Numerical Simulations for Transonic Flow in Control Valve

Pawel Flaszynski[1], Piotr Doerffer[1], Grzegorz Mikułowski[2], Jan Holnicki-Szulc[2]

Address   [1]Institute of Fluid Flow Machinery Polish Academy of Sciences (IMP PAN), Gdansk, Poland
        [2]Institute of Fundamental Technological Research Polish Academy of Sciences (IPPT PAN), Warsaw, Poland

Abstract: Results of numerical simulations for transonic flow in control valve are presented. The valve is the main part of an adaptive pneumatic shock absorber. Flow structure in the valve domain and the influence of the flow non-uniformity in the valve on a mass flow rate is investigated. Numerical simulation results are compared with experimental data.

Keywords: pneumatic valve, transonic flow, numerical simulations

Introduction

An important trend of "greening" the modes of transport is being widely observed presently. This tendency, brought about by economical and ecological reasons, is noticeable in many transportation branches, e.g. electric vehicles in surface transport. One of such 'eco' problems is the necessity of using large amounts of technical fluids that are difficult to recycle, which makes them expensive in service. For that reason a noticeable trend of promoting the fluid-less technologies occurred. In the case of air vehicles another important aspect is also considered - relatively large weight of the hydraulic systems. The examples of removing the hydraulic installations from aircraft are fly-by-wire, fly-by-light or fly-by-wireless systems [1]. The example of a classical fluid-based component are shock absorbers, where the operational medium is hydraulic fluid together with a pressurised gas. These devices are of relatively small size and high efficiency [2]. A fluid-less alternative for the fluidic designs can be purely pneumatic solution. However, the classical passive pneumatic absorbers have efficiency limited to the level of 50 % [2], which disqualify them in the competition with the mentioned oleo-pneumatic designs characterized by 80% efficiency factor. The presented work is a part of study aimed at development of a new concept of an pneumatic adaptive shock absorber with ability of semi-active energy dissipation.

The adaptive pneumatic shock absorber concept was assumed to be a piston-cylinder device with an active high performance valve incorporated into the piston. This valve enabled regulating the gas pressure levels in the absorber and therefore allowed to adapt the mechanical response of the device to the particular impact energy. Because of the fact that the shock absorber was devoted to be adjusted in real-time under impact conditions, the valve must have had: electronic controllability, mechanical response time less than 2 ms [3], compact design and predefined mass flow rate of certain value [4]. These requirements were impossible to be met with the common electromechanical systems so the designs with the use of functional materials were taken under consideration.

The investigations of the flow structure influence on the mass flow rate in the valve are presented in the paper

Corresponding author : Pawel Flaszynski pflaszyn@pg.gda.pl

m	mass flow rate (g/s)	**Subscripts**	
p	pressure (Pa)	0	total conditions in container
R	gas constant [J/(mol K)]	1	valve inlet
T	temperature [K]	2	valve outlet
V	volume of container (m^3)		

Experimental Setup

Operational characteristics of the valve were obtained using a stand, which allowed to conduct experiments under conditions of pressure gradient. The set-up consisted of two gas accumulators connected with a pipeline that had a valve holder. The stand instrumentation allowed to measure and acquire the thermodynamic parameters of the gas as depicted in Fig. 1.

The experimental program consisted of a series of flow experiments aiming at determination of mass flow rate on the valve according to the pressure drop p1-p2, inlet pressure p1 and the gas temperature T1. The gas used in the experiments was technically pure nitrogen. A single test was started with filling the inlet container up to a predefined initial pressure and cutting off the supply installation. During the test, the accumulated gas flew through the valve to the outlet container in which the conditions were determined by an outlet pressure regulator set up. The range of gas temperatures, which occurred

during the tests allowed to assume the ideal-gas relation. By considering the inlet container as of constant volume and the process as developed in equilibrium states, it was possible to calculate the mass of gas in the container at any time instant on the basis of measurement of the stagnation pressure p0 and the stagnation temperature T0. By differentiation of the ideal gas law the rate of the gas outflow from the container was calculated:

$$m = \frac{V}{R} \frac{d}{dt} \left(\frac{p_0(t)}{T_0(t)} \right) \qquad (1)$$

On the basis of the mass continuity principle, the container discharge rate was assumed to be equal to mass flow rate on the tested valve. In order to obtain a predefined time history of the pressure drop in the testing system the pressure regulator was utilized on the outlet of the second container. The regulator provided a real-time control of the pressure value in the pipeline behind the tested valve.
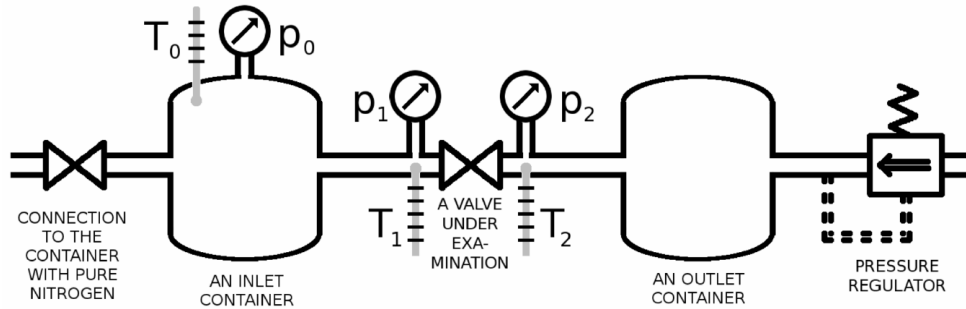

Fig.1 Scheme of the set-up for the valve performance measurements

Geometry and Numerical Model Description

The configuration of the valve assembled of the two plates is investigated (Fig.2). The similar configuration can also be found in the airplane suspension system. In the shown configuration, the first plate is fixed, the second one is translated along the axis. The translation can be driven by piezo-actuators. If the valve is opened the flow is going through the cylindrical holes and the gap between the plates. The valve is working at the high pressure difference and low axial translation (low gap height – distance between plates). There are three

important features of the presented valve geometry: holes distribution, circumferential grooves and gap height. The circumferential grooves have 0.6 mm height and 0.6 mm width. The critical parameter is the gap height (0.05 mm) between the plates, which has a strong influence on the flow distribution and valve performance.

Computational domain is shown in Fig.3. The cylindrical holes at the both plates and circumferential grooves are shown. The inlet plate is marked by blue color. The unstructured hexahedra mesh is generated by means Hexpress/Numeca. The mesh consists of ~7.5M

cells and it is refined close to the wall, so y^+ is less than 3 in holes and grooves. General view on the mesh in the hole and the gap area is presented in Fig. 4 (left). The simulations are performed for two configurations which differ to each other by the drilling hole depth. The difference can be noticed in Fig.4, where configuration G1 and G2 are shown. Inflow area from the holes to the groove domain is larger in case G2.

Numerical simulations, presented in the paper, are carried out in Ansys/Fluent v14. Second order upwind scheme for convection terms for transport equations and ideal gas model for nitrogen is applied.

The performance of the valve ($m=m(p_2/p_1)$ is obtained for k-ω SST model [6].

Total conditions are applied at the inlet. Results for two total pressure values are presented: 600 kPa and 800 kPa. In both cases total temperature is equal to 293 K. Turbulence intensity is set 0.5% and turbulent viscosity ratio 10. At the outlet static pressure is applied according to the required pressure ratio in the valve.

Table 1 Turbulence model influence on mass flow rate

	SST	SA - laminar
m/m_{SA}	0.998	0.992

A-A

Fig. 2 Valve main elements

Fig. 3 Computational domain (inlet plane colored by blue)

The gap height between the plates is very low, so the question arises if the flow should be considered as the laminar or turbulent one. The additional simulations are done for Spalart-Allmaras turbulence model [7] and the zonal approach, where the laminar flow is assumed in the gap and out of this domain Spalart-Almaras model is applied. Results for the selected flow condition are presented in Tab.1. Mass flow rate for turbulent case with k-ω SST model and for "laminar" approach are related to mass flow rate obtained for Spalart-Allmaras
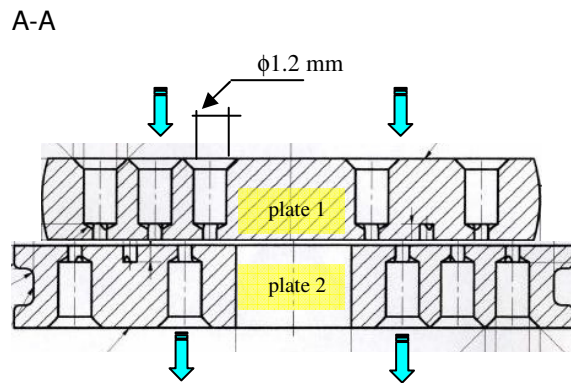
model. It is shown that differences are less than 1% in both cases.

Flow structure and Valve Performance

Flow distribution in the gap influences on the valve performance. General view on the plane at the middle height of the gap for G1 configuration is shown in Fig. 5. One can see the impinging jet areas downstream of the holes (plate 1). Flow structure near the holes in the circumferential groove is visualized by streamlines and velocity magnitude distribution at the plane in the gap. The flow is contracted in the hole outlet and the impinging jet area is small. Vortical structures are created in the groove between the neigbouring holes. Such vortices influence on the flow distribution at the gap inlet. The highest inflow velocity is observed near the jet, it decreases further from the jet in circumferential direction. Non-uniform distribution of the flow between the plates influences on locally high velocities reaching supersonic values. It is dependent on the pressure ratio (p_2/p_1), but even lower values local expansion leads to supersonic flow.
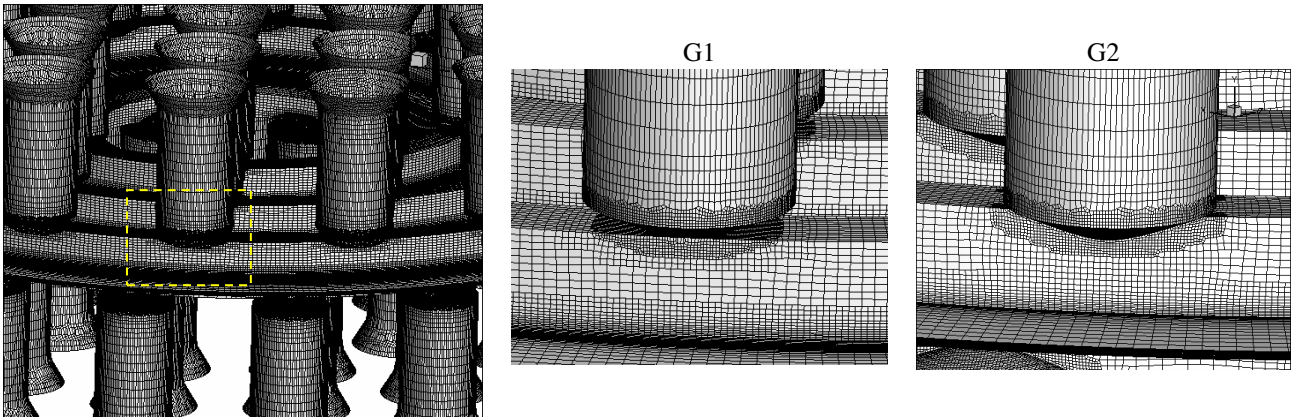.

Fig. 4 Computational mesh on holes and grooves walls (left) and close-up of two geometry configurations G1 (middle) and G2 (right)

Such non-uniformity is shown in Mach number distribution at the gap. In Fig. 7, the two cases are compared: G1 and G2. Mach number distribution for G2 configuration indicates much less non-uniformity

Close-up of Mach number distribution near the selected holes is shown in Fig. 8. On the left side, the results for G1 are presented. One can notice the rising circumferential non-uniformity at the holes located further fro the center of the plate. It arises from higher mass flow rate at the outer ring of holes than at the inner one. White areas in Mach number distribution indicate supersonic flow. In case of G2 configuration, the flow contraction at the holes outlet is much lower, so the flow in the groove is much more uniform than in G1case. At the same flow conditions (shown in Fig.8) the inflow to the gap domain is subsonic in G2 case. The both cases differ from each other not only with Mach number at the gap inlet, but at the gap outlet also. However, at the outlet in G2 Mach number is higher, what arises from lower losses in the gap and uniform distribution at the gap outlet (inflow to grooves in plate 2).

Different flow structure and flow distribution in the gap influences on the mass flow rate in the valve. Increase of the mass flow rate for G2 in comparison with G1 is shown in Fig.9. The experimental data and numerical results for G2 configuration are compared in Fig.10.
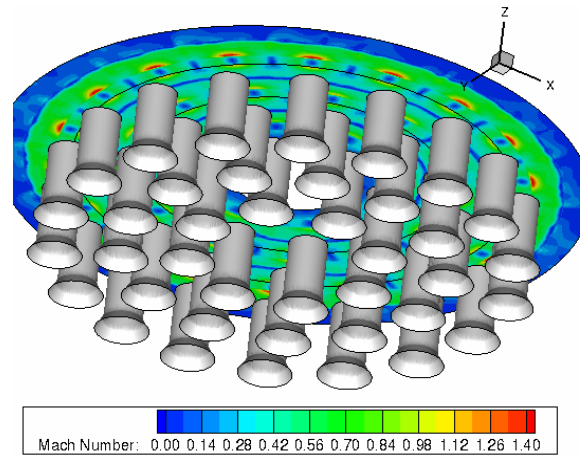

Fig. 5 Mach number in the middle plane between plates (view from plate 1)
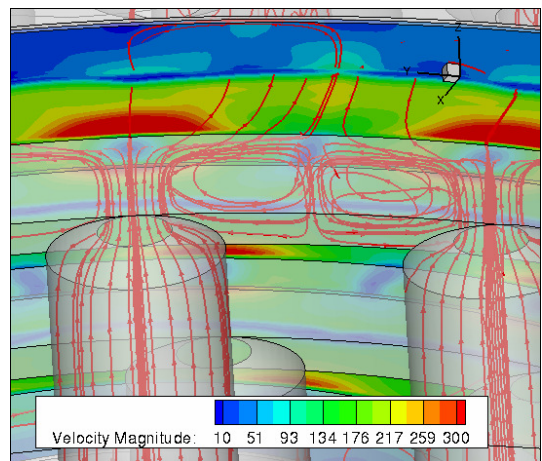

Fig. 6 Streamlines in plate 1 (holes and groove) and velocity magnitude in the middle plane between plates (view from plate 1)
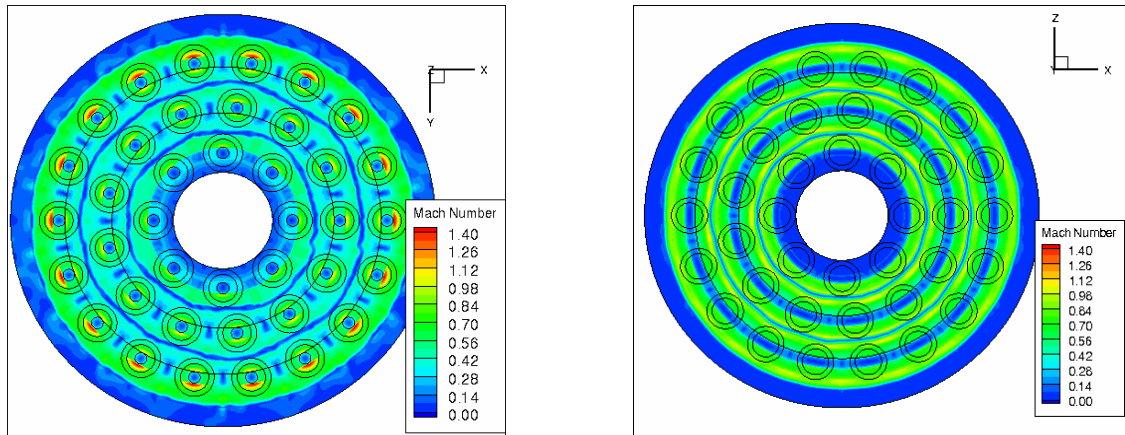
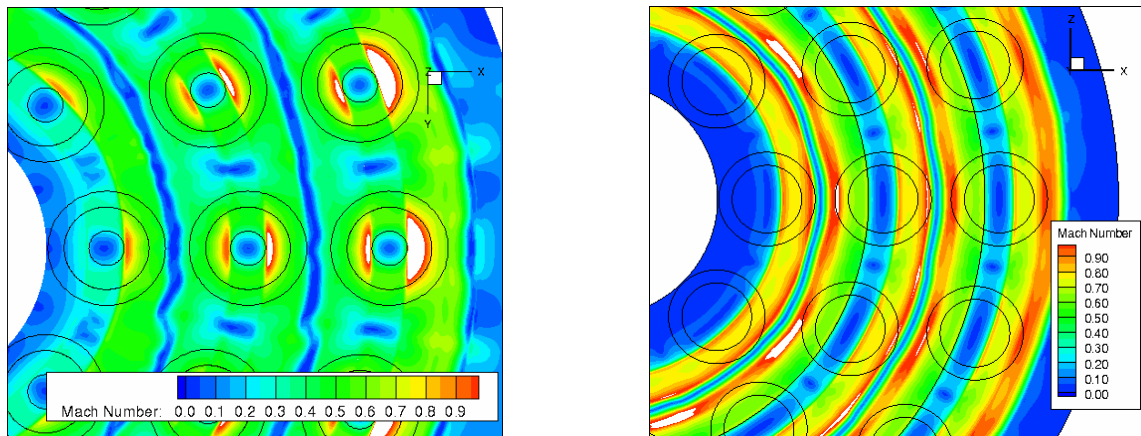**Fig. 7** Mach number in the middle plane between plates for G1 (left) and G2 (right)

Fig. 8 Mach number in the middle plane between plates for G1 (left) and G2 (right) – view on selected holes
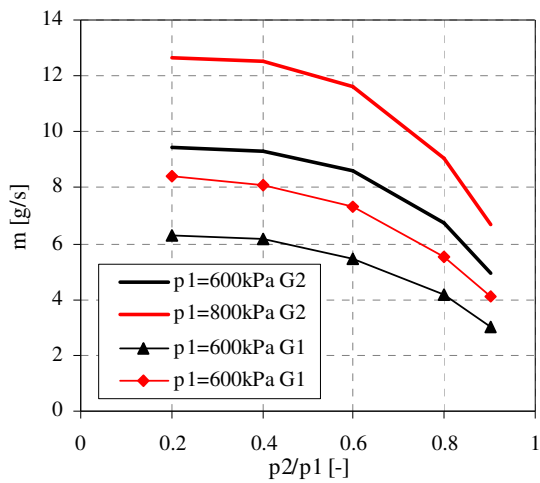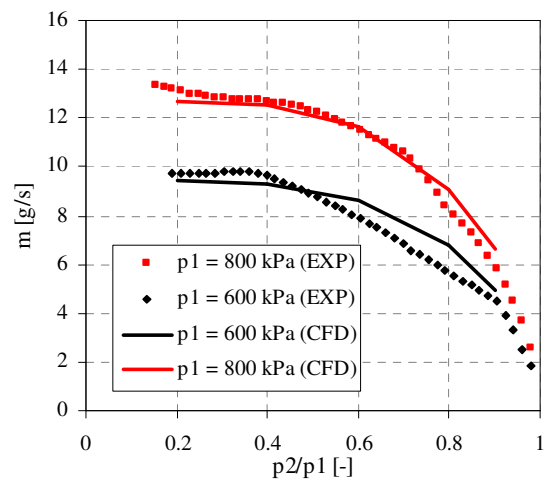

Fig. 9 Mass flow rate in valve (G1 and G2)-CFD

Fig. 10 Mass flow rate in valve (G2)- CFD vs. EXP

Summary

Numerical simulations are carried out for the control valve and the results indicate of good agreement with experimental data. It was found that there are no meaningful differences between the turbulent and laminar flow approach in the gap domain, in the investigated case.

Flow structure non-uniformity in the gap between the plates arises from the inlet geometry configuration and the ratio of the lower holes outlet area to the circumferential groove area.

Acknowledgement

References

[1]   Isermann R.. Mechatronics systems | innovative products with embedded control. Control Engineering Practice, 16:14{29, 2008.

[2]   Currey N.S. Aircraft landing gear design: principles and pratices. AIAA, 1988

[3]   Mikulowski G., Pawlowski P., Graczykowski C., Wiazowaty R., and Holnicki-Szulc J. On a pneumatic adaptive landing gear system for a small aerial vehicle. In roceedings of the IV Eccomas thematic conference on Smart structures and materials, Porto, 13-15 July 2009.

[4]   Wiazowaty R., Biczyk J., Graczykowski C., Mikulowski G. Method of impact energy dissipation by the use of the pneumatic impact absorber with a piezo-valve. In Proceedings of the 5th ECCOMAS Thematic Conference on Smart Structures and Materials SMART'11, Saarland University, Saarbrucken, July 6 - 8 2011.

[5]   Mikulowski G., Holnicki-Szulc J. Adaptive landing gear concept-feedback control validation. Smart Mater. Struct. , 16:2146-2158, January 2007.

[6]   Menter F.R., Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA Journal. 32(8). 1598–1605. August 1994

[7]   Spalart, P. R. and Allmaras, S. R., 1992, "A One-Equation Turbulence Model for Aerodynamic Flows" AIAA Paper 92-0439