A Seat Microvalve Nozzle for Optimal Gas Flow Capacity at Large Controlled Pressure

Wouter van der Wijngaart\textsuperscript{1}, Anders Thorsén\textsuperscript{2}, and Göran Stemme\textsuperscript{1}

\textsuperscript{1}Royal Institute of Technology, Dept. of Signals, Sensor and Systems, Stockholm, Sweden
\textsuperscript{2}Acreo AB, MicroTechnology, Kista/Stockholm and Lund, Sweden

Abstract
Seat microvalves are the most common microvalve type for gas flow control. This paper presents a general method for optimising the flow capacity of a seat valve nozzle and diminishing the requirements on the valve actuator’s stroke-length. Geometrical analysis and finite element (FE) simulations show that for controlling large gas flow at elevated pressure, the optimal nozzle design in terms of flow capacity for a given actuator performance is a multiple-orifice arrangement with miniaturised circular nozzles. Experimental results support the design introduced in this paper.

1 Introduction
Most of today’s gas microvalves are developed for applications requiring leak tightness. They are of the type illustrated in Fig.1, having a boss or a membrane structure suspended above a substrate with an orifice, surrounded by a seat. Because the pneumatic force acts in the same direction as the boss moves, an actuation force proportional to the controlled pressure and the orifice area is required.

Comparing micromachined gas microvalves with conventional valves in the same price class, the flow capacity is typically a factor 10 lower for the micromachined ones. There are two main reasons: the small cross-sectional areas in micromachined ducts form flow restrictions and the high pneumatic force (commonly several bar) demands high actuator performance. However, microactuators are limited either in terms of the force or the stroke they can deliver.

Previously published seat microvalve designs [1-6] are overviewed in Table 1. The actuator stroke of devices [1-3, 5] is one order of magnitude smaller than their orifice size and the control gap forms a limit to the flow handling capacity. Further, the actuator area of the devices [2-6] is one order of magnitude larger than the orifice size and the actuator size rather than the flow path dimensions determine the device’s footprint area.

In this paper a novel design approach is introduced optimising the seat microvalve design by minimising the required device footprint area and actuator stroke.

2 Concept & Design
The mass flow, $\dot{m}$, of a non-viscous compressible gas through a hole (area $A$) in a thin plate can for subsonic flow be written as [7, 8]:

$$\dot{m} = \frac{P_m C_d A}{\sqrt{R T}} \sqrt{\frac{P_m}{P_a}} \frac{1}{\gamma} \left(\frac{P_m}{P_a}\right)^{\frac{\gamma+1}{2\gamma}} - 1 = C_d(P_m, P_a) \cdot A$$

(1)

where $P_m$ and $P_a$ are the upstream and downstream stagnation pressures, $\gamma$ the ratio of specific heats for the gas, $R$ is the universal gas constant over molecular weight ratio, $C_d$ is the gas discharge coefficient and $T$ the upstream temperature of the gas.

For sonic flow the mass flow can be written as

Table 1. Overview of seat microvalves with typical dimensions.

<table>
<thead>
<tr>
<th>Ref</th>
<th>Actuation principle</th>
<th>Orifice size</th>
<th>Actuator stroke-length</th>
<th>Actuator size</th>
<th>Pneumatic performance</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>[1]</td>
<td>electrostatic / pneumatic</td>
<td>$\varnothing$ 3 mm</td>
<td>$5 \mu m$</td>
<td>3.6 mm</td>
<td>800 sccm @ 550 kPa</td>
<td>Pressure balanced</td>
</tr>
<tr>
<td>[2]</td>
<td>Bimetal</td>
<td>$\square$ 230 $\mu$m</td>
<td>27 $\mu$m</td>
<td>2.5 mm</td>
<td>100 sccm @ 170 kPa</td>
<td></td>
</tr>
<tr>
<td>[3]</td>
<td>Bimetal</td>
<td>$\square$ 245 $\mu$m</td>
<td>30 $\mu$m</td>
<td>4 mm $\times$ 4 mm</td>
<td>1 150 sccm @ 800 kPa</td>
<td>3-2 way valve</td>
</tr>
<tr>
<td>[4]</td>
<td>Bimetal</td>
<td>$\square$ 180 $\mu$m</td>
<td>100 $\mu$m</td>
<td>8.8 mm (chip)</td>
<td>1 000 sccm @ 1.03 MPa</td>
<td>Hewlett Packard</td>
</tr>
<tr>
<td>[5]</td>
<td>SCE</td>
<td>$\varnothing$ 1.5 mm</td>
<td>$50 \mu$m</td>
<td>6 mm $\times$ 6 mm</td>
<td>5 000 sccm @ 500 kPa</td>
<td>Fluistor™, Redwood Microsystems</td>
</tr>
<tr>
<td>[6]</td>
<td>SMA</td>
<td>$\square$ 100-500 $\mu$m</td>
<td>150 $\mu$m</td>
<td>8 mm $\times$ 5 mm</td>
<td>1 000 sccm @ 200 kPa</td>
<td>TiNi Alloy Company</td>
</tr>
</tbody>
</table>
\[ m = P_m C_d A \frac{\sqrt{\gamma}}{\sqrt{T}} = C_2 (P_m) \cdot A \]

where \( C_1 \) and \( C_2 \) are the respective mass flow per unit of flow cross-sectional area and depend only on the pressure, the upstream gas temperature and the gas properties.

Typically one single flow restricting cross-sectional area determines the flow capacity of a valve. In a seat microvalve (Fig. 1), this minimum cross-sectional area is either the seat area, \( A_{\text{seat}} \), between the valve seat and the boss, perpendicular to the substrate, or the orifice area, \( A_{\text{orifice}} \), in the substrate enclosed by the seat. These configurations will further be called seat-controlled, respectively orifice-controlled. The orifice area determines the pneumatic force exerted on the boss and thus the actuator performance requirements. Increasing a valve’s flow capacity by enlarging the orifice area demands a more powerful (larger) actuator. Enlarging only the seat area does not pose this drawback. For an optimal design, the seat area consequently should be maximised, while the orifice area should be kept minimal and form the one flow-restricting cross-sectional area. The orifice area is then fixed by the required flow performance, \( m_{\text{req}} \), under certain work pressure conditions:

\[ A_{\text{orifice}} = \frac{m_{\text{req}}}{C_i}, \quad i = 1 \text{ or } 2 \text{ for subsonic resp sonic flow}. \]

Edge induced flow losses start playing a role at small orifice dimensions. In order to minimise these edge effects, the optimal orifice shape must be circular (geometrical area with the minimum perimeter).

For a nozzle with a circular orifice with radius \( r \) and stroke \( z \) in seat-controlled designs, the mass-flow per orifice footprint area \( \dot{m}_{fp} \) can be expressed as

\[ \dot{m}_{fp}(r, z) = \frac{m}{A_{\text{orifice}}} \cdot C_i A_{\text{seat}} = \frac{C_i \cdot 2 \pi r z}{r^2} = C_i \cdot \frac{2 z}{r} \cdot \eta \cdot C_i \]

where \( \eta = A_{\text{seat}} / A_{\text{orifice}} = 2z/r \) has been introduced. For seat-controlled designs, \( \eta < 1 \). Similar, in orifice-controlled designs (\( \eta > 1 \)) the flow capacity is expressed as

\[ \dot{m}_{fp}(r, z) = \frac{m}{A_{\text{orifice}}} \cdot C_i A_{\text{orifice}} = C_i. \]

In the latter designs, the flow capacity is thus independent from \( r \) and \( z \), and the actuator stroke \( z \) (and orifice radius \( r \)) can be reduced as long as \( \eta > 1 \), i.e. \( z > r/2 \), without limiting the valve’s flow capacity. This diminishes the required actuator performance.

The analysis above is valid for compressible, turbulent gas flows as long as the turbulent boundary layers are thin compared with the dimensions. Then the non-viscid flow is an acceptable approximation. According to [9] flow through an orifice has a transition Reynolds number around 15. For air at atmospheric pressure and speed of sound, this equals an orifice diameter of about 0.7 \( \mu \text{m} \). As long as the dimensions

<table>
<thead>
<tr>
<th>Nozzle</th>
<th>Orifice area +</th>
<th>Seat length</th>
</tr>
</thead>
<tbody>
<tr>
<td>( n=1 )</td>
<td>( A_{\text{orifice}}=A_0 )</td>
<td>( S=S_0 )</td>
</tr>
<tr>
<td>( n=4 )</td>
<td>( A_{\text{orifice}}=A_0 )</td>
<td>( S=2S_0 )</td>
</tr>
<tr>
<td>( n=16 )</td>
<td>( A_{\text{orifice}}=A_0 )</td>
<td>( S=4S_0 )</td>
</tr>
</tbody>
</table>

Figure 2. Schematic visualisation of the increasing total seat length \( S \) for an increasing miniaturisation factor \( n \) at a constant total orifice area \( A_{\text{orifice}} \).

are significantly larger it is convenient to assume the flow turbulent.

For turbulent flow, the losses mainly occur in the turbulent boundary layers and these losses increase in significance as the dimensions decrease and limit the minimum size of a practical nozzle giving an \( A_{\text{orifice}, \text{min}} \) and \( r_{\text{min}} \). This is not predicted by the above analytic model, but will later be considered in numerical simulations.

An optimised nozzle will now have a flow \( C_i A_{\text{orifice}, \text{min}} \) while an optimised valve has specific required overall mass flow \( m_{\text{req}} \). This flow can be achieved using a multiple-orifice design with \( n = \frac{m_{\text{req}}}{C_i A_{\text{orifice}, \text{min}}} \) parallel orifices and an actuator stroke not larger than \( z_{\text{opt}} = \frac{r_{\text{min}}}{2} \).

Fig. 2 illustrates how at a constant total (summed) orifice area \( (A_{\text{orifice}}) \) for an increasing value of \( n \), the seat length \( S \), and thus the total seat area \( (A_{\text{seat}}) \), increases \( \sqrt{n} \) times.

Figure 3. Two implementations of the novel multiple-nozzle design. In the top design, the gas flows through the device from top to bottom. The actuator is positioned on the side of the boss structures. In the bottom design, an in-plane flow feeds the orifices. The actuator is positioned on top of the boss structures.
The reduced control gap/stroke $z_{opt}$ makes stroke-limited actuation principles, such as electrostatic actuation [1,10] and stacked piezoelectric actuation, feasible for control of large flows. Two implementations of multiple-nozzle designs are shown in Fig. 3.

As mentioned above, flow losses will eventually limit the nozzle miniaturisation. Moreover, the cleanliness of the handled gas puts a practical limit to the nozzle miniaturisation due to the risk for clogging. Particle sensitivity of the increased seat length needs to be considered here. Yet another limitation is posed by the manufacturability of increasingly small nozzles.

### 3 CFD SIMULATIONS

To support the proposed optimisation, simulations were done using the computational fluid dynamics (CFD) software ANSYS/FLOTRAN (Fig. 4). Details on the simulations and a full result overview can be found in [11].

![Figure 4. 3D illustrations of FE simulated pressure distributions in a nozzle.](image)

Figure 4. 3D illustrations of FE simulated pressure distributions in a nozzle. The left plot shows the seat-controlled pressure distribution for a $z=2.5$ µm stroke with the main pressure drop at the seat area. The right plot shows the orifice-controlled pressure distribution for a $z=40$ µm stroke, with the main pressure drop at the orifice area. The orifice radius $r=10$ µm, the boss radius 13 µm, the substrate thickness 10 µm, and the inlet pressure $P_{in}=100$ kPa, relative to atmospheric pressure $P_{out}$.

### 4 DISCUSSION

An overview of the $r-z$ parameter optimisation space is shown in Fig. 5. The analytic mass-flow per orifice area for a 100 kPa pressure drop is greyscale plotted as background. The parameter values for which $\eta=1$ divide the area in a seat-controlled region and an orifice-controlled region. Results from FE analysis are displayed as round, greyscale-coded dots. Note the overall (grey-tone) agreement between the numerical results (dots) and the analytic model (background). The mass flow per unit of orifice area is independent of both the orifice radius and the stroke-length when the flow is orifice-controlled. The optimal nozzle design parameters are at the junction of the $r=r_{min}$ line and the $\eta=1$ line. The white squares indicate earlier reported microvalve nozzle designs [2-6]; for valves with square orifices with side $x$, the radius position is approximated as the value $r=x/2$.

For values of $r>r_{min}$ ($r_{min}=10$ µm in the FE model) numeric and analytic results are in accordance. Where $r<r_{min}$ numeric results show the strong flow losses not accounted for in the analytic model. ANSYS uses the “Log-Law of the Wall” to predict the losses in the turbulent boundary layer and the used default values for ANSYS correspond to smooth walls [12]. Modified FEA parameters can be used to simulate different wall roughness, but this is not included in this work. The model may need adaptation to include gas rarification effects [13] and the scope of such study is too large to be included in this paper and should be subject to a general completion of the Moody chart.

### 5 ALTERNATIVE DESIGNS

A different approach for increasing the seat area was described earlier [7,14] and involves the shaping of the seat in a meander. For the same total footprint...
area, however, such design will always require a larger actuator stroke, and the mass flow per footprint area in this optimisation procedure therefore converges to a sub-optimum. Fig. 6 illustrates that the seat length of the meander shape can always be further increased, and thus the required actuator stroke decreased, by cutting of the meander arms and arranging them as separate orifices. Repeating this procedure results eventually in an improved design, similar to the multiple-orifice design described above.

6 EXPERIMENTAL

A. K. Henning, partially basing on our preparatory work [15], constructed and tested nozzle structures that confirm the flow analysis [7]. He also suggests an exponential curve fitting to describe the transition between orifice-controlled and seat-controlled flow.

7 CONCLUSION

The above study shows that by changing the nozzle design of a seat microvalve, the flow through the valve can be optimised and the valve actuator requirements can be reduced. This results in an improved flow performance and a valve footprint area (cost) reduction. A novel nozzle design is presented, containing multiple orifices and a limited stroke-length. Design limits caused by manufacturability, flow losses, gas cleanness and particle sensitivity are identified. Analytic predictions and FE simulations confirm the geometrical analysis of the nozzle design.

ACKNOWLEDGEMENTS

This work was partly supported by the Swedish Agency for Innovation Systems.

REFERENCES