notion.

6. Acknowledgements

Evelyn Melissa Serino Monteiro thanks the support of IFPE, Instituto Federal de Educação, Ciência e Tecnologia de Pernambuco in Brazil.

References

[1] Whitehead, Nicholas Paul; Arezki, Slaouti; Howard, Taylor. (2007). Optimisation of Flow Through a Pneumatic Control Valve using CFD Analysis and Experimental Validation. International Journal of Fluid Power, 8:3, DOI: 10.1080/14399776.2007.10781284, pp. 31-41.

[2] VIGOLO, V. Theoretical-Experimental Study to aid in the design of pneumatic actuation systems. (2018). Master's thesis (in Portuguese), UFSC, Florianópolis, Brazil, pp. 27.

[3] Endler, L, A Solution to Saving Compressed Air in Pneumatic Position Control Systems. (2014). Doctoral Thesis Postgraduate Program in Automation and Systems Engineering (in Portuguese), UFSC, Florianópolis, Brazil, pp. 135.

[4] Blasiak, Sławomir; Takosoglu, Jakub E.; Łaski, Paweł A. (2014), Transactions of the VŠB – Technical University of Ostrava, Mechanical Series, No. 2, vol. LX.

[5] HARRIS, P.; NOLAN, S.; O'DONNELL, G. E. (2014). Energy Optimization of Pneumatic Actuator Systems in Manufacturing. Journal of cleaner production, v. 72, pp. 35-45, ISSN 0959-6526.

[6] GREIGARN, K. (2016) Industry 4.0 toward Sustainability. Bangkok, Tailândia: EAU Heritage Journal.

[7] Jiang, Y., Valdiero, A. C., Andrighetto, P. L., Chong, W., & Bortolaia, L. A. (2008). Analysis of pneumatic directional proportional valve with CFX mesh motion technique. In ABCM Symposium Series in Mechatronics,vol. 3, pp. 510-518.