COMPUTATIONAL AND EXPERIMENTAL INVESTIGATION ON THE FLOW CHARACTERISTICS OF SMALL-SCALE PNEUMATIC SOLENOID VALVES

V. Szente
Ph.D. student, Department of Fluid Mechanics, Budapest University of Technology and Economics, H-1111. Budapest, Bertalan Lajos u. 4 – 6. Tel: (+36-1)-463-3187, e-mail: szente@simba.ara.bme.hu

J. Vad
Associate Professor, Department of Fluid Mechanics, Budapest University of Technology and Economics, H-1111. Budapest, Bertalan Lajos u. 4 – 6. Tel: (+36-1)-463-2464, e-mail: vad@simba.ara.bme.hu

SUMMARY
A simplified one-dimensional (1D) simulation model has been elaborated in AMESim environment, which is capable of analyzing and predicting the flow characteristics of small electropneumatic (EP) valves within a wide range of pressure ratios. The flow coefficient in the model has been identified on the basis of experiments and was compared with the Perry model. The flow coefficient values have been successfully reproduced by axisymmetric quasi-3D (Q3D) computations using the FLUENT code. A number of different geometries were analyzed to determine the correlation between the geometry and the flow coefficient.

INTRODUCTION
Electropneumatic (EP) valves are widely applied in several areas of industry. The knowledge on the dynamic flow characteristics of such valves is especially important in the cases when they are integrated in fast-response, controlled fluid power systems as control devices. A typical application of this kind is realization of control functions in intelligent EP braking systems of motor vehicles. The dynamic flow characteristics of EP valves influence the successful operation of the entire fluid power equipment.

As illustrated in SZENTE et al. [1], a simplified 1D simulation tool can be effectively used in design, research and development regarding controlled fluid power systems. The EP valve models integrated in such models must represent reliably the transmission characteristics of the EP valve, without time-consuming and practically unnecessary resolution of 3D effects.

This paper presents an advanced 1D fluid mechanical submodel prepared in AMESim environment, supplying reliable information on the flow transmission characteristics of the EP valve. The 1D model has been parameterized by experiments carried out on an EP valve of case study. The flow transmission characteristics identified experimentally were compared with literature data and have been successfully resolved by axisymmetric Q3D flow computation using the FLUENT code. After the validation of the computational tools, a number of different geometries were calculated and analyzed to serve as a knowledgebase for future developments.

NOMENCLATURE
\[ x \quad \text{valve body displacement [m]} \]
\[ q_m \quad \text{mass flow rate [kg/s]} \]
\[ p \quad \text{absolute pressure [bar]} \]
\[ T \quad \text{temperature [ºK]} \]
\[ C_q \quad \text{orifice flow coefficient [}/(m/s)Kº]\]
\[ C_m \quad \text{orifice mass flow parameter [-]} \]
\[ A \quad \text{orifice cross-section [m²]} \]

Greek letters
\[ \alpha \quad \text{orifice plate angle [º]} \]

Subscripts
\[ \text{up} \quad \text{upstream values} \]
\[ \text{down} \quad \text{downstream values} \]

EP VALVE AND TEST CASE
The valve under investigation is applied in fast-response pneumatic systems as control valve providing e.g. pressure signal for relay valves. Such miniature valves must provide rapid, pulsed fluid transmission between enclosures of relative pressures in the order of magnitudes of 10 bar and 0 bar within a time period in the order of magnitude of 0.01 s. It is of critical importance to elaborate a reliable fluid dynamical model for the valve to be applied in design of the fluid power hardware and its control.

Fig. 1 shows the simplified scheme of the valve (SZENTE and VAD [2]). The valve body position coordinate \( x \) is also indicated in the figure. The valve body is equipped with flexible seal and contact surfaces. In absence of solenoid excitation, the valve body is kept at its closed end-position by the return spring. The solenoid is energized by DC voltage. The frame and the jacket assist in development.
of a magnetic circuit. The resultant magnetic force displaces the valve body against the return spring. As a consequence, a flow cross-section develops through the orifice.

![Fig. 1. Scheme of the EP valve](image1)

In order to investigate experimentally the flow transmission characteristics of the EP valve under present investigation, a test facility has been set up. The scheme of the test system is shown in Fig. 2a. It comprises two pneumatic chambers and the valve described above. At the beginning of each case, the large chamber has been pressurized while the small one has been set to atmospheric pressure. By setting the pressure in the large chamber, various pressure ratios (downstream to upstream absolute pressure) have been adjusted to initialize the experiment. Then the valve has been opened and the small chamber has been loaded with the pressurized air from the large chamber. The pressure and temperature in the large chamber stayed practically constant during the loading process. Pressure transducers have been incorporated to both chambers. No temperature measurement was applied on the small chamber.

![Fig. 2a. Scheme of the test system](image2)

1D VALVE MODEL AND EXPERIMENTAL PARAMETER IDENTIFICATION

The through-flow area of the EP valve has been modeled as a pneumatic orifice with a geometrical cross-section controlled by the valve body position. The flow through the valve has been approached as series of short-term stationary sub-processes. The mass flow rate $q_m$ through the orifice is a function of upstream absolute pressure $p_{up}$, upstream temperature $T_{up}$, orifice cross-section $A$, flow coefficient $C_q$ and mass flow parameter $C_m$ ([3], BIDEAUX and SCAVARDA [4]):

$$q_m = A \cdot C_q \cdot C_m \cdot \frac{p_{up}}{T_{up}}$$

The experimental set-up has been represented in AMESim environment [5], making possible 1D dynamic flow simulation (Fig. 2b). No heat transfer is considered in the simulation model of the small chamber. The quantities in Eq. (1), except for $C_q$, have been set according to the geometrical and measurement circumstances. Considering that the effect of heat transfer is negligible only at the initial time instance of the loading process, the initial rate of pressure increase in the small chamber was considered as a basis of experiment-based calculation of mass flow rate. The value of $C_q$ has been selected in such a way that the initial rate of simulated pressure increase in the small chamber is equal to that experienced in the measurements, representing that the initial sonic mass flow rate (i.e. the left-hand side of Eq. (1)) is equal for the measured and simulated cases. Comparative sample data for measured and simulated pressures in the small chamber are shown in Fig. 3. Except for the initial period immediately after opening the EP valve (where the effect of heat transfer through the chamber wall is negligible), a minor departure can be observed between measured and simulated pressure values due to heat exchange between the small chamber and the environment.
Using the above described procedure, a pressure ratio-dependent $C_q$ parameter has been established experimentally for the 1D model. The experimental results of $C_q$ have compared to data according to values according to the Perry polynomial established for sharp-edged orifices (McCLOY and MARTIN [6]). The results are presented in Fig. 6 and will be discussed later.

**Q3D SIMULATION MODEL AND SIMULATION**

The experimentally determined $C_q$ values built in the 1D AMESim model were attempted to be reproduced by axisymmetric Q3D computations using Computational Fluid Dynamics (CFD) code FLUENT [7]. Because of axial symmetry, the 3D valve model has been transformed to Q3D axisymmetric domain. Fig. 4a shows the 3D layout of the valve, while Fig. 4b shows the 2D scheme that has been used in CFD simulation. Because of present limitations in the 3D simulation software available, the movement of the valve body has not been incorporated into the model.

The simulation software computed the mass flow rate for the initial state, the cross-section of the flow has been given. $p_{up}$, $p_{down}$ and $T_{up}$ have been given as boundary conditions. A computed Mach number contour distribution can be seen in Fig. 5 as example. Such diagrams can be used in the future for a detailed analysis of flow field within the valve. Considering that $C_m$ is constant through the sub-critical pressure ratio domain (sonic flow, BIDEAUX and SCAVARDA [4]), the value of $C_q$ has been deduced from the CFD simulation using Eq. (1) and has been compared to the experimental results and the Perry model in Fig. 6.
VALIDATION

Fig. 6 shows that, contrary to the Perry model, \( C_q \) is practically constant within the sonic range, as the experiment-based data built in the 1D AMESim model show. To explore the reasons of departure is a subject of future work. The measurement results show that the simple 1D model derived from Eq. (1) can be effectively used for simulation of this EP valve with a constant \( C_q \) value of 0.74 in the sonic flow range. This value is the same as the one that can be computed from the Perry polynomial for the critical pressure ratio of 0.53. Nevertheless, at smaller pressure ratios the difference between the Perry model and the measurement increases continuously. The constant \( C_q \) tendency has been properly resolved by the FLUENT Q3D computations. The \( C_q \) value computed from the Q3D model is approximately 0.8, so the relative difference between the Q3D model and the measurement is about 8%. Possible causes of this deviation are the geometrical simplification, the measurement uncertainties and the residual errors from the discretization.

Based on the above, it can be stated that the FLUENT Q3D results resolve the 1D measurement data qualitatively and, within a certain tolerance, quantitatively as well.

MODELS WITH DIFFERENT GEOMETRIES

In order to analyze the influence of the geometry on the flow parameters, a number of different Q3D models were prepared, based on the previously validated FLUENT code. The current investigations were concentrated on the angle of the plate \( \alpha \) at the inlet of the orifice (see Fig. 4b.).

The minimum distance between the valve body and the orifice body were kept constant. Fig. 7 shows the geometries of the largest and the smallest angles, and the numerical grid applied for the discretization as well. The pressure ratio domain was widened to incorporate smaller pressure differences as well, in order to obtain a picture of the total operating range of the EP valve.

Fig. 7. Geometries of the largest (20º) and the smallest (-20º) angles used

SIMULATION RESULTS

In Fig. 8 the results for the different geometries are plotted against the pressure ratio. The values of the Perry model are also shown for comparison purposes.

Fig. 8. \( C_q \) values of the different geometries

Fig. 8 shows that the influence of the angle variation is very small, hardly more than the difference between the measured and simulated values obtained in the validation phase (10.6% vs. 8%). Nevertheless, there is a tendency which shows that the increase of the angle decreases the flow coefficient, as the separation bubble developing at the inlet edge of the orifice reduces the flow cross-section. The angle variation has more influence at higher pressure ratios, as it can be seen in Fig. 9. Furthermore, it is apparent that the domain can be separated into two parts: at lower pressure ratios up to 0.4 the difference between the \( C_q \) values for the largest and smallest angles almost remains constant, and then starts increasing linearly from there (see the two jagged lines in Fig. 9.). This partitioning is the same as it is with the flow coefficient characteristics: \( C_q \) remains constant at lower pressure ratios, and starts to decrease at about 0.4-0.45.

The difference between the simulation and the Perry model is quite large, reaching almost 25% at higher pressure ratios. Moreover, the computed characteristics are quite different from the Perry model as well, particularly at lower pressure ratios. Such departure is a subject of future work.

Fig. 9. \( C_q \) differences of the different geometries

Fig. 9. differences between min. and max. \( C_q \) values
CONCLUSION AND FUTURE REMARKS

With the results concluded, a simple 1D model was developed which predicts the flow coefficient within a certain tolerance (8%). This model can be used effectively in complicated systems with high calculation speed and precision. The database built from the flow coefficient values for different geometries can be useful at later developments. The difference between the largest and smallest angles is somewhat low, but can be just enough to fine-tune existing or future systems.

ACKNOWLEDGEMENT

This work has been supported by the Hungarian National Fund for Science and Research under contract No. OTKA T 038184.

REFERENCES


