ABCM Symposium Series in Mechatronics - Vol. 3 - pp.510-518

ANALYSIS OF PNEUMATIC DIRECTIONAL PROPORTIONAL VALVE WITH CFX MESH MOTION TECHNIQUE

Yansheng Jiang, yanshengjiang@hotmail.com
Antonio Carlos Valdiero, valdiero@unijui.edu.br
Pedro Luís Andrighetto, pedro@unijui.edu.br
Wang Chong, wang@unijui.edu.br
Luís Antonio Bortolaia, luis.bortolaia@unijui.edu.br
UNIJUI- Universidade Regional do Noroeste do Estado do Rio Grande do Sul,
Departamento de Tecnologia, Campus Panambi
Caixa Postal 121, CEP 98280-000, Panambi, RS, Brasil

Abstract. *An analysis of directional proportional pneumatic valve with a computational fluid dynamic software, ANSYS-CFX, is presented. The spool sliding movement is defined with a sinus function of time. Aerodynamic spool valve forces are computed in taking into account the fluid domain deformation. This was reached by using the so-called mesh motion technique of ANSYS-CFX.*

Keywords*: pneumatics, CFD simulation, proportional directional valve, fluid power.*

1. INTRODUCTION

Pneumatic actuators are very common in industrial application because they have easy and simple maintenance, relatively low cost, self cooling properties, good power density (power/dimension rate), fast acting with high accelerations and installation flexibility. Also, compressed air is available in almost all industry plants. This requires the use of pneumatic proportional valves and development of suitable control techniques in order to deal with nonlinearities, which are common in servo pneumatics. However, high costs, a few of available manufacturers and the necessity of importing abroad are factors which impact many industrial applications with this type of valve. For this reason, we are developing a pneumatic proportional directional valve with sliding spool.

Conventionally, the flow through pneumatic valve is calculated with theoretical or empirical formula. For example, Gouveia (1996) studied a pneumatic proportional valve of four ways with equations of uni-dimensional compressible fluid mechanics in following an earlier study by Andersen (1967).

The recent development and popularization of CFD software make it possible and easy to achieve a better understanding and solution of flow through the complex channel of various valves. However, we have found very few CFD analyses in relation to pneumatic application. In context of hydraulic application, Bao *et al*. (2001) performed a CFD approach to pressure loss analysis of hydraulic spool valve. Jiang *et al* (2006) analyzed with ANSYS-FLOTRAN the air flow inside the so called three-lands-five-ways-on-off pneumatic proportional valve, as shown in Fig.(1).

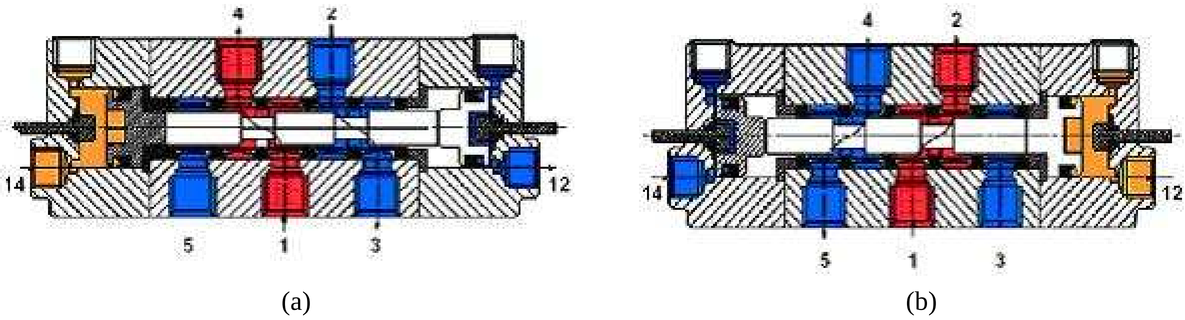


Figure 1. Directional proportional valve

The valve is connected with five ports numbered from 1 to 5 to the environmental pneumatic application system. Air is supplied through the port 1. In Fig. (1a), the right half area of the port 4 forms a by-pass area for the supplied air metering from the port 1 to 4. The right half of the port 2 forms a by-pass area for exhausted air metering out to the port 3. In Figure (1b), on the other hand, the right half of port 4 is closed for the supplied air while the left half of the port 2 is opened. This allows the supplied air metering from port 1 to port 2 and the exhausted air from port 4 to port 5.

The spool section is cylindrical with three lands and two chambers. The spool slides in a go-and-return way of a frequency equal to 20 Hz and of the stroke equal to 3 mm. As for materials to be used for manufacturing the valve, a first study conducted that the bore would be made of PVC and the spool of normal carbon steel.

In our last study (Jiang *et al*, 2006), we considered, as the first approximation, the sliding movement of valve spool surface as moving walls with constant velocity. This made it possible to avoid dealing the flow domain change and the use of fixed meshes of elements in stead of the moving mesh. As the wall velocity is constant, it was not possible to take into account the inertial effect due to the periodical sliding movement of valve spool.

In this study, we have investigated the use of the mesh motion technique available in ANSYS-CFX 10 in dealing with the flow domain change due to the sliding movement of valve spool.

In the following sections, the mathematical model for the turbulent compressible fluid is presented shortly. It is followed by a detail description of CFD geometrical model, finite element meshes, boundary conditions and solution strategies. Numerical results are discussed and lead to some conclusions..

2. MODEL ANALYSIS

Before performing CFD simulations, we have first checked the Reynolds number with target working conditions of the valve. It is found that the Reynolds number was as high as 5500 when the supply port is half opened. Hence, we have considered that the flow through the valve to be completely turbulent.

2.1. Mathematical Models

Governing equations for three-dimensional turbulent flow of compressible fluid were represented by the continuity, Reynolds equations and heat transport equations. For the sake of completeness of presentation, we write in the followings the set of governing equations valid in the context of this application. For detailed descriptions of these equations in their complete form, one may refer, for example, to Versteeg and Malalasekera (1995) .

The Continuity Equation (Eq. 1), where ρ is the fluid density and **U** is vector of the mean velocity.

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

The Reynolds equations (Eq. 2 to 4), where P is pressure, μ is the fluid viscosity, u’, v’ and w’ are turbulent fluctuating components of velocity.

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

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

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

In the Heat transport equation (Eq. 5), C_P is the specific heat, T is temperature, Γ_T is heat conduction coefficient. T’ is turbulent fluctuant of temperature.

$$\frac{\partial(\rho c_p T)}{\partial t} + \mathrm{div}(\rho T \mathbf{U}) = \mathrm{div}(\Gamma_T \cdot \mathrm{grad} T) + \left[-\frac{\partial\left(\rho \overline{u'T'}\right)}{\partial x} - \frac{\partial\left(\rho \overline{v'T'}\right)}{\partial y} - \frac{\partial\left(\rho \overline{w'T'}\right)}{\partial z} \right] \tag{5}$$

The equation of state (Eq. 6) for air considered as compressible fluid relates static pressure, density and temperature by the gas constant, R, and the ratio of specific heat at constant pressure to the specific heat at constant volume, γ=1.4.

$$\rho = \frac{P_0}{RT}\left(\frac{P}{P_0}\right)^{\frac{1}{\gamma}} \tag{6}$$

The standard k-ε turbulent model (Eq. 7 and 8) is used in this work:

$$\frac{\partial(\rho k)}{\partial t} + \mathrm{div}(\rho k \mathbf{U}) = \mathrm{div}\left(\frac{\mu_t}{\sigma_t} \cdot \mathrm{grad} k\right) + \mu_t \phi - \rho \varepsilon \tag{7}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\mathrm{div}(\rho\varepsilon\mathbf{U})=\mathrm{div}\left(\frac{\mu_t}{\sigma_t}\cdot \mathrm{grad}\varepsilon\right)+C_1\mu_t\frac{\varepsilon}{k}\phi-C_2\rho\frac{\varepsilon^2}{k} \tag{8}$$

in which k and ε are turbulent kinetic energy and dissipation respectively, μ_t is the turbulent viscosity, σ_t is the Schmidt number for the energy (temperature) equation, ϕ is viscous dissipation, C_1 is the multiplier of the shear rate generation term of the turbulent kinetic energy dissipation rate equation, C_2 is the multiplier of the dissipation source term in the turbulent kinetic energy dissipation rate equation.

2.2. CFD Geometry Models

With reference to Fig.(1), the flow from the supply port 1 to port 4 will be analysed in this work. Its geometry is three-dimensional and has one plan of symmetry as shown in Fig.(2) (we have inversed the direction of valve with the supply port 1 on top). As compared with the model used in our last staudy (Jiang *et al*, 2006), the exit cylinder was extended in this study. This is to avoid the vortex at the immediate exit of the chamber and achieve uniformed velocity distribution at the outlet.

In the system of actual coordinates used in this work, the *xy* plan is the plan of symmetry. Due to the symmetry, only half of fluid domain, i.e., with coordinates *z* greater than or equal to zero, was analysed.

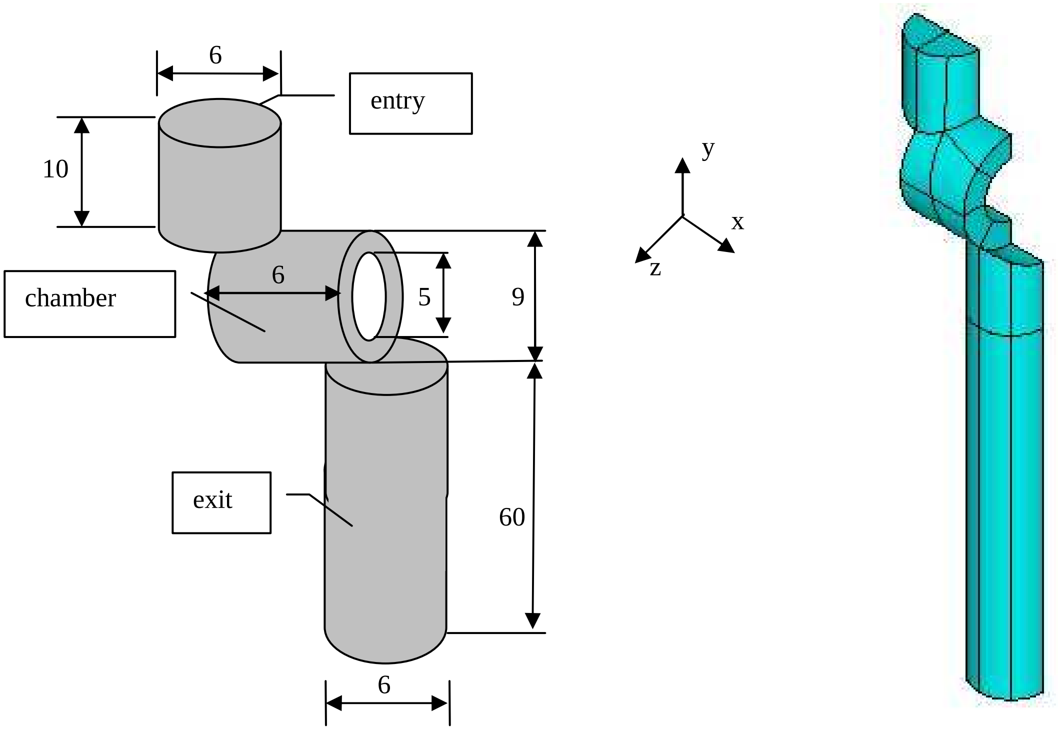


Figure 2. Geometry of proportional valve fluid domain and computational model

The spool slides rightwards from the position corresponding to half opening of the supply port. This results in the periodic movement of the left and right extreme sessions of the chamber. The movement is assumed to be in the following sinus function:

$$x = r\sin\omega t \tag{9}$$

in which x is mediated rightwards from the centre of the entry session, r is the radius of the entry, ω is the angular frequency of spool movement. When the time t is equal to zero, the left session of the chamber is at the half open position, i.e., at the centre of the entry session.

The angular frequency, ω, is related to the frequency of the spool go-return sliding movement, f, by the following well-known relation:

$$\omega = 2\pi f \tag{10}$$

As the frequency considered in this study is equal to 20 Hz, the value of the angular frequency is 40π rad/s. The period of the spool go-return movement is equal to 0.05 s. A quarter of this is the time for that the spool goes from the half open position to the close of the valve. That is 0.0125 s. At this time, the left session of the chamber arrives at the border of the entry session. No fluid can flows into the chamber from the entry.

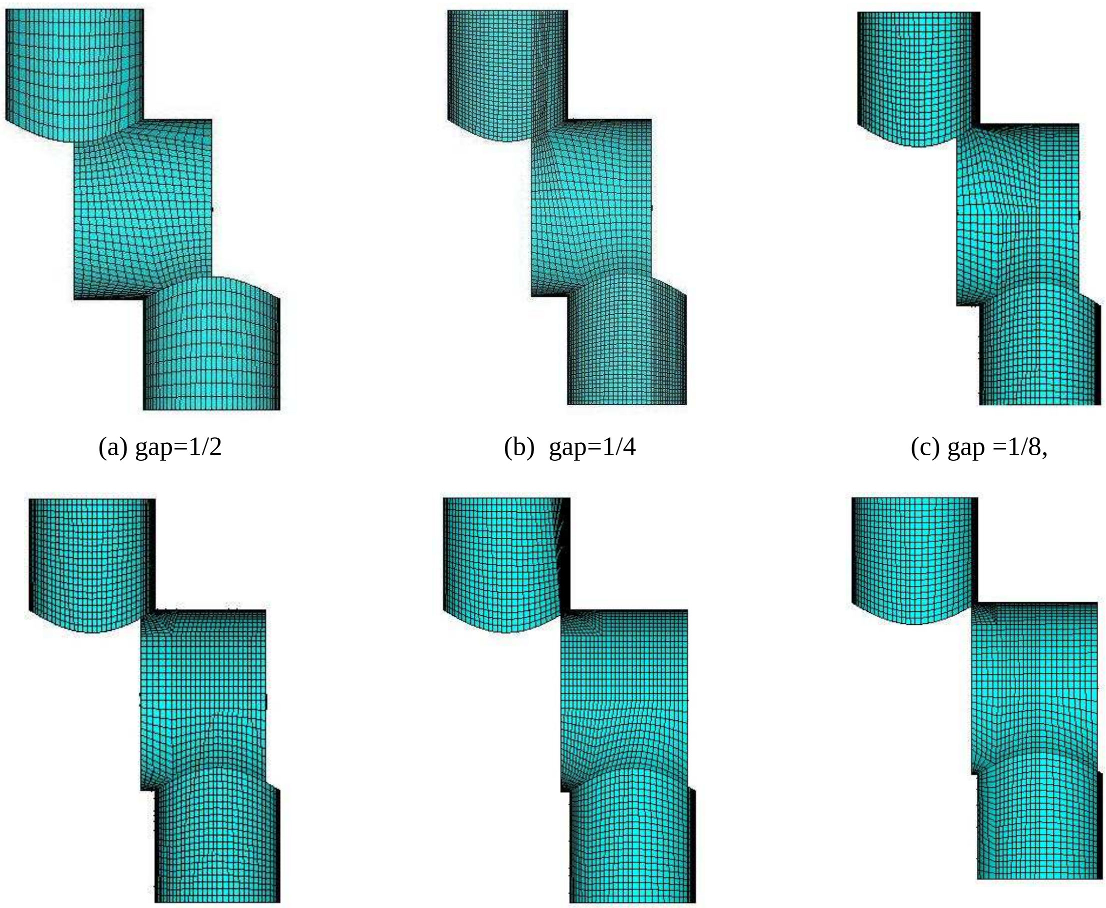
2.3. Numerical Solutions with ANSYS-CFX 10

In this work, we have used ANSYS-CFX 10 to achieve numerical solutions of the governing equations. The finite element mesh was built within ANSYS and exported to CFX.

From the point of computational fluid dynamic view, this problem may be classified as the transient flow with moving domain because the spool movement changes the chamber position with reference to the entry and the exit domains.

Transient flows with moving domain are usually solved in CFD with remeshing techniques. There are two kinds of remeshing techniques. One consists in changing the form of elements (through the change in nodal positions) but not in the topology of elements, i.e., the connectivity of elements remains unchanged. With more advanced remeshing techniques, both element form and connectivity may be subjected to changes during the flow domain deformation. Further more, some elements may disappear, some others be generated. Only with these advanced remeshing features that one can simulate the large deformation of fluid domain. Unfortunately, the mesh motion technique of ANSYS-CFX 10 does not possess these advanced remeshing features. It allows only the change in element forms. It is thus anticipated that the simulation of directional proportional pneumatic valve with ANSYS-CFX 10 can not go through the entire stroke of spool. Some elements may be distorted, when the spool displacement is too large, in such a way that their Jacobian turned to be negative and let to the computation aborted.

Given this situation, instead of trying to simulate continually the whole stroke, we investigated six particular positions of the chamber corresponding to the so-called 1/2, 1/4, 1/8, 1/16, 1/32 and 1/64 openings, as shown in Fig. 3. A 1/4 opening, for example, means that the ratio of the open area in the entry session, the gap, over the area of the entire entry session is equal to 1/4.



(a) gap=1/2 (b) gap=1/4 (c) gap =1/8,

(d) gap =1/16 (e) gap = 1/32 (f) gap = 1/64

Figure 3. Finite element meshes (for the sake of space, the exit cylinders are only partially represented.)

For each of above six meshes, we first considered a fixed domain problem and computed a CFD solution as we did in our earlier study (Jiang *et al*, 2006). With this solution as the initial solution, a moving domain problem is then solved in which the spool surface is moving continually rightwards following the relation given in Eq. 9. The spool displacement, xCylinder, is defined with ANSYS-CFX Expression Language as shown in Table 1. The value of xCylinder is applied as the displacement in x direction of Mesh Motion in ANSYS-CFX.

Table 1. Mesh motion parameters

Variables	Values defined in CFX
frequency	20 [s^-1]
nstep	50
omiga	2*pi*frequency
radius	3 [mm]
tStep	0.00001 [s]
tTotal	nSteps*tStep
tStart	in function of gaps as given in Table 2
phi	omiga*tStart
xCylinder	radius*sin(omiga*Time+phi) - radius*sin(phi)

As can be seen in Table 1, the time step is equal to 0.00001 s. The number of time steps is equal to 50. The total time that the spool surface undergoes the rightwards movement is equal to 0.00001 times 50, which results in 0.0005s. This represents 4% of the stroke time (0.0125 s). The resulted spool displacement is about 6% of the stroke (3 mm). We have chosen these values because otherwise, for larger values of the simulation time, most meshes had some elements degenerated and resulted in the negative volume.

Table 2. Initial time

Gaps	tStart (s)
1/2	0
1/4	0.00330928
1/8	0.00547191
1/16	0.00703601
1/32	0.0082156
1/64	0.00912436

Obviously, 4% of the stoke time or 6% of the stroke are small but enough to investigate the effect of the inertial forces on the spool valve due to its acceleration. In our earlier study (Jiang *et al*, 2006), we have assumed that this effect was small enough to be neglected. The argument was that the spool velocity was small as compared to the maximum velocity of air flow inside the chamber. However, this has not been proved. As shown in Table 3, although the spool velocity computed with Eq. 9 is indeed small, its acceleration is not. As the spool is reaching the close position, the acceleration value in modulo is as high as four times the gravitation ($9.81 m/s^2$). Hence, the effect of the inertial forces on the spool valve due to its acceleration needs to be investigated.

Table 3. Spool velocity and acceleration.

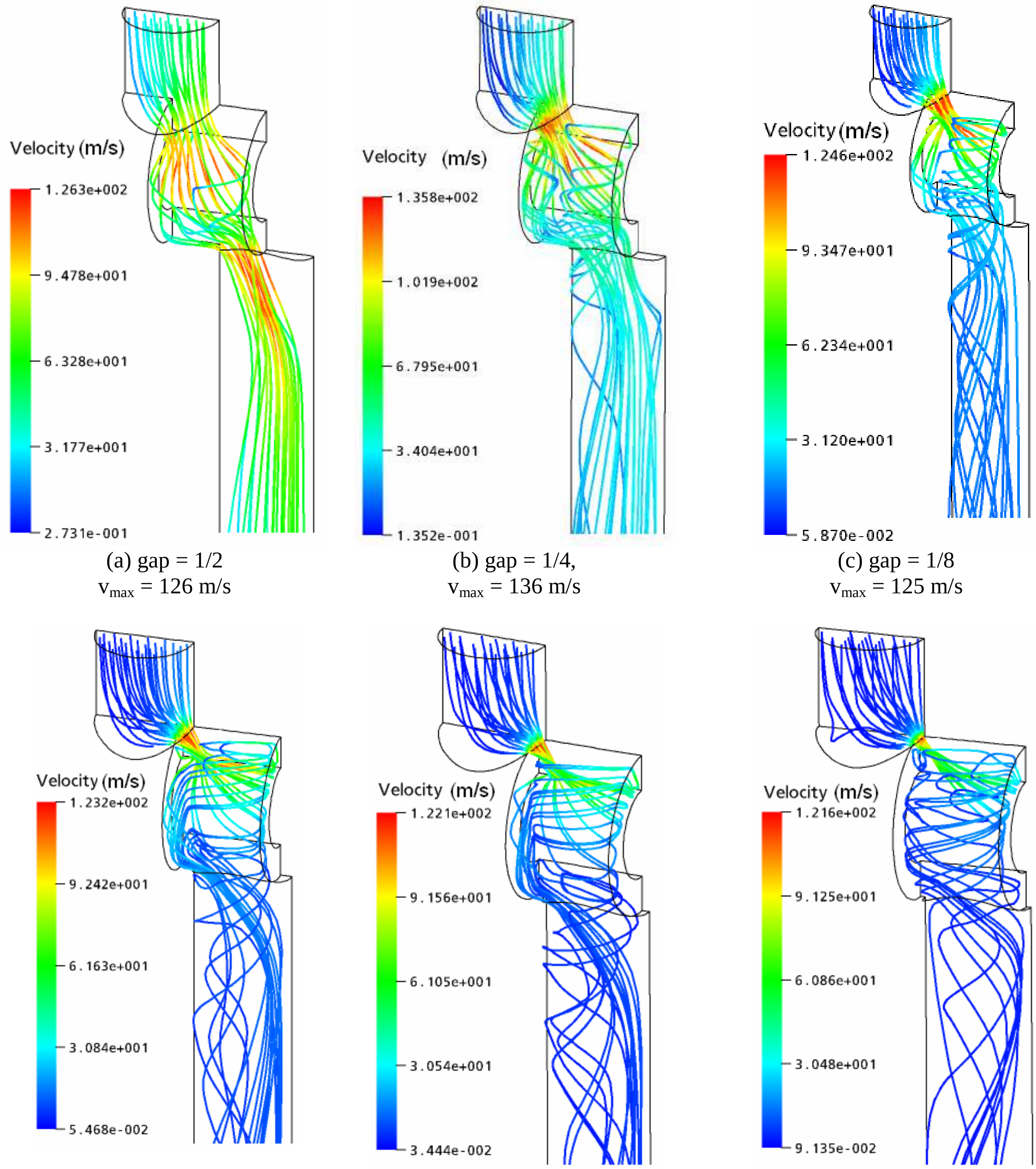
Gaps	Velocity (m/s)	Acceleration (m/s^2)
1/2	0.376991	0
1/4	0.344860	-19.1379
1/8	0.291323	-30.0683
1/16	0.238986	-36.6387
1/32	0.193305	-40.6722
1/64	0.155165	-43.1753

As for boundary conditions, a uniform pressure equal to 6 bar is applied to the entry section of the supply port. On the wall of the valve bore, the no slip condition for velocity is applied. At the exit section, the choice of boundary condition is quite subtle. In this study, a uniform pressure equal to 5.4 bar is applied at the exit. That is, we considered that pressure loss across the valve is 0.6 bar. This is commonly accepted pressure loss in the design of directional pneumatic valve.

3. RESULTS AND DISCUSSIONS

3.1 Velocity and Streamlines

Streamlines coloured with velocity are shown in Fig. 4. These indicate the complexity of the flow pattern inside the valve especially when gaps are small. The maximum velocity is about 120 m/s in all the six cases and is located at the gaps where air enters into the chamber. Large vortexes are developed in the chamber. For small gaps, 1/32 or 1/64 for instance, most air particles have experienced several zigzags before reaching the chamber exit.



(a) gap = 1/2, v_{max} = 126 m/s

(b) gap = 1/4, v_{max} = 136 m/s

(c) gap = 1/8, v_{max} = 125 m/s

(d) gap = 1/16, v_{max} = 123 m/s

(e) gap = 1/32, v_{max} = 122 m/s

(f) gap = 1/64, v_{max} = 122 m/s

Figure 4. Streamlines coloured with velocity (for the sake of space, the exit cylinders are only partially represented.)

3.2 Aerodynamic spool valve force

Obviously, spool valves are not good conduct of air flow in the view of aerodynamics. But they are good in the view of fluid power control because the valve shown in Fig. 1, like most spool valves, is statically balanced. That is, in the chamber, the modulated pressure acts on two equal areas in opposite directions

This does not completely eliminate the aerodynamic force, however, because the momentum of the gas changes as it passes through the chamber. Andersen (1967) stated that if the chamber was in *rectangular* form, the aerodynamic force would tend to close the valve and would rise for small strokes and then become virtually constant. Because of the variation of aerodynamic force with chamber shape, general empirical data are not available. Lee and Blackburn (1952) demonstrated that the maximum aerodynamic force can be reduced to about 10 percent of its ordinary value by proper shaping of the spool chamber. Hence, CFD may play an important role in the optimal design of pneumatic valves as it makes it easy to compute the aerodynamic force.

Figure 5 shows aerodynamic forces acting the spool surface against time steps obtained with CFX Postprocessor. The abscise represents time steps of which the unity is 0.00001 second. As much as fifty time steps corresponding to the total time of 0.0005 seconds were performed for each mesh as long as there was no element had negative volume in the course of the spool surface motion. The vertical axe represents the aerodynamic force acting on the spool surface.

It can be seen that the maximum magnitude of the aerodynamic force is about 0.2 N. All curves, except that of “1/2”, tend to some constant values as the number of time steps is sufficiently large. This agrees with what Andersen (1967) stated for the case of rectangular chamber as we mentioned earlier. But there is difference: these curves do not tend to a unique constant value; instead, each curve tends to a different constant value. The curve “1/4” tends to 0.16 N, the curve “1/8” tends to 0.12N, the curve “1/16” tends to 0.045N and the curve “1/32” tends to 0.019N. Based on these, it seems that the aerodynamic force decreases along with the gap decreases. But this tendency is reversed when the gap is further reduced. One can see that the curve “1/64” tends to 0.04 N which is greater than that of the curve “1/32”.

In Fig. 6a, the last values of the six curves in Fig.5 are drawn in function of the ratio of its corresponding stroke over the radius of the air supply port. It can be seen clearly that the aerodynamic force does not follow Andersen’s observation which was for the case where the chamber was rectangular. In our case, along with the movement of spool starting from “1/2” position rightwards, the aerodynamic force on the spool surface first increases from a negative value (resisting the spool to be closed) to a maximum positive value (dragging the spool rightwards) at “1/4” gap. Then, it decreases until “1/32” gap. After that, it increases again to “help” the valve to be closed. This complicate behaviour of the aerodynamic force for the directional proportional pneumatic valve has never been reported in literatures. It constitutes the main original contribution of the current investigation.

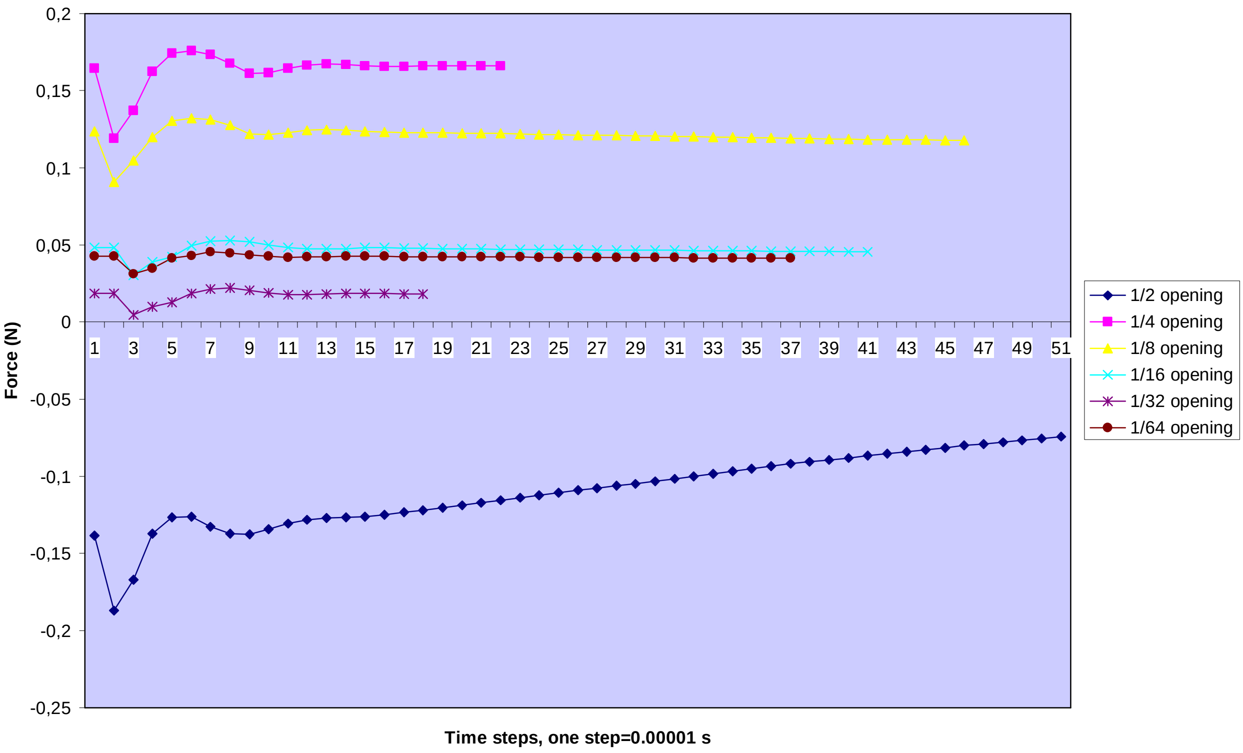
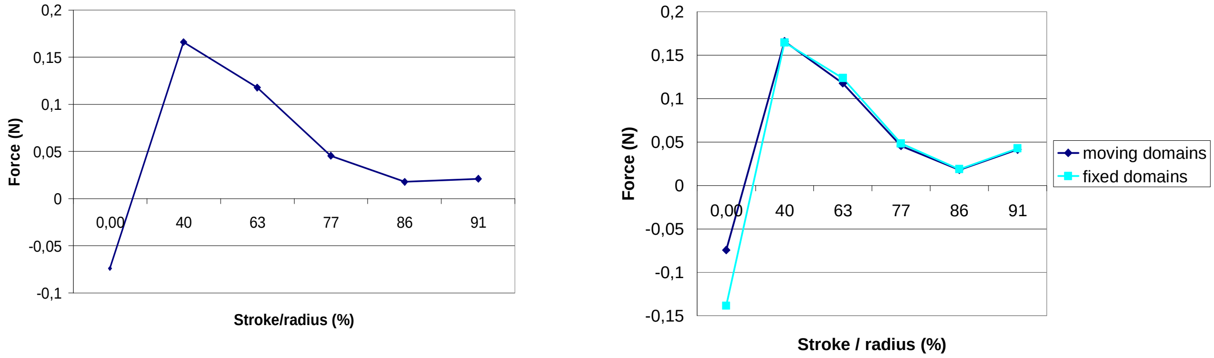


Figure 5. Aerodynamic spool valve force vs. time steps



a. Computed with moving domains b. Computed with moving and fixed domains

Figure 6. Aerodynamic spool valve force vs. stroke

In Fig.6b, results in Fig.6a which were computed with the moving spool are compared with those computed on fixed domains. The essential difference in view of physics is that the spool acceleration was taken into account in the former case while it is neglected in the later case. Results do not show much difference except for the position where the spool begins to slide rightwards. It could be then concluded that the spool acceleration might be neglected in computation of the aerodynamic force. However, we should remain conservative in face of such “conclusion”. Remember that, in this study, the spool did not slide continuously over the whole stroke. Instead, six particular positions of stroke from which the spool moved very slightly were simulated. This is an approximation based on an intuition of physics. That is, the reaction in term of forces happens instantly when applying the acceleration. We need to prove its validity by the experimentation on the valve or by using other independent CFD software which has capacity of dealing with the mesh motion such as Fluent for instance.

4. CONCLUSIONS

Three-dimensional air flow through a pneumatic proportional directional spool valve has been investigated with CFD method. To take into account the influence of spool movement, we have built and analyzed six finite element models with respective valve gaps equal to 1/2, 1/4, 1/8, 1/16, 1/32 and 1/64 of the supply port opening. Numerical results have shown that there are large vortexes around the spool so that the flow in the valve is very complex.

Compared with our earlier study (Jiang, 2006), one of improvements in the current work consists in increasing the length of the exit cylinder to be ten times its radius. This avoids perturbations of the vortex just below the chamber exit.

The most significant improvement of the current study is that by using the mesh motion technique in CFD, the spool movement in a sinus function of time has been simulated. This made it possible to take into account the effect of the spool acceleration, hence, the effect of the inertial force of fluid acting on the spool surface.

To avoid too much element distortions due to the spool movement, we computed first solutions using fixed domains corresponding to the six position of valve as if the valve had already moved to that position. Then we started the spool movement according to its sinus function of time. Although the absolute time with the spool movement that we simulated seems to be small, it is enough to study the importance of the spool acceleration on the aerodynamic force. From the well known Newton’s second law in physics, reactions in term of forces happen instantly when applying accelerations.

With the use of CFD technique, this work has demonstrated that it is possible to examine and predict flow characteristics of a pneumatic valve, especially to compute the aerodynamic spool valve force. From the view of fluid power control, this is essential information for an optimal design of pneumatic proportional directional valve.

5. REFERENCES

Andersen B.W., 1967, “The Analysis and Design of Pneumatic System”, Jonh Wiley & Sons, Inc.

Bao, M., Fu, X., Chen, Y, 2001, “Computational Fluid Dynamics Approach to Pressure Loss Analysis of Hydraulic Spool Valve”, Proceedings of the Fifth International Conference on Fluid Power Transmission and Control.

Gouveia M.G., 1996, “Válvula Pneumática Proporcional de Quatro Vias”, Dissertação de Mestrado, Universidade de São Paulo.

Jiang, Y., Valdiero, A.C., Andrighetto P. L., Wang Chong, 2006, “Analyse and Numerical Simulation of A Directional Proportional Pneumatic Valve”, Proceedings of IV Congresso Nacional de Engenharia Mecânica, 22 a 25 de Agosto 2006, Recife-PE, Brasil.

Lee S.Y., Blackburn J.F., 1952, “Steady state axial forces on control valve pistons”, Trans ASME, 74, 1005 (1952).

Stringer J., 1976, “Hydraulic System Analysis – An Introduction”, Appendix A, Spool Valve Stroking Forces, p.135-136, Thomson Press Limited, New Delhi.

Versteeg H. K. & Malalasekera W, 1995, “An Introduction to Computational Fluid Dynamics-The Finite Element Volume Method”, Longman scientific & technical.

Wang, T., Cai, M., Kawashima K. and Kagawa T., 2005, “Modelling of a Nozzle-Flapper Type Pneumatic Servo Valve Including the Influence of Flow Force”, International Journal of Fluid Power 6 (2005) No. 3 pp. 33-43.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.