Solid State Science and Technology, Vol. 19, No2 (2011) 48-60
ISSN 0128-7389

NUMERICAL INVESTIGATION OF PRESSURE DROP IN HYDRAULIC SPOOL VALVE

M.O. Abdalla*, T. Nagarajan and F.M. Hashim

Mechanical Engineering Department, Universiti Teknologi PETRONAS, 31750 Tronoh, Perak, Malaysia

** Corresponding author: mohamed.osman1@yahoo.com*

ABSTRACT

Pressure drop normally occurs within spool valves in the power hydraulic systems. The complexities of the valve geometry make it complicated to calculate the pressure loss analytically. The flow characteristics inside a hydraulic spool valve were investigated in this study using FLUENT software. The relation between mass flow rate and pressure drop across the spool valve was studied by the simulation of the flow using different mass flow rates. Results of the study showed that, the pressure drop increases rapidly as the mass flow rate increases. The relation between the valve geometry and the pressure drop was studied as well. Different geometrical designs were proposed for both the valve chamber and the spool. Simulation of flow in the proposed valves was carried out. Results show that the pressure drops is highly affected by the valve chamber geometry and not by the spool geometry. It was noticed that the shape of outlet port of the valve chamber has the most significant effect on the flow behavior of the fluid, and hence on the pressure drop across the valve. The study concluded that, the modification of the outlet area of the valve chamber is the most effective way to reduce the pressure drop.

Keywords: Computational Hydraulic; Hydraulic Pressure drop; Hydraulic Transmission; Spool Valve;

INTRODUCTION

Overview

The common problem encountered in power hydraulics is the unnecessary pressure drop through the entire system. Spool valve is the one of the parts that causes a lot of pressure drop in power transmission and control systems. This pressure loss significantly affects the tracking and positioning accuracy of the hydraulics system. It also increases the operating cost of the system, since any drop in pressure should be compensated by additional power input. By improving the efficiency of the valves it will be possible to impart great benefits to all related applications. The complexities of the valve geometry make it complicated to calculate the pressure loss analytically. The pressure drop in valves is normally calculated by empirical formulas. The Computational Fluid Dynamics (CFD) software makes it easy to find out and attain a better solution of the pressure drop within the intricate channels of various hydraulic components. Numerical simulation provides essential indications during the preliminary phase of the design and allows minimizing the time of development and the cost of the subsequent experimental analysis. By using CFD it will be possible to examine and forecast valve characteristics and to optimize the valve geometry before the manufacturing processes start. In this study the CFD simulation was conducted on 3-D models of the hydraulic spool valves. The pressure pattern obtained from the different valve models was discussed.

Literature review

The properties of flow field in a spool valve play an essential role in energy and pressure dissipation. There has been many works concerning the study of the flow in hydraulic valves.

Bella et al. [1] developed a simulation tool for the analysis of the fluid dynamical field in general 2d domains. They proposed a simple and yet powerful method for the approximation of conservation laws. They successfully applied it to three test cases of increasing complexity. One of the test cases is on Butterfly valve simulation. Relatively large pressure gradients were detected near the valve disk edges. Bingang [2] studied the flow field in a poppet valve by using the Boundary Element Method in his research. Stevenson et al. [3] have provided a curve showing the relations between different valve gaps and their pressure drops. The results show the increase of pressure drop with the decrease of the valve gap. Oshima et al. [4-5] studied the pressure distribution and cavitations in a poppet valve using a half cut model of poppet valve. Linxiang et al. [6] used laser induced fluorescence imaging and particle image velocimetry to study the velocity field in a spool valve. Ito [7] used streamline coordinates to calculate the pressure distribution in a poppet. Min et al. [8] used CFD to study the pressure drop in a low pressure spool valve at low mass flow. Bertelli and Pariona [9] studied the flow of the fluid inside the valve. They concluded that the flow had a turbulent behavior for all the valve openings. The turbulence was more notorious for the small opening of the valve. Results of their study showed a linear behavior of the maximum velocity, as well as of the exit velocity of the flow as a function of the entrance speed of the fluid.

Pressure drop calculations

Pressure drop is the decrease in pressures from the inlet of the valve to outlet due to flow restrictions. The oil flow passes through different cross sections from the inlet to the outlet of the valve. Each change in flow cross section causes some pressure drop in the fluid. The oil flow also changes its direction at different locations on its path. This leads to creation of turbulence in the flow field which in turn consumes some energy from the flow. All energy losses are converted to heat.

In the steady state, incompressible , inviscid, laminar flow in a short pipes (no change in elevation) with negligible frictional losses, Bernoulli's equation and continuity equation can be reduced to give the pressure drop at any cross section in the flow path. This relationship is shown in Equation 1.

Solid State Science and Technology, Vol. 19, No2 (2011) 48-60
ISSN 0128-7389

$$\Delta P = \rho\left(\frac{Q}{K}\right)^2 \qquad (1)$$

Where:
ΔP = Pressure drop (at any cross-section change), (kPa).
ρ = Fluid density, (kg/m^3).
Q = Volumetric flow rate (at any cross-section),(m^3/s).
K = Discharge coefficient (at any cross-section), (m^2).

As it seen from equation 1, the pressure drop is a function of the square of the flow rate for a definite fluid. Equation 1 is difficult to be applied directly to the spool valve due to the complexity of the valve structures. Therefore empirical formulas are used to calculate the pressure drop in valves.

COMPUTATIONAL ANALYSIS

The objective of this study is to use a computational method for calculation and optimization of the pressure drop in the spool valve. The computational study has followed the steps as shown in Figure 1 below:

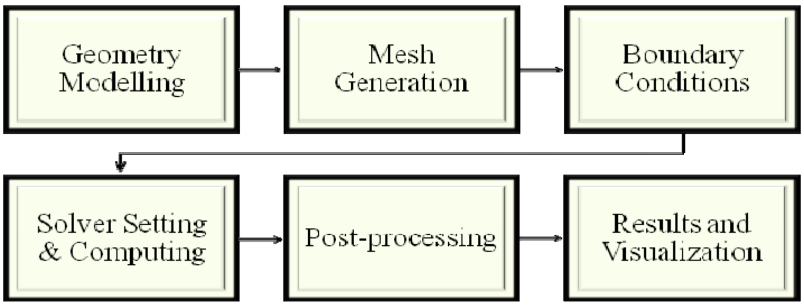
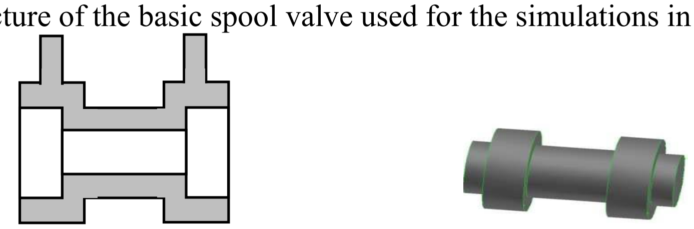


Figure 1: Solution steps

3D Spool Valve model

The geometries of the spool and the valve chamber are very complex. It consists of many stages and hydraulic resistance such as orifices, pipes, etc. Some of these parts are omitted in this study for simplification purpose. The simplification of the valve geometry can help to reduce the difficulty of mesh creation. Figure 2 shows the simplified structure of the basic spool valve used for the simulations in this study.



a- Fluid zone b- Basic spool

Figure 2: Simplified basic spool valve structure

The meshed geometry of the valve fluid zone shown in Figure 3 is created using Gambit software.

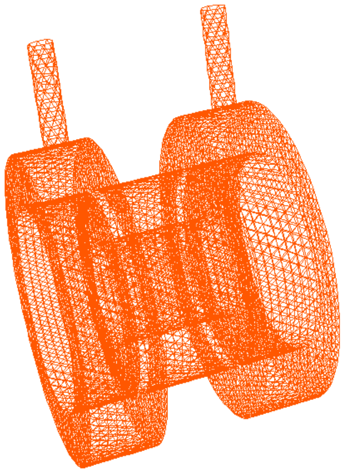


Figure 3: Meshed fluid zone geometry of the spool valve

The whole fluid zone inside the valve was used for the simulation. An appropriate mesh size was selected so as to give accurate and stable results.

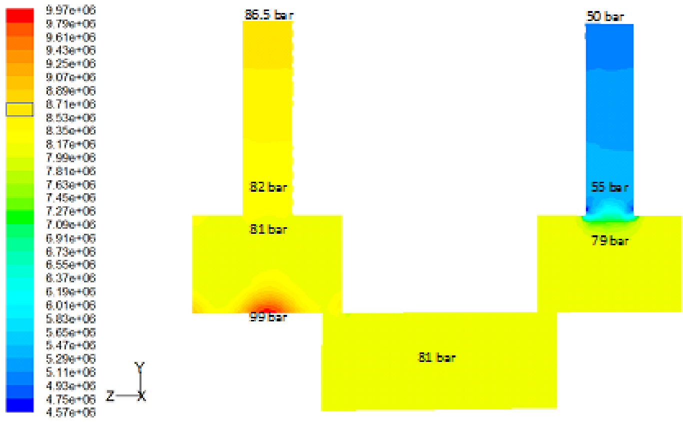
CFD Modeling

Fluent Code was used to perform the CFD calculations presented in this study. The high pressure flow inside the valve is assumed to be completely turbulent. The realizable k-epsilon turbulence model was applied to all the simulations because of its numerical stability under a condition of large pressure gradient. The fluid used was standard hydraulic oil having a viscosity of 46 m^2/s at 40°. Viscosity and density of the oil were all considered to be constant, i.e. the oil is Newtonian and incompressible. A segregated implicit steady state solver was used. Second order equations are selected in the solution control. Convergence criteria of 10^{-6} for velocities, energy and continuity are applied. A no slip velocity was taken as boundary conditions at the wall. A pressure outlet was applied to all the cases of this spool valve.

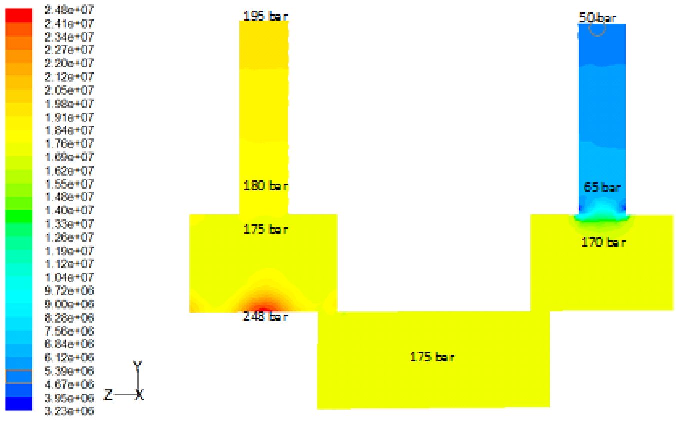
SIMULATIONS OF BASIC GEOMETRY

To analyze the effect of the fluid flow rate on the pressure drop in a spool valve, several valves having different geometries with different mass flow were simulated. Low pressure and low mass flow spool valves were studied by Min et al. [8], while high flow rates and high operating pressures along with different geometry modification were considered in this study. Pressure of 50 bar were employed at the outlet in contrast to the 5 bar used by Min et al. [8].

The basic spool valve shown in Figure 3 was simulated on Fluent code. The mass flow rate was varied over a range of 1~2 kg/s with the increments of 0.125 g/s. The simulation of pressure gave results as shown in Figure 4 and Table 1. These results were presented graphically as in Figure 5.



(a) mass flow rate of 1 kg/s



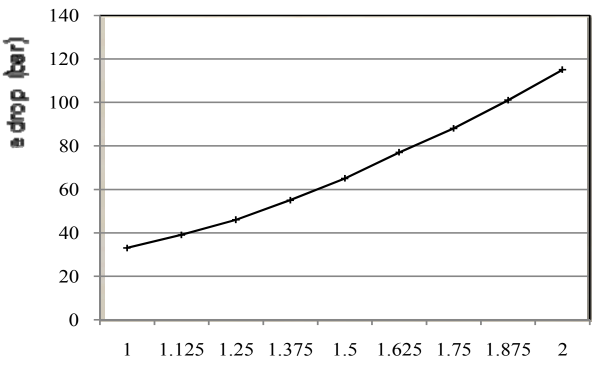
(b) mass flow rate of 2 kg/s

Figure 4: Contours of static pressure for basic spool valve

Table 1: The relation between mass flow rate and pressure drop in the basic spool valve

Mass flow rate (kg/s)	*Inlet pressure (bar)*	*Outlet pressure (bar)*	*Pressure drop, ΔP (bar)*
1.000	87	54	33
1.125	94	55	39
1.250	103	57	46
1.375	113	58	55
1.500	125	60	65
1.625	138	61	77
1.750	150	62	88
1.875	164	63	101
2.000	180	65	115

Solid State Science and Technology, Vol. 19, No2 (2011) 48-60
ISSN 0128-7389



Mass flow rates (kg/s)

Figure 5: Pressure drop vs. mass flow rates

The curve shows how rapid is the pressure drop increases over the mass flow rates which is agreed with the previous studies, especially of Min et al. [8]. Also it agreed with Equation 1 of the laminar flow, where the pressure drop is a function of the square of the flow rate. These results vary with the valve size and the inlet and outlet diameters. But all of them showed the same behavior and trend.

Figure 6 shows a velocity vector at the nozzle-like cross section plane of the fluid inside the valve chamber. It is easy to notice that all the annular space of valve chamber is used by the fluid flow. This is contrast to that of low flow rate carried by Min et al. [8] in which most the fluid flows in the upper part of the chamber.

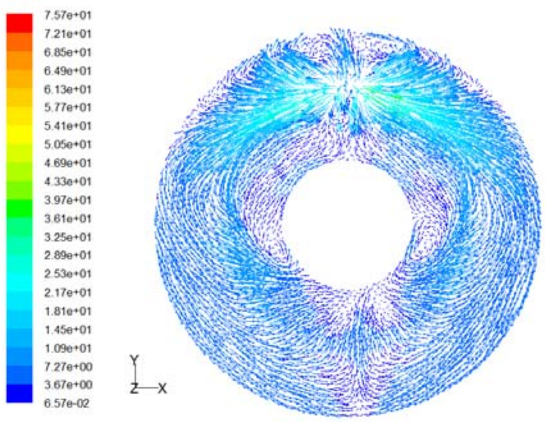


Figure 6: Velocity vectors of one x-cross plane

The flow stream forms two symmetrical vortices on the sides of the spool as shown in Figure 7. This is one sign of existence of turbulence in the fluid. These two vortices diminish into the main middle flow near the outlet area.

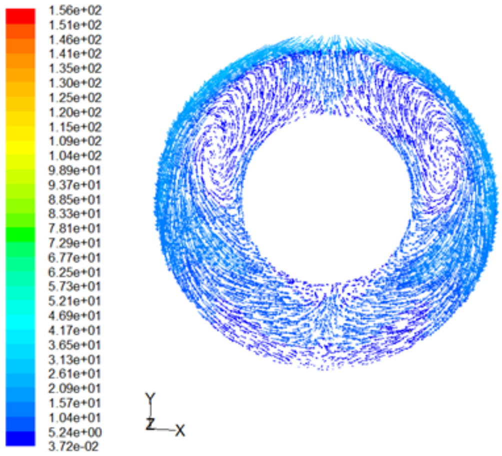


Figure 7: Contours of vorticity magnitude

OPTIMIZATION OF VALVE GEOMETRY

Optimization of the valve required study and measurement of pressure drop in all points along the fluid path between the inlet and the outlet of the valve. Figure 8 show contours of absolute pressure along the plane of symmetry for the basic valve. The fluid flows downward from the upper inlet through the inner part of the chamber and up to the right outlet.

A very high contour gradient was detected at location ‘A’ on the upper side of the spool, facing the inlet indicating a high pressure increase. This increase of pressure is due to stagnation of the fluid at that point. On other hand, the contours of the static pressure show a high pressure gradient at location ‘B’ near the outlet, indicating high pressure drop at that area of the valve. This area of the valve was considered the most critical location that need close attention in the optimization process. The structure of this location needs to be modified to allow easy flow, and hence, to reduce the pressure gradient.

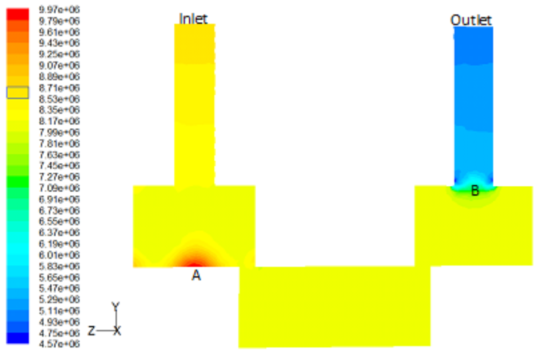


Figure 8: Contours of absolute pressure for the basic valve

Solid State Science and Technology, Vol. 19, No2 (2011) 48-60
ISSN 0128-7389

The velocity vectors for the fluid were illustrated in Figure 9. Flow vortices at different corners are clearly detectable on the vertical central plane of the valve. The inlet stream is divided by the upper spool edge. One part builds a circulation zone near the inlet, the other part flows into the chamber over the spool edge.

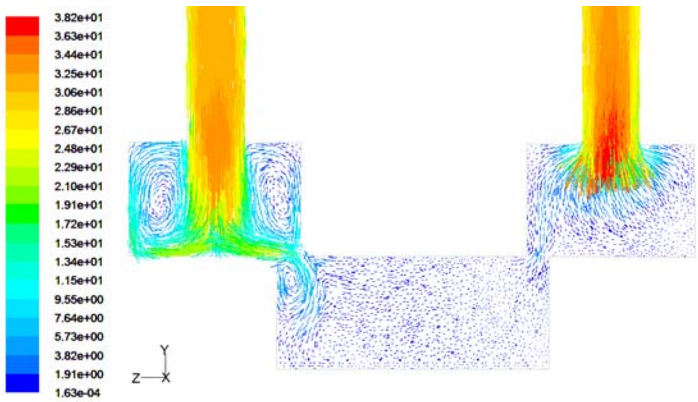
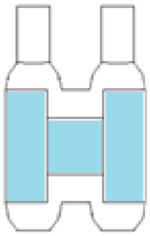


Figure 9: Velocity vectors for 3-D valve model

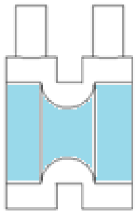
By analyzing the flow fields above of the basic spool valve, it was noticed that, the sudden change of the areas in the flow path causes many problems such as flow stagnation, flow separation, corner vortices and high pressure gradients. These cause undesired pressure drop in the fluid. These results should be taken into account when searching for a solution to energy loss in the spool valves. Any modified valves have to facilitate smooth fluid flow with the least pressure drop.

This study proposed four different valve's geometries as shown in Figure 10. Each of the valves changed one part of its structure (the spool or valve chamber). Each one of the modified valves was simulated separately. The same boundary conditions and mass flow rates were applied to the different modified valves. The pressure drop resulting from the different valves simulation was recorded.

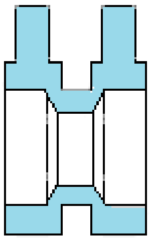
The modifications shown in Figure 10 (a and b) were proposed in a previous research [8], but at low pressures and flow rates. In this study the same modifications along with other two other modifications (c) and (d) had been simulated for high operating pressures and high flow rates.



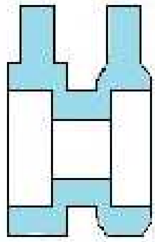
(a) Round Inlet/outlet.



(b) Round edge spool.



(c) Tapered spool.

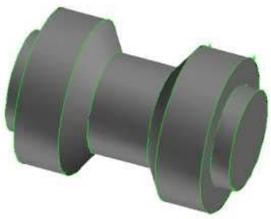


(d) Round/Conical outlet.

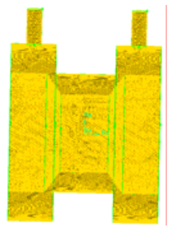
Figure 10: The proposed structures of the valve

Spool modification

The structure of the spool was modified by removing the sharp corners as shown in Figure 10 (b) and (c). Although the spool's structure was changed, the function of the valve remains unaffected. Figure 11 shows the proposed tapered spool together with its meshed fluid zone.



a- Spool valve body



b- Meshed fluid zone.

Figure11: 3D Tapered spool valve and fluid zone

The results of simulation of both valves showed very little improvement regarding the pressure drop. The pressure drop remains similar to that of the basic geometry spool valve. Figure 12 shows the static pressure contour of a modified spool valve.

Solid State Science and Technology, Vol. 19, No2 (2011) 48-60
ISSN 0128-7389

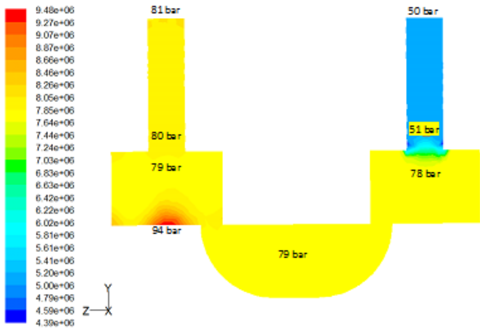


Figure 12: Static pressure contour for a valve with modified spool

Inlet/Outlet modifications

Modification of both inlet and outlet (at the same time) with concaved surfaces resulted in a little improvement, especially at the area of the inlet. Figure 13 shows high pressure drop between the inlet and outlet (ΔP = 29 bar).

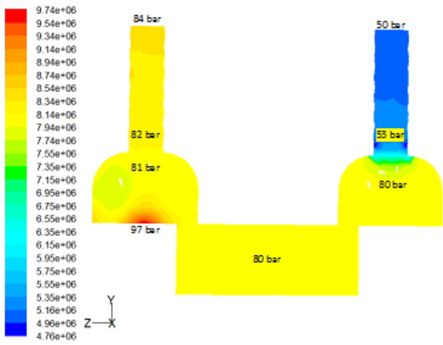


Figure 13: Static pressure contour for modified valve

Inlet modifications

The inlet area of the valve chamber was modified without changing the function of the valve. The modified geometry combines both concaved and conical surfaces at the inlet. The modified valve was simulated, and the results gave the flow pattern and pressure contour as shown in Figure 14 and Figure 15 respectively.

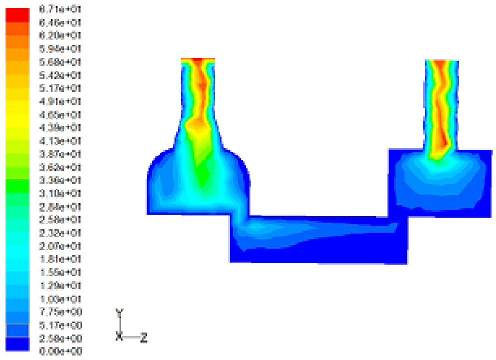


Figure 14: Velocity magnitude contours for inlet-modified valve

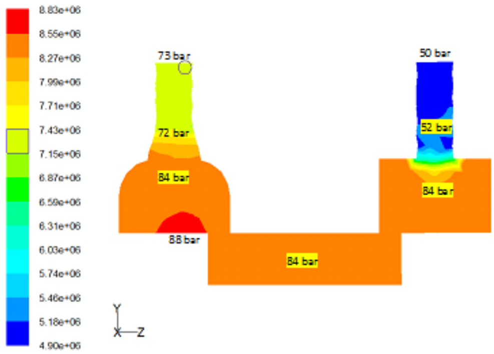


Figure 15: Static pressure contour for inlet-modified valve

Results showed a fair improvement in pressure drop (ΔP = 20 bar), but this was accompanied with noticeable increase in pressure inside the whole valve chamber.

Outlet modifications

The outlet area of the valve chamber was modified without changing the function of the valve. The modified geometry combines both concaved and conical surfaces at the outlet as shown in Figure 16.

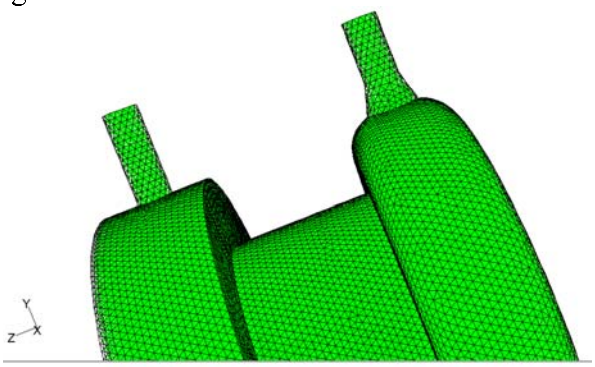


Figure 16: Wall grid of the outlet-modified valve

Solid State Science and Technology, Vol. 19, No2 (2011) 48-60
ISSN 0128-7389

Simulation of this proposed geometry gave the flow pattern and pressure contour as shown in Figure 17 and Figure 18 respectively.

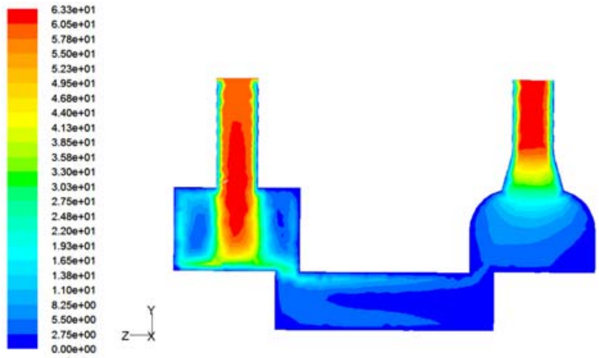


Figure 17: Velocity magnitude contours for outlet-modified valve

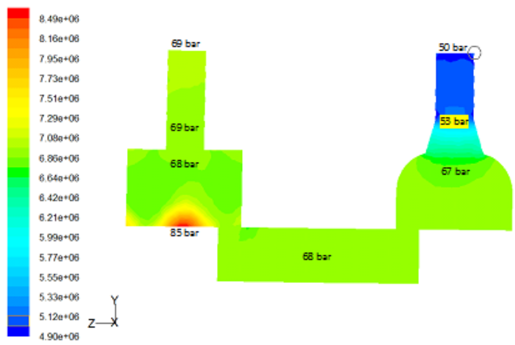


Figure 18: Static pressure contours for outlet-modified valve

The computed results showed that pressure drop for this geometry decreased significantly (ΔP =16 bar). Therefore, careful modifying of the valve chamber outlet geometry contributes good improvement (50%).

CONCLUSIONS

The relation between mass flow rate and pressure drop across the spool valve was studied by the simulation of the flow using different mass flow rates. Results showed that, the pressure loss increases rapidly as the mass flow rate increases. The relation between the pressure drop and the valve geometry was studied as well. Different geometrical designs were proposed for both the valve chamber and the spool. The proposed valves were simulated. Results show that the pressure drops is highly affected by the valve chamber geometry and not by the spool geometry. It was noticed that the shape of inlet and outlet areas of the valve chamber has the most significant effect on the flow behavior of the fluid, and hence on the pressure drop across the valve. The study concluded that, the modification of the outlet area of the valve chamber is the most effective way to reduce the pressure drop.

ACKNOWLEDGEMENT

We would like to express our gratitude to all those who gave us the possibility to complete this study. We want to thank the Department of Mechanical Engineering at Universiti Teknologi PETRONAS (UTP) for giving us permission to commence this study in the first instance, to do the necessary research work and to use departmental facilities. We have furthermore to thank the Center for Graduate studies at UTP for sponsoring this study.

REFERENCES

[1] G. Bella . M. Burroni, M Cerimele, F Pistella, *Mathematics and Computers in Simulation,* **51** (1-2) (1999) 35-45
[2] C. Bingang . *Machine Tool and Hydraulics*, 1991.2
[3] M. Stevenson and X. Chen, *Journal of Food Engineering,* **33** (1-2) (1997) 151-165
[4] S. Oshima and I. Tsuneo, *Bulletin of JSME* , **28** (244) (1985) 2264-2271
[5] *S. Oshima, T. Leino, M. Linjama, K. T. Koskinen, M. Vilenius International journal of fluid power* , 2(3) (2001) 5-13
[6] W. Linxiang , C. Ying , R. Xiaodong., L. Yongxiang, "Experiment study on fluid flow in fluid power system" , *ICFP*, 97 , 1997
[7] K. ITO, "Flow in a poppet valve," *Proc of 1th international symposium on FPTC*, 1991 Beijing , China.
[8] Min., Fu, X., Chen, Y., "Computational fluid dynamics approach to pressure loss analysis of hydraulic spool valve," *SEM and Zhejian Provincial Natural Science Foundation*, FCEM., (2000033501)
[9] F. Bertelli, M. M. Pariona, " *Numerical simulation of fracture mechanics in a bronze valve that leads water flow*" 17 ºCbecimat - Congresso Brasileiro de Engenharia e Ciência dos Materials, 15-19 Nov. 2006, Foz do Iguaçu, PR, Brasil.