# Investigation Of The Flow Field Inside Fluid Power Directional Control Valves By Means Of Particle Image Velocimetry

G. Del Vescovo [1], A. Lippolis [2] , S. Camporeale[1]

[1] Dipartimento di Ingegneria Meccanica e Gestionale,
POLITECNICO di BARI , Via Re David 200 - 70125 Bari - Italy
camporeale@poliba.it,g.delvescovo@poliba.it,

[2] Dipartimento di Ingegneria dell'Ambiente e per lo Sviluppo Sostenibile,
POLITECNICO di BARI - V.le del Turismo, 8 74100 Taranto - Italy
lippolis@poliba.it

## ABSTRACT

The analysis of the flow field inside hydraulic valves has become in the last few years a very important issue to be addressed in order to optimize the spool shapes and to obtain the desired flow metering characteristics.

The Fluid-Power research unit of the Polytechnic of Bari realized many works dealing with the Computational Fluid Dynamics (CFD) analysis of directional control valves.

Experimental measurements concerning discharged flow rate and needed driving force values have been performed in order to validate the numerical results, but the validation of CFD results has been limited, in the past, only to the global fluid dynamic response of the valve and not to the local flow conditions.

In order to provide the flow field inside the valve through PIV (Particle Image Velocimetry) measurements, a valve model, equipped with a great optical access, has been realized. The scaled model has been realized according to the fluid dynamic similarity laws in order to use a fluid different from the oil (water) and to increase the dimensions of the valve. In particular, the realized model has a circumferential development useful to acquire important data about the circumferential flows and their effects on the global flow rate crossing the valve. The experimental results show that the velocity profiles in the metering sections are strongly influenced by the three-dimensional effects of the flow, confirming the results obtained numerically in previous works.

## 1 INTRODUCTION

The directional valves are widely used in Fluid Power systems. For this reason they have been studied in many works and researches. In particular, very often the attention has been focused on the study dealing with the forces needed to move the spool. The flow force values are strictly related to the fluid dynamic conditions, in particular to the efflux angles in the metering sections and are strongly affected by the three-dimensional behaviour of the flows inside the valves.

In the last years, many works have been focused on the study of the fluid dynamic conditions in the hydraulic valves by using CFD methods. In the past, the effects of the spool geometry [1,3] , the effects of the circumferential flows [2] and the flow forces in the open centre directional valves [5] have been investigated.

In these numerical works the local velocity and pressure fields have been analyzed, but only the global results, as far as the flow rate and the flow force values regard, have been validated experimentally [1,5]. In fact, the experimental study of the local flow conditions inside the valve is very difficult, because of the small geometrical dimensions of the valves. Moreover it is very difficult to create an optical access for the internal volumes.

For the last reasons, in order to make use of the modern laser anemometry techniques such as PIV (Particle image velocimetry) or LDA (Laser Doppler anemometry), a scaled model of the original valve has been realized.

Infact the anemometric techniques are not invasive (if the seeding is correct) but they need a proper optical access. In literature a work [4] using the PIV in the study of the flow inside the valve is already present but it uses a valve model with a linear development. This approach is unable to provide information about the effects of the circumferential flows.

In this work, instead, a study of a three-dimensional valve with a circumferential development will be presented. Moreover the model has been realised using the Reynolds and Euler similitudes, in order to use a bigger model with a liquid different from the oil. In this way the efflux conditions, the shape of the jet flow and the turbulence characteristics in the model, if opportunely scaled, can be used to analyze the actual oil flow in the valve.

## 2 HYDRAULIC VALVES AND FLOW FORCES

An example of a hydraulic directional control valve is shown in Fig. 1a while Fig. 1b shows the unbalance ($F_2$<$F_1$) of axial forces originating the flow force in the spool opening phase.

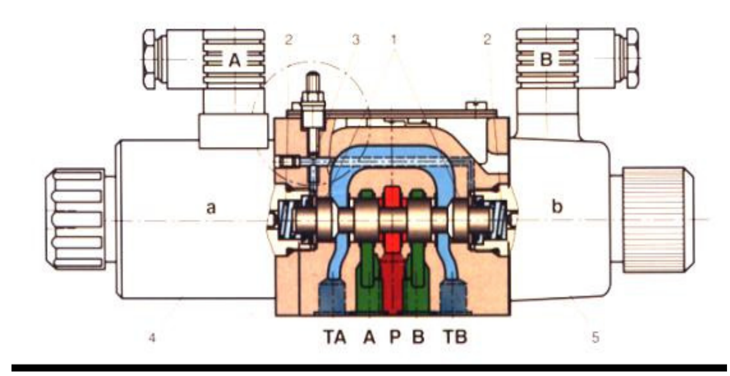


Fig.1a: An example of hydraulic directional control valve

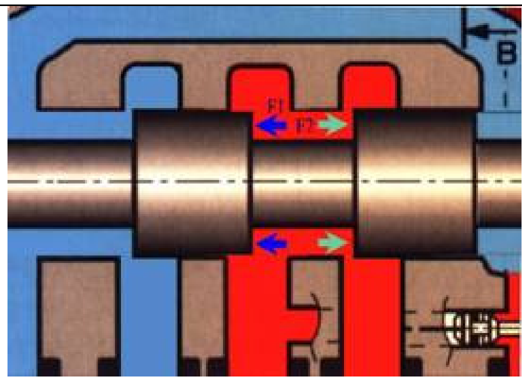


Fig. 1b: the axial forces acting on the spool

It can be shown starting from the application of the momentum balance equation that the flow force has two different expressions in the two different phases of the valve opening.
With the hypothesis of constant discharge coefficient it can be obtained that in the first phase :

$$F_{flow} \propto x\Delta p \tag{1}$$

where x is the axial spool opening while $\Delta$p is the pressure drop enforced on the valve.
In this phase, being the pressure drop constant, the flow force increases linearly with the spool opening. When the flow rate crossing the valve reaches the pump flow rate the pressure drop decreases and the flow rate is constant. There are two different constrasting effects that are the increasing of the valve opening and the decreasing pressure drop. In this case the following relation can be found:

$$F_{flow} \propto \frac{1}{x} \tag{2}$$

It is clear that the flow force that the driving system must overcome has a maximum value located between the two phases.
In this brief and approximate outline, strong hypotheses have been assumed; in fact the previous relations 1 and 2 can be reached considering that the flow is not affected by the fluid dynamic structures in the valve while experimental and numerical three-dimensional analyses [1,2] have shown (see Fig.2) that the flow force trend in the first phase is not linear. This is due to the strong influence that the outflow angle, the boundary layer, the pressure recovery, the discharge coefficient, the three dimensional effects have on the driving forces.

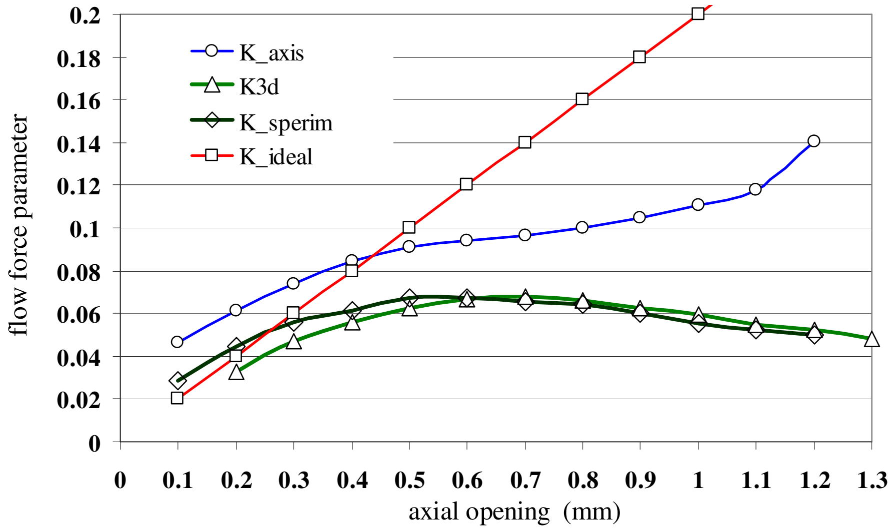


Fig.2: flow force parameters in the axis-symmetric simulation, 3D simulation, experimental measurements and theoretical estimation

From this observation it is obvious that CFD analysis is a proper tool to study the last effects and in the last years has become very widespread even in the industrial applications.
The driving forces depend strongly on the spool and body valve shapes because the fluid dynamic structures inside the valves are strongly affected by macro and microgeometrical features of the valve.
CFD can be considered a proper tool to design a valve avoiding experimental campaigns but many problems related to its applicability must be solved, first of all a local validation could be a starting point to set proper turbulence model parameters.

## 3 FLUID DYNAMIC SIMILARITY

The dynamic similarity is obtained if the ratio between forces of the same type in the actual and in the scaled model remains constant at all the different corresponding points of the two models. Thus, the dynamic similarity is necessary in order to obtain correct information dealing with the forces in a scaled model.
It is possible to show that in geometrically similar models the equality of the Reynolds and Euler numbers is the necessary condition in order to fulfill the dynamic similarity between two incompressible flows.
The Reynolds and Euler numbers of the two models can be expressed as follows:

$$\mathrm{Re}_{H_2O} = \frac{\rho_{H_2O} V_{H_2O} L_{H_2O}}{\mu_{H_2O}} \qquad \mathrm{Re}_{oil} = \frac{\rho_{oil} V_{oil} L_{oil}}{\mu_{oil}} \tag{3}$$

$$Eu_{H_2O} = \frac{\Delta p_{H_2O}}{\rho_{H_2O} V_{H_2O}^2} \qquad Eu_{oil} = \frac{\Delta p_{oil}}{\rho_{oil} V_{oil}^2} \tag{4}$$

where the velocity values in the metering sections have been considered.
In the water model and in the oil model the Reynolds and the Euler numbers must be equal. Combining the numbers it is possible to evaluate the pressure drop in the water model that is needed to obtain the similarity:

$$\Delta p_{H_2O} = \Delta p_{oil} \left( \frac{\rho_{oil}}{\rho_{H_2O}} \right) \left( \frac{L_{oil}}{L_{H_2O}} \right)^2 \left( \frac{\mu_{H_2O}}{\mu_{oil}} \right)^2 \tag{5}$$

The following values of the geometrical scale and of the typical oil and water characteristics can be considered:

$$\left(\frac{L_{H_2O}}{L_{oil}}\right)=5 \tag{6}$$

$$\rho_{oil}=800Kg/m^3,\ \rho_{H_2O}=1000Kg/m^3$$
$$\mu_{oil}=0.03Kg/ms,\ \mu_{H_2O}=0.001Kg/ms$$

From Eq. 5 if a typical hydraulic valve pressure drop is considered, the corresponding value of the pressure drop on the scaled model can be computed. In particular for $\Delta p_{oil}$=200 bar, in the water model the pressure drop $\Delta p_{H2O}$ equals 711 Pa. From these values the dynamic similarity can be obtained with very low pressure drop values on the scaled model; considering a global discharge coefficient equal to 0.4 [2] it is possible to show that the water velocity in the metering section equals approximately 0.5 m/s.
The modelling of the flow jet dissipation downstream of the metering section is the most important phenomenon to study. Considering that the water is less viscous than the oil and the water model is 5 times bigger than the oil-model, the viscous dissipation acts on the water jet in the same way it acts on the oil-jet if the water velocities values are much lower than the oil models; an oil-jet with about 200 m/s velocity value is in fluid dynamic similarity with a water-jet with a velocity of about 0.5 m/s.
From the previous relations it is evident that the pressure losses inside the valve model are very low if compared to the pressure losses occurring in the hoses (this is the opposite situation of directional fluid power valves where pressure losses in the lines are negligible if compared to pressure losses caused by the metering sections); in fact, in the valve model test rig, flow rate doesn't change with the valve opening.
In order to realize fluid dynamic conditions that could reproduce the first opening phase conditions (see Equ.1), a pressure drop measurement should be needed but a pressure tranducer, able to appreciate the foreseen very low pressure values is not , at the moment, available in the laboratory.
For this reasons starting from the orifice equation:

$$Q=C_d\pi Dx\sqrt{\frac{2\Delta p}{\rho}} \tag{7}$$

and assuming that the discharge coefficient is constant with the spool opening, the flow rate crossing the valve should be increased using a linear proportionality between the flow rate and the spool opening. In this way the analysis of the flows inside the valve in the first opening phase, that is the most interesting phase, could be possible although the last procedure is not rigourously valid.

## 4 VALVE MODEL DESIGN AND TEST RIG

The following figure shows a section of the valve model: the element 1 is the spool realised in polyethylene (black). The elements 2 are transparent disks in polycarbonate. The pipe connection 3 is the INLET section while the pipe connection 4 is the OUTLET section. The element 5 is a transparent tube in methacrylate. The elements 6 are the gaskets needed to reduce leakages between the spool and the disks. Figure 4 shows a picture of the scaled model.

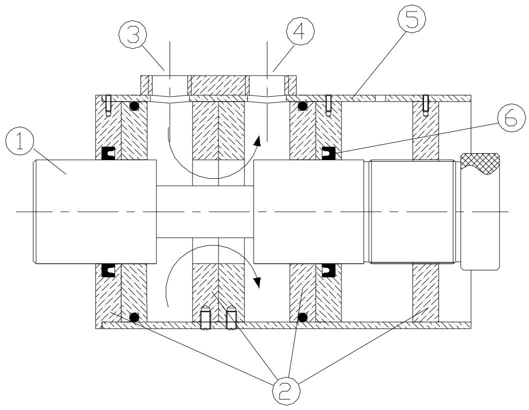


Fig.4 The model valve

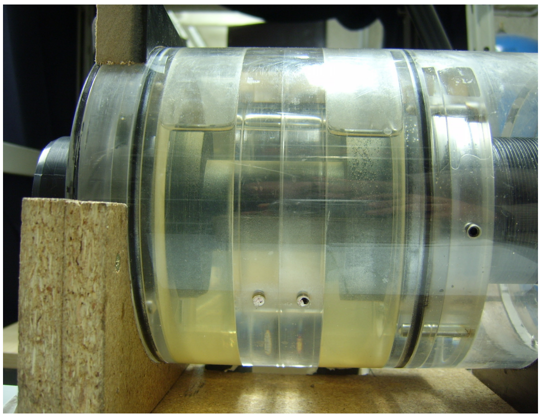

Fig.5 Valve model picture

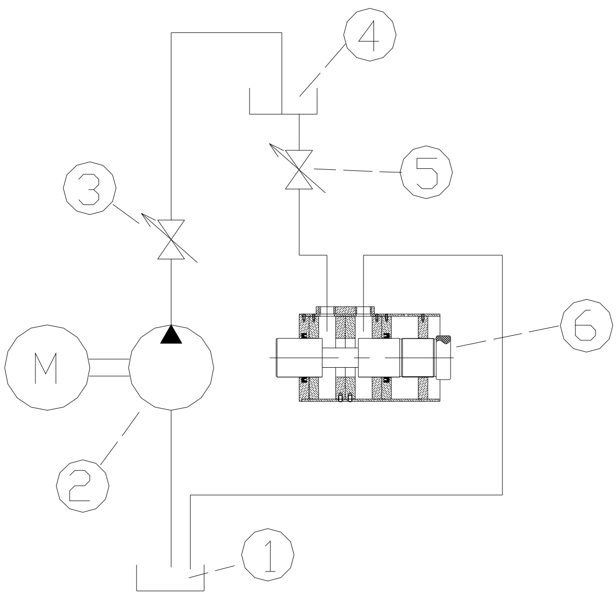


Fig.6 The test Rig

Fig.6 shows the realized test rig. A sixty litres tank (1) is located on the ground. The element (2) is a centrifugal pump needed to the recirculation of the water.
The pressure drop needed to obtain the flow inside the valve is provided by a second tank (4) located 2 meters higher than the first. Two valves (5,3) allow to calibrate the flow rate inside the model valve. Fig.7 shows an image of the laser head in front of the model valve and three different angular positions of the camera.
The tests have been performed by means of a pulsed Gemini 2000 Nd:YAG laser (Neodymium Yttrium Aluminum Garnet) while the image processing has been realized by a LaVision™ software system.

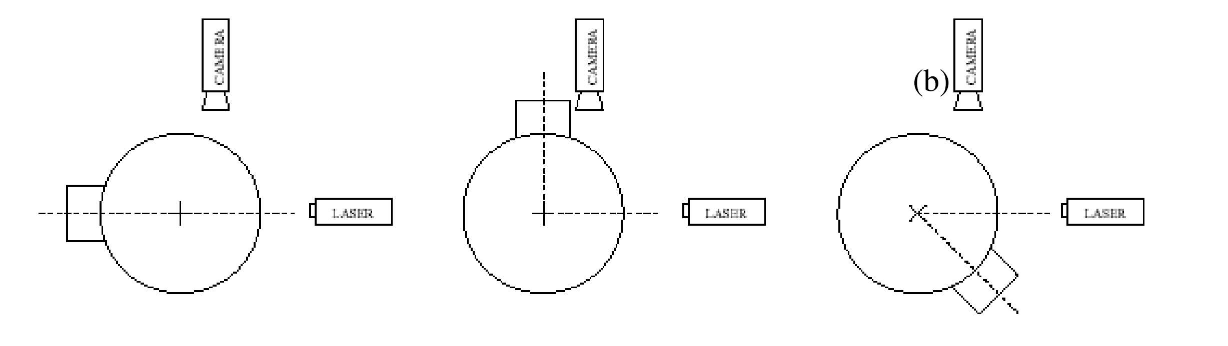


Fig. 7 PIV instrumentation at three different angular positions

## 5 EXPERIMENTAL RESULTS

The initial tests have been performed at different spool displacements. The used seeding particles are silver powder, the temporal step between frames is 200 ms and the laser power was the 80% of the maximum value. The obtained images are the average fluid dynamic fields of 100 vector fields acquired at a frequency of 15 Hz.
The following images show the velocity profiles on a meridian plane located at 180° angle value with respect to the inlet and outlet sections and with different spool displacements.
The test was performed with a spool opening of 14 mm that corresponds to a 2.8 mm axial opening of the actual valve.
The jet is oblique and the flow is not attached to the wall (A). On the two side of the efflux jet two recirculation zones are clearly visible (B,C).

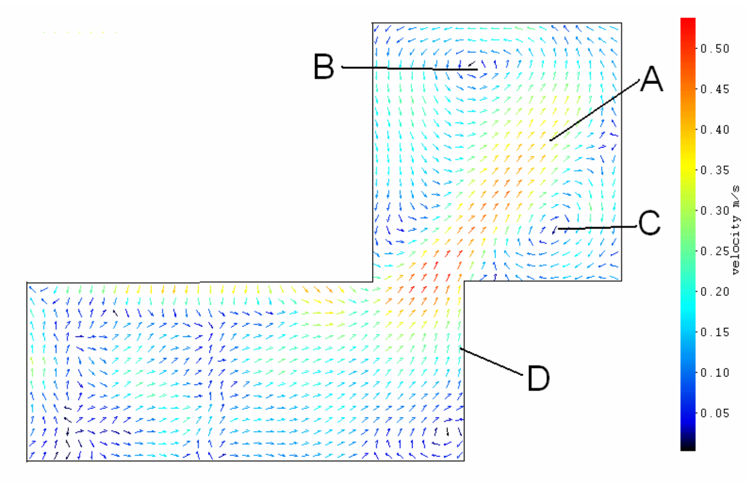


Fig. 8 PIV Velocity vectors on a 180° plane ( 14 mm spool opening, 1 l/s flow rate)

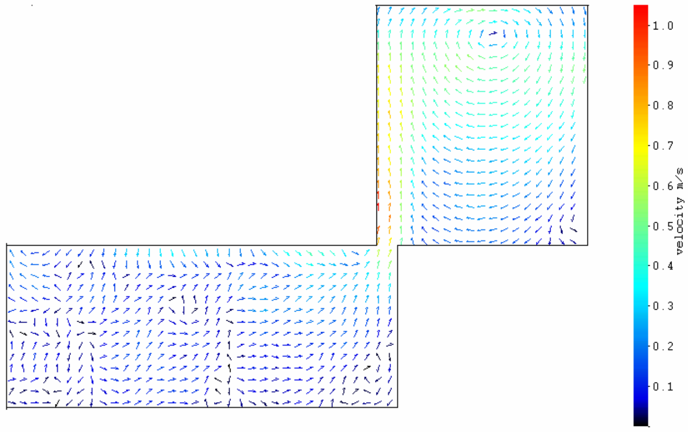


Fig. 9 Velocity vectors on a 180° plane (4 mm spool opening, 1 l/s flow rate)

A great acceleration of the flow in the restricted section can be noticed and the kinetic energy dissipation is also visible when the flow enters the discharge chamber.
Moreover, the acceleration near the wall of the spool (D) is visible. This phenomenon produces a decreasing profile of the pressure distribution on the spool face and it is at the origin of the flow forces [2].
The velocity profiles for different spool openings are reported in Figs. 8–12. The angular position of the valve is constant with an angle of 180°.
When the axial opening is small, the flow entering the discharge chamber remains attached to the wall ("Coanda effect"). When the axial opening increases the flow detaches from the wall.
These tests have been performed with a constant flow rate and not at a constant pressure drop enforced on the valve.

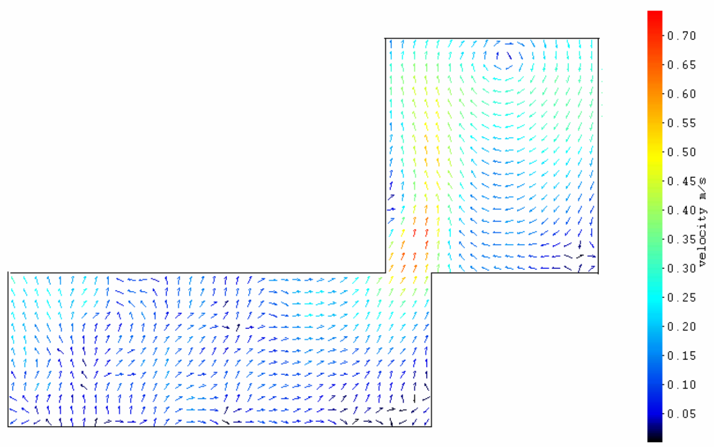


Fig. 10 Velocity vectors on a 180° plane (8 mm spool opening, 1 l/s flow rate)

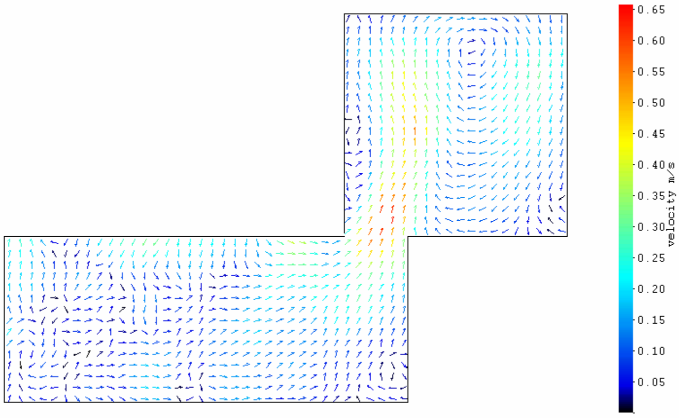


Fig. 11  Velocity vectors on a 180° plane (10  mm spool opening, 1 l/s flow rate)

The experimental tests have been realized at the same valve opening (12 mm) but at different angular positions (45°, 90° and 180°).
The flow rate for these tests was equal to 0.9 l/s.  This case corresponds to an opening of 2.4 mm in the actual valve.

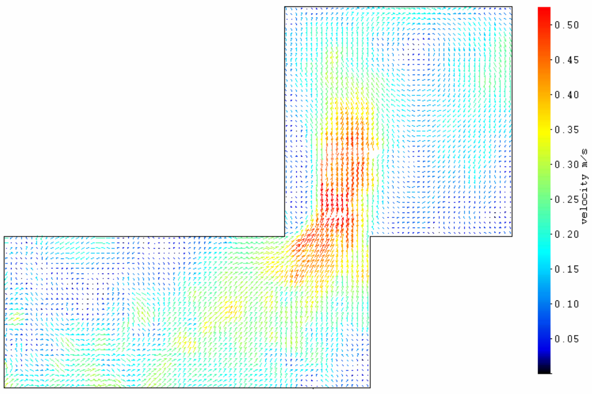


Fig. 12  Velocity vectors on a 180° plane (12  mm spool opening, 0.91 l/s flow rate)

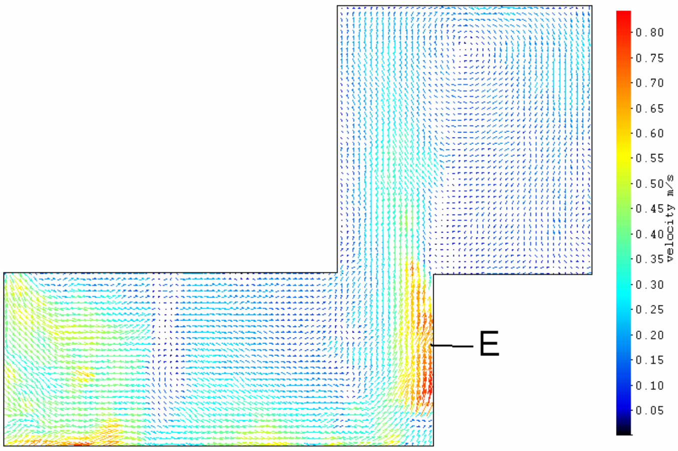


Fig. 13  Velocity vectors on a 90° plane (12  mm spool opening, 0.91 l/s flow rate)

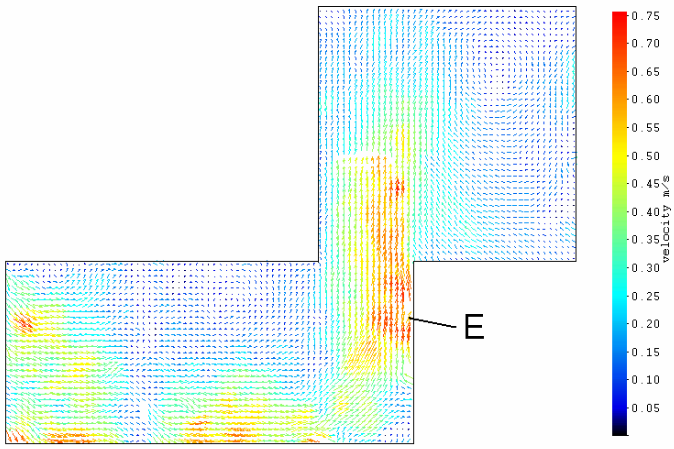


Fig. 14  Velocity vectors on a 45° plane (12  mm spool opening, 0.91 l/s flow rate)

The deep differences between the velocity profiles in the meridian planes show that the flow is highly three-dimensional and is very far from the axis-symmetric conditions. In particular the 45° and 90° meridian planes show an acceleration zone (E) attached to the spool wall while the 180° plane shows a distributed acceleration along the whole metering section and in this case the experimental results seem to disagree with the previous numerical results.
The reason of this difference has been found in the small inlet and outlet sections of the valve model that,  because of design constraints, are constituted by two water connections of one inch diameter each one.
It can be argued that for large valve openings, the inlet and outlet sections and the circumferential metering section have comparable hydraulic diameter values.
The last behaviour doesn't correspond to the actual conditions of a hydraulic valve where the metering section is the zone of the valve where the great part of the pressure drop must occur.
For this reason the high velocities in the inlet section cause a distortion in the velocity profiles. In order to validate this hypothesis two numerical simulations have been realised. In the first simulation, the inlet and outlet sections were equal to the sections of the model. In the second simulation, two larger sections, more similar to the commercial valve, were used.
The results have not been reported for the sake of brevity but they validate the previous hypothesis.
The PIV analysis has not allowed to estimate the outflow angle and the boundary layer effects in the metering section because the interrogation windows are not sufficiently small and the camera is not able to focus a smaller region. In fact the used interrogation window is 32 by 32 pixels that corresponds to  2.24mm by 2.24mm. If this values are scaled in the actual valve we obtain  0.448 mm by 0.448 mm that if compared to the typical values of the metering section that are around 1mm - 2 mm corresponds to a very large cell.

## 6 CONCLUSIONS

The work shows that the realized valve model is suitable for PIV investigations and can be considered a good test bench for future research activities.
The instrumentations and the used seeding particles are optimal for the study of the velocity profiles. The obtained results show qualitatively that the velocity profiles in the metering sections are strongly influenced by the three-dimensional effects confirming the numerical results obtained in previous works.
In fact the circumferential wall that limits the discharge chamber produces recirculation zones on the two lateral sides of the jet-flow and a strong diversification of the flow structures in the different meridian planes. In the future the research will be focused on a more accurate and quantitative analysis of the differences between the meridian planes and on the measurement of the tangential components of the velocities.

The valve model will be modified, in particular bigger inlet and outlet sections will be realized and accurate pressure transducers will be used in order to keep constant the pressure drop on the valve model. Moreover a lateral optical access will be realized in order to make PIV measurements in the cross sections.

## 7 ACKNOLEDGEMENTS

The authors are in debt to Prof. Bernardo Fortunato for the use of experimental equipment and to Eng. Marco Camporeale for the experimental measurements.

## 8 LIST OF NOTATIONS

| | |
|---|---|
| $F_{flow}$ | flow force |
| $x$ | axial spool opening |
| Q | flow rate |
| $C_d$ | discharge coefficient |
| D | spool external diameter |
| $v_{H2O}$ | water characteristic velocity |
| $v_{oil}$ | oil characteristic velocity |
| $Re_{H20}$ | water Reynolds number |
| $Re_{oil}$ | oil Reynolds number |
| $r_{H2O}$ | water density |
| $r_{oil}$ | oil density |
| $\Delta p_{oil}$ | pressure drop on the hydraulic valve |
| $\Delta p_{H2O}$ | pressure drop on the model |
| $m_{H2O}$ | water viscosity |
| $m_{oil}$ | oil viscosity |
| $L_{H2O}$ | model characteristic length |
| $L_{oil}$ | valve characteristic length |

## 9 REFERENCES

1 Borghi M. , Milani M., and Paoluzzi R., “Stationary axial flow force analysis on compensated spool valves” International Journal of Fluid Power **Vol. 1 No. 1** p.p. 17-25, 2000.

2 Del Vescovo G., Lippolis A., “Three-dimensional analysis of flow forces on directional control valves”, International Journal of Fluid Power, **Vol.4 No 2**, July 2003.

3 Del Vescovo G., Lippolis A., “CFD analysis of flow forces on spool valves”, Proceedings of the 1st International Conference on Computational Methods In Fluid Power Technology, Melbourne November 26-28, 2003.

4 D. Gao, “Investigation of flow structure inside spool valve with FEM and PIV methods”, International Journal of Fluid Power **Vol 5, No 1,** March 2004.

5 G. Del Vescovo, “Experimental and Numerical Analysis of the Flow Forces on an Open-Center Directional Control Valve” *Proceedings of the 3rd FPN International PhD Symposium*, Terrassa, *BARCELLONA SPAIN*, 30 June - 2 July, 2004.

6 LA Vision™, *DaVis Flow Master Hardware, DaVis Flow Master Software*, 2004.

7 A.A. Abdel-Rahman, S.F. Al-Fahed and W. Chakroun, “The near-field characteristics of circular jets at low Reynolds numbers, Mechanics Research Communications”, **Vol. 23 No.3,** 1996.

8 Shames I.H., *Mechanics of Fluids*, McGraw-Hill International Editions, 1992.

9 M. Raffel C. Willert J. Kompenhans, *Particle Image Velocimetry: A Practical Guide*, Third corrected edition, 1998.

10 Huang Dabiri Gharib, “On errors of digital particle image velocimetry”, Technical report, California Institute of Technology, 1997.

11 S. T. Wereley C. D. Meinhart, “Second-order accurate particle image velocimetry”, Technical report, Department of Mechanical and Environmental Engineering of University of California, 2000.

12 John C. Tannehill Dale A. Anderson Richard H. Pletcher, *Computational fluid mechanics and heat transfer. Computational and Physical Processes in Mechanics and Thermal Sciences*, Taylor & Francis, second edition, 1997.