International Journal of Automotive Technology, Vol. 8, No. 5, pp. 615−622 (2007)

1229−9138/2007/034−10

CFD-FEA ANALYSIS OF HYDRAULIC SHOCK ABSORBER VALVE BEHAVIOR

M. SHAMS[1)*], R. EBRAHIMI[1)], A. RAOUFI[1)] and B. J. JAFARI[2)]

[1)]Mechanical Engineering Department, K. N.Toosi University of Technology, No.15 Pardis Sq., Mola Sadra St., Vanak Sq., Tehran, Iran
[2)]Research and Development Department, Indamin Saipa Shock Absorber Mfg. Co., 14th Km, Karaj Special Road, Tehran, Iran

(Received 7 Jaunary 2007; Revised 3 August 2007)

ABSTRACT–In this study, a Coupled Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA) method are used to predict and evaluate the performance of an automotive shock absorber. Averaged Navier-Stokes equations are solved by the SIMPLE method and the RNG k-ε is used to model turbulence. CFD analysis is carried out for different intake valve deflections and piston velocities. The force exerted on the valve in each valve deflection is obtained. The valve deflection - force relationship is investigated by the FEA method. The force exerted on the valve in each piston velocity is obtained with a combination of CFD and FEA results. Numerical results are compared with the experimental data and have shown agreement. Dependence of valve deflection as a function of piston velocity is investigated. Effects of hydraulic oil temperature change on valve behavior are also studied.

KEY WORDS : Shock absorber, Hydraulic damper, Hydraulic valve, CFD simulation, FEA method

NOMENCLATURE

C	: temperature sensitivity (K)
D	: valve diameter (mm)
d_{max}	: maximum valve deflection (mm)
T	: absolute temperature (K)
T_0	: reference absolute temperature (K)
α	: volumetric thermal expansion coefficient (K^{-1})
μ	: oil viscosity (mPas)
μ_0	: reference oil viscosity (mPas)
ρ	: oil density (kg/m^3)
ρ_0	: reference oil density (kg/m^3)
μ	: oil viscosity (mPas)
v	: velocity (m/s)

1. INTRODUCTION

A shock absorber or damper is an energy dissipating device, whether an old, dry friction one or a newly developed MR damper which uses Magnetically Responsive fluid, all turn the kinetic energy of movements into thermal energy. Shock absorbers are used where undesirable oscillations are to be damped as in vehicle suspension systems. Using dampers in vehicle suspension systems not only provides a good ride comfortable but also improves vehicle handling, which is due to their better tire-road contact.

Today's most used shock absorbers are twin tube hydraulic ones. A twin tube hydraulic damper consists of two coaxial tubes: an inner or pressure tube and an outer or reserve tube. There are also two hydraulic valve mechanisms: one moving piston valve which is fitted into a piston body that moves within hydraulic oil inner tube, and one fixed base valve which is fitted into the body of damper at the end of the inner tube. A schematic view of a twin tube hydraulic damper is depicted in Figure 1. Most hydraulic dampers use valves which are usually variable area orifices. The area of flow passage varies while pressure is changing. This mechanism is usually provided by either a disc with a coil spring, or just a deflective disc made of spring steel (Figure 2). Sometimes, however, a combination of both is used. There are also some fixed area orifices in parallel with these valves to give some flow even when the valves are fully closed. When a damper is acting with a specific speed of a moving piston, the hydraulic oil is displaced through these valves, making a pressure difference across the valves and creating a force opposing piston motion (Baracat, 1993). The explosive view of the piston valve model is shown in Figure 3.

Two methods are used for modeling or numerical simulation of the automotive shock absorber. One is

*Corresponding author. e-mail: shams@kntu.ac.ir

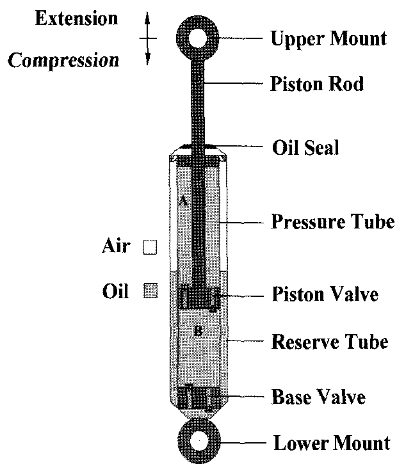

Figure 1. Schematic of a twin tube hydraulic damper.

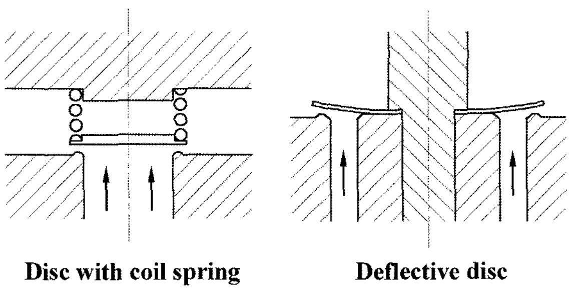

Figure 2. Two kinds of valves usually used in hydraulic dampers.

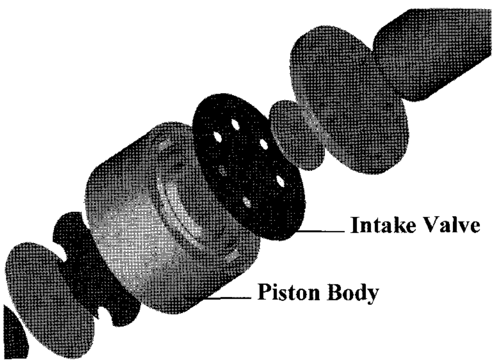

Figure 3. Explosive view of the piston valve 3D model.

governed by ordinary differential equations. Examples of this type can be found in hydraulic and electric power control systems. In this method of modeling, the system is represented as springs and dashpots networked in some style. The overall structure of modeling consists of a pressure model(s) and valve pressure/flow characterizations. The pressure models are a selection of first order non-linear differential equations that are utilized to determine the various internal chamber pressures and are derived from pressure dependent oil compressibility models.

The other type is governed by partial differential equations. Detailed fluid mechanics and structural dynamics are obtained in this method. This method involves the integration of two sets of partial differential equations.

Shock absorbers have been studied both numerically and experimentally. A physical model for a shock absorber was developed by Surace *et al.* (1992). This model provided a more realistic representation of the stiffness characteristics than previous simple models.

Reybrouck (1994) presented a nonlinear physical model for predicting shock absorber performance. Morinaga *et al.* (1997) studied the mechanism of shock absorber rattling noise by valve behavior analysis and CFD simulation.

Duym and Reybrouck (1998) identified valve parameters from several simple dynamometer tests and an additional incompressible model; these parameters were then used to analytically determine valve flows for given pressure drops in the main model. Duym (2000) attempted to more readily generate an identifiable physical damper model.

Herr *et al.* (1999) evaluated the flow and performance of an automotive shock absorber by coupling a Computational Fluid Dynamics method with a dynamic modeling technique. They evaluated the pressure flow characteristics of the differing valve components using Computational Fluid Dynamics (CFD). Disk deformation information from finite element analysis simulations are incorporated into the calculation of blow-off valve opening. They obtained the damping force versus piston velocity curves.

The radial flow of magnetic fluid under the piston of a magneto fluid shock absorber was studied by Krakov (1999). Purdy (2000) developed a non-linear model that includes the compressibility of the fluid, trapped gas and expansion of the cylinder. The performance of the shock absorber model was examined as the parameters were varied. Interaction of fluid and deformable structural parts in the shock absorber caused difficulties e.g. compelling the kinematic compatibility at the fluid-structure interface and updating the geometry of the domain. Tallec and Mouro (2001) presented a Lagrangian-Eulerian approach to overcome these difficulties and an improved solution by using CFD. Lee and Moon (2005, 2006) proposed a new mathematical dynamic model of a displacement-sensitive shock absorber to predict the dynamic characteristics of an automotive shock absorber. They simulated the fluid flow in the shock absorber, on the base of the modes of damping force according to the position of piston, to the hard, transient, and soft zone.

It is obvious that much attention should be paid to the design of damper valves for the damper to act properly.

Usually, the design of hydraulic damper valves has been usually a trial and error process and needs skilled people to obtain the best damper behavior because of the complex nature of fluid flow in a hydraulic damper. Numerical simulation of hydraulic damper valves not only saves much of the time and money usually spent in making specimens and performing tests, but also provides design insight that in turn makes the practical design problems easier to handle. The shock absorbers perform through the interaction between working fluid and mechanical valve structure. The ability of predicting valve behavior while the damper is in action using Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA) methods is the concern of this paper.

2. FLOW SIMULATION

The CFD modeling involves the numerical solution of the conservation equations in the laminar and turbulent fluid flow regimes. Therefore, the theoretical predictions were obtained by simultaneous solution of the continuity and the Reynolds averaged Navier-Stokes (RANS) equations. The governing equations for an incompressible flow were:

Mass conservation:

$$\vec{\nabla}.(\vec{v})=0 \tag{1}$$

Momentum conservation:

$$\rho\left[\frac{\partial(\vec{v})}{\partial t} + (\vec{v}.\nabla)\vec{v}\right]=\rho g-\nabla P+\mu\nabla^2\vec{v} \tag{2}$$

Since the flow in the shock absorber is in a state of turbulent motion, it is important to use an appropriate turbulence model for evaluating the flow field. The standard k–ε model is a semi-empirical model based on transport equations for the turbulent kinetic energy (k) and its dissipation rate (ε). While the k–ε model is widely used in industrial applications, it suffers from several shortcomings. Improvements have been made to the model to improve its performance by the RNG model (Yakhot and Orszag, 1986). In the RNG k–ε model, the effect of small-scale turbulence is represented by means of a random forcing function in the Navier-Stokes equations. The RNG procedure systematically removes the small scales of motion from the governing equations by expressing their effects in terms of larger-scale motion and a modified viscosity. RNG k–ε model have shown substantial improvements over the standard k–ε model where the flow features include strong streamline curvature, vortices and rotation. The two-equation k–ε model consists of an equation for the turbulent kinetic energy, k, and the other for the energy dissipation rate, ε, (Yakhot and Orszag, 1986).

In the present work, the finite volume method with unstructured meshes is used to solve the three dimensional incompressible Navier-Stokes equations. The flow governing equations are solved with FLUENT™ (1998). The velocity and pressure coupling used the SIMPLE algorithm. RNG k–ε model is used to model turbulence. A grid with about 1,300,000 tetrahedral cells is generated for analyzing the flow in the shock absorber. The grid independency is checked in order to ensure that the solution is independent to the grid number and size. Figure 4 shows the details of the computational grid for the shock absorber model. A fine grid is used in the regions with high rate of changes.

3. RESULTS

In practice, the first step of the damper valve CFD simulation process begins with drawing a CAD model of the valve. The computational domain consists of a deflective disc called intake valve seated on the piston body. The 3D models of the piston body, intake valve and belonging parts are shown in Figure 3. The intake valve's role in the shock absorber is as follows: when the piston is moving down in the inner tube (compression motion), oil pressure in chamber 'B' (see Figure 1) increases and that of chamber 'A' decreases. Therefore, oil flows from chamber 'B' to chamber 'A' through the intake valve which deflects to let the oil displace. At the same time, a volume of oil equal to the volume of the piston rod inserted into the inner tube flows from chamber 'B' into the reserve chamber (i.e. the space between inner and outer tubes) through the base valve. The intake valve's holes play the role of fixed area orifices to let the oil flow

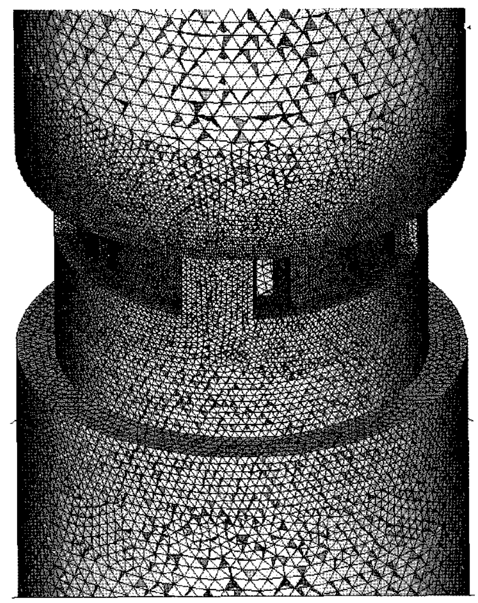

Figure 4. CFD Mesh for the shock absorber.

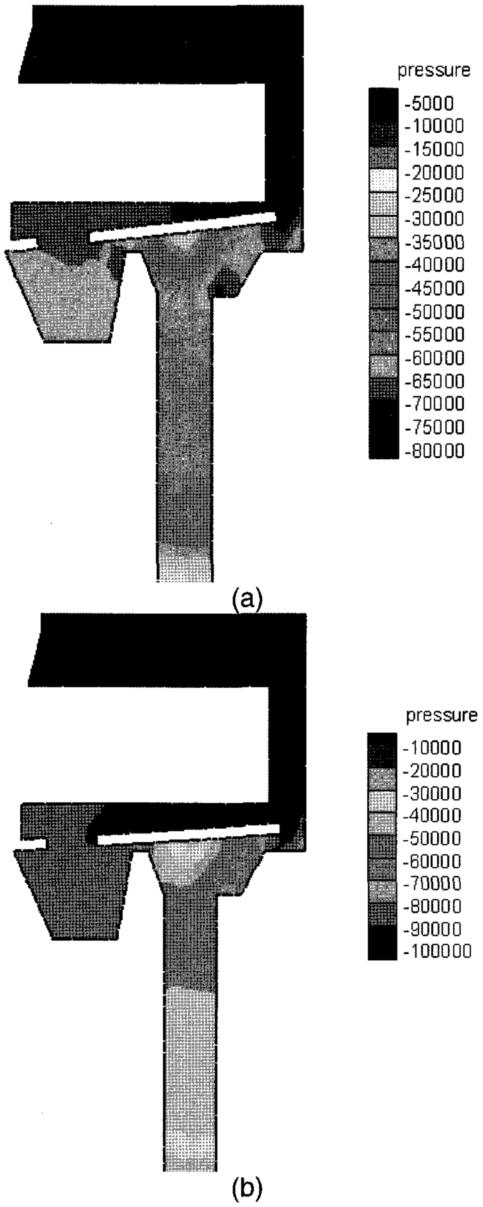

Figure 5. Contours of relative pressure (Pa) obtained from valve CFD simulations for the frequency of 1.254 Hz and valve deflections of (a) 0.5 mm (b) 0.3 mm.

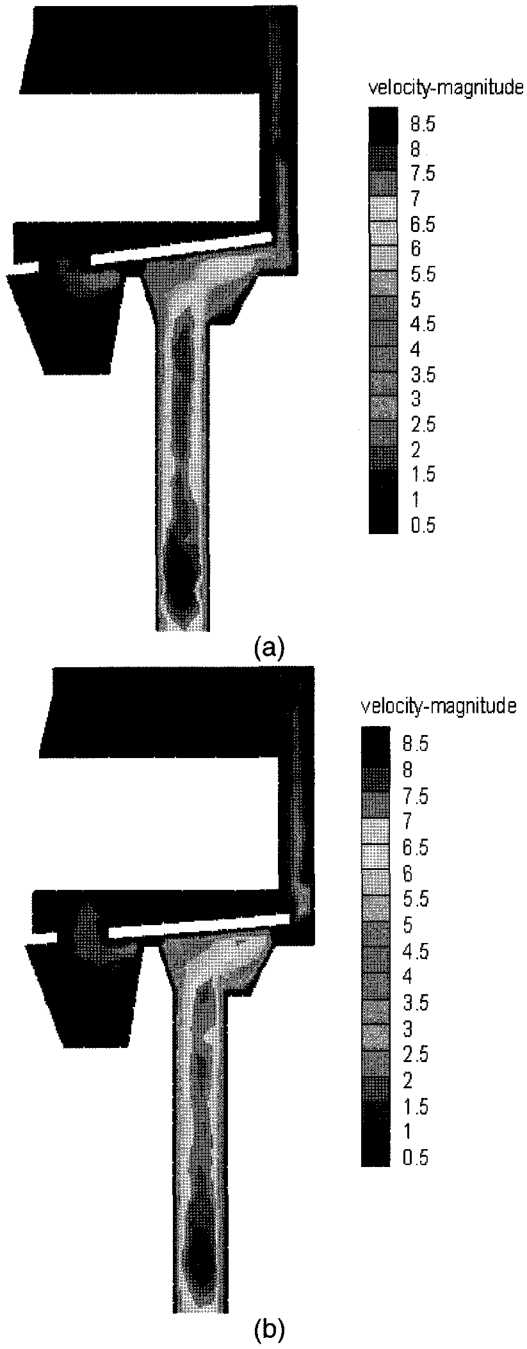

Figure 6. Contours of velocity (m/s) obtained from valve CFD simulations for the frequency of 1.254 Hz and valve deflections of (a) 0.5 mm (b) 0.3 mm.

when the intake valve is closed, i.e. during compression motion at low piston velocities, and during extension motion at all piston velocities.

The operating temperature of the oil is 20°C. Seven different intake valve deflections: 0.05, 0.1, 0.2, 0.3, 0.4, 0.5, and 0.6 mm are considered. These are the maximum deflections that happen at the edge of the valve while the piston displaces. Six different piston velocities for each valve deflection, i.e. 0.05, 0.1, 0.2, 0.3, 0.4 and 0.5 m/s are used. This is the range of the piston velocity that is usually used for behavior evaluation of a conventional vehicle damper. The corresponding frequencies of the mentioned piston velocities are 0.209, 0.418, 0.836, 1.254, 1.671 and 2.089 Hz.

Figure 5 shows Contours of relative pressure obtained from valve CFD simulations for the frequency of 1.254 Hz and different valve deflections. A strong pressure is

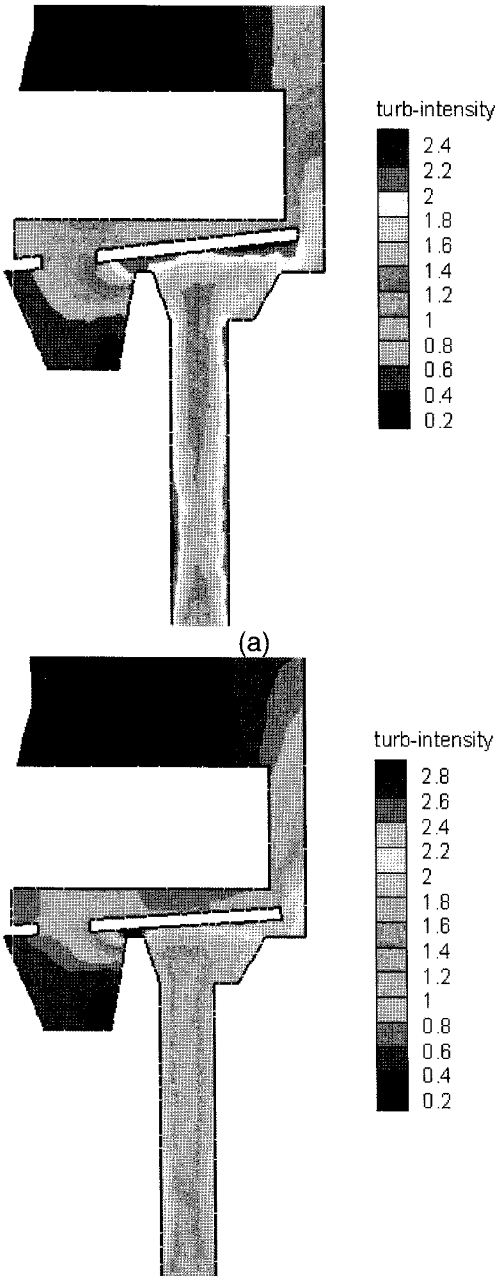

Figure 7. Contours of turbulence intensity (%) obtained from valve CFD simulations for the frequency of 1.254 Hz and valve deflections of (a) 0.5 mm (b) 0.3 mm.

observed. By increasing valve deflection from 0.3 to 0.5 mm, the pressure drop also increases.

Figure 6 shows the contours of the velocity magnitude for the frequency of 1.254 Hz and valve deflections of 0.5 and 0.3 mm. This figure shows that velocity distribution only changes in the vicinity of the valve deflector.

The flow in the shock absorber is turbulent. One of the

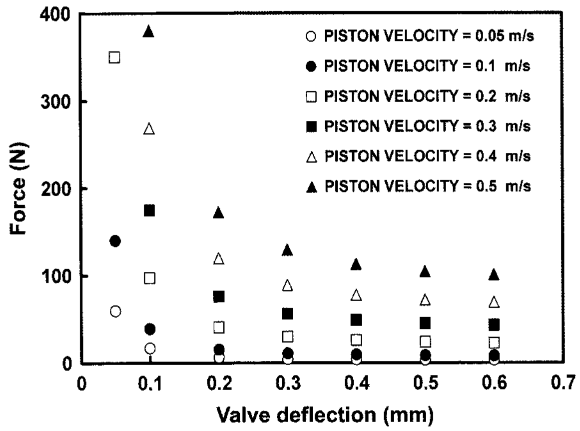

Figure 8. Dependence of force exerted on valve as a function of valve deflection based on CFD simulations data for different piston velocities.

important parameters for investigation of turbulence is turbulence intensity. Contours of turbulence intensity (%) for the frequency of 1.254 Hz and valve deflections of 0.5 and 0.3 mm are indicated in Figure 7. The turbulence intensity increases by increasing valve deflection.

The resultant force exerted on the valve in each valve deflection and for the piston velocities stated previously is calculated and presented in Figure 8. This resultant force includes viscous and pressure forces. This figure shows that the net force decreases by increasing valve deflection due to an increasing pressure drop across the valve. Also, increasing piston velocity increases this force.

A quasi-steady assumption is adopted for considering fluid structure interaction due to low inertial force of the deflector. It means that first, the flow field is simulated and calculated by CFD and the hydrodynamic forces on the deflector are determined. Then, by knowing the forces, the valve deflection is determined by a finite element code (I-DEAS) by solving elasticity equations of the deflector. The valve inertia is very low and hence a small amount of deflection exists. Therefore, a decouple scheme for FEA and CFD is reasonable. Figure 9 schematically indicates the boundary conditions used in CFD and FEA simulations. A free mesh with the 4 node elements is used for finite element analysis. Young's modulus is 206800 MPa and Poisson's ratio is 0.29. The valve deflection is calculated and presented in Figure 10. It shows almost a linear relation between the force and the valve deflection. It is to be noted that the force (N) is the net force acting on the surface of the valve. Since the maximum valve deflection is much smaller than the valve diameter (d_{max}/D=0.025), it is reasonable to consider a linear valve deflection-force curve.

The results of the FEA code and the CFD simulation

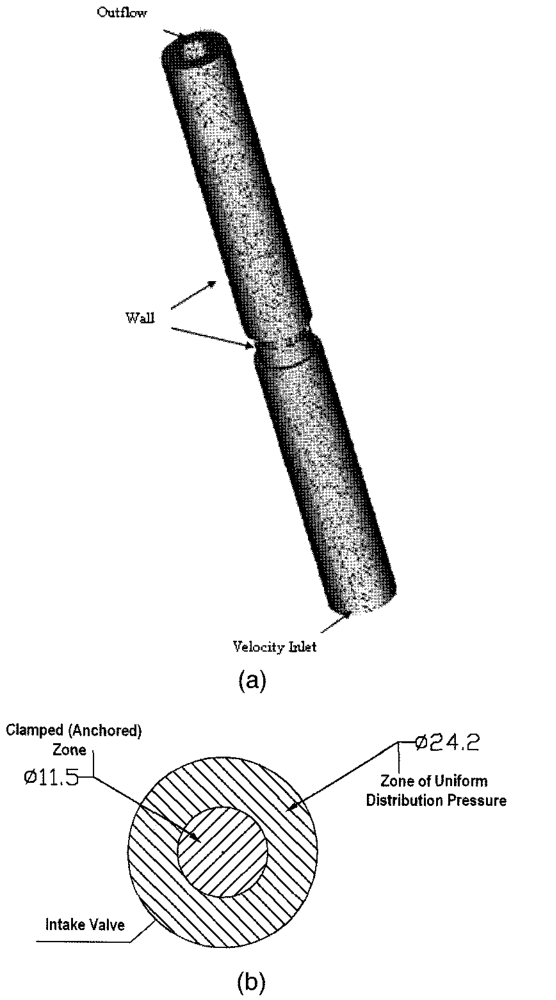

Figure 9. Boundary conditions for (a) CFD simulation (b) FEA simulation.

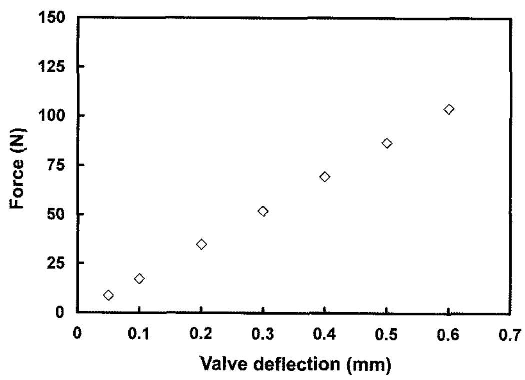

Figure 10. Valve deflection-force relationship based on FEA data.

are reconstructed and presented in Figure 11. The intersection of these two curves is the corresponding valve deflection at each piston velocity.

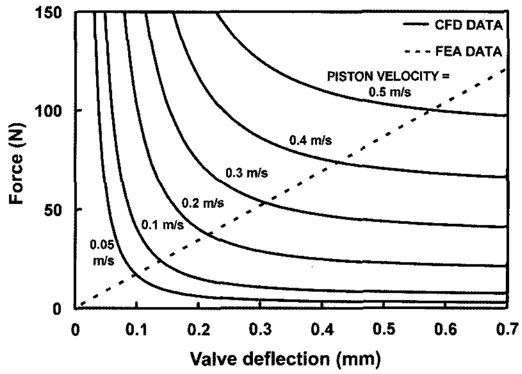

Figure 11. Combination of CFD and FEA simulation data.

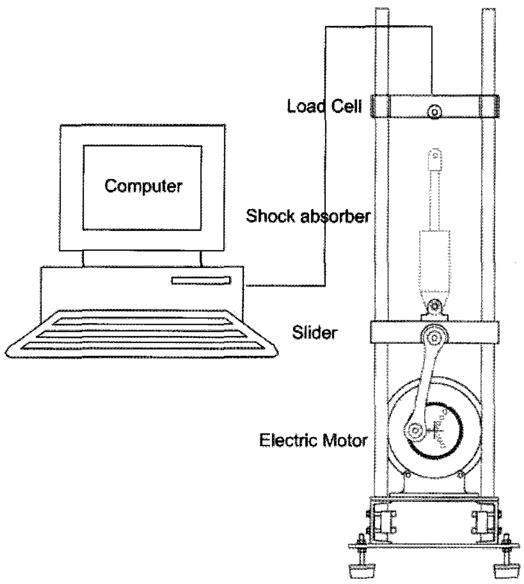

Figure 12. Shock absorber dynamometer.

An experimental setup is also used to validate the accuracy of the numerical scheme. A Shock absorber dynamometer is used and its schematic view is illustrated in Figure 12. The shock absorber dynamometer is a machine to test shocks and generate graphs for the shock characteristics. These graphs could be printed for the shocks or stored so the user could develop a database of how each shock works under the test conditions. This machine replaces the trial and error approach with a reliable and efficient method to determine the shocks used during a race. The shock absorber dynamometer consists of a metallic main structure, a motor with gear box to reduce motor speed, a scotch yoke or a piston-crank slider mechanism to cycle the shock, and a computer with Dumper-Analyzer-Software. The software acquires data and displays graphs of the test. With proper selection of the stroke of the damper and the frequency of

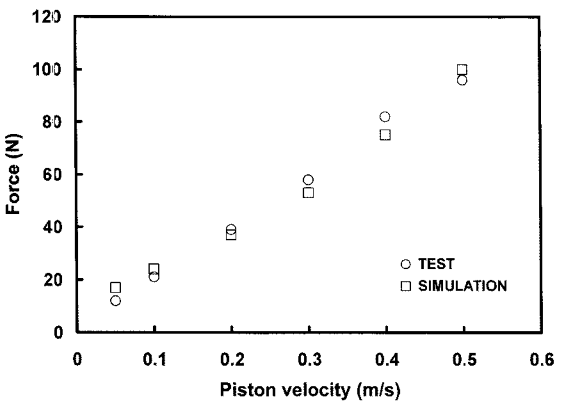

Figure 13. Comparison between valve CFD-FEA simulation data and test results.

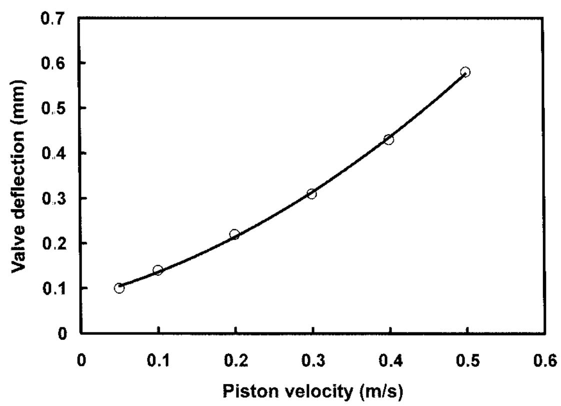

Figure 14. Dependence of valve deflection as a function of piston velocity.

the rotating crank, it is possible to achieve the desired piston valve velocities.

The force exerted on the valve is measured by the dynamometer and presented in Figure 13. Also, the results are compared with numerical simulations that show good agreement.

The dependency of the piston velocity and the valve deflection is also measured and shown in Figure 14. It shows that by increasing piston velocity, valve deflection is also increased.

In order to investigate the effects of hydraulic oil temperature change (environmental conditions) on valve behavior, simulations are repeated for oil temperatures of 20°C, 60°C, and 100°C and for piston velocities of 0.1, 0.3 and 0.5 m/s. The effect of oil temperature on density and viscosity is expressed by the following equations (Dixon, 1999).

$$\rho=\rho_o[1-\alpha(T-T_o)] \tag{3}$$

Where α is the volumetric thermal expansion coefficient and equals to 0.001 K^{-1}.

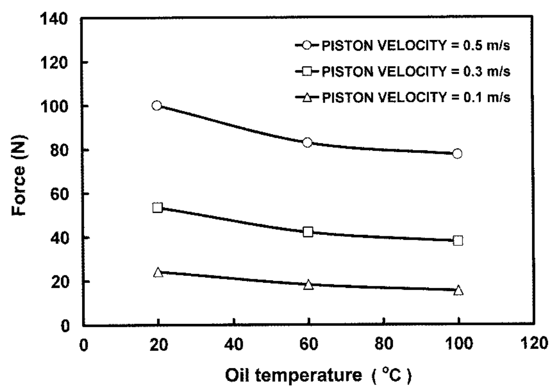

Figure 15. Force exerted on valve as a function of temperature for three piston velocities.

The Guzmann-Carrancio equation is used for oil viscosity (Lee and Moon, 2005):

$$\mu=\mu_o e^{C\left(\frac{1}{T_o}-\frac{1}{T}\right)} \tag{4}$$

Where C is the temperature sensitivity. The effect of oil temperature on the hydraulic damper force is presented in Figure 15. The results show that the force decreases by increasing temperature for all piston velocities. This figure also shows that the force reduction is almost 0.35%/°C on average.

4. CONCLUSION

In this study, a simulation method for estimating the performance of an automotive shock absorber is developed. A numerical scheme based on the CFD is used to determine the flow characteristics of a hydraulic valve working in a hydraulic shock absorber, while FEA code solves the elasticity equations and determines the valve deflection. The effects of fluid temperature change on valve flow characteristics are also verified by this numerical scheme. The CFD analysis shows the detailed fluid flow in the shock absorber. The simulation results are compared with the test data. The results show good agreement with the experimental data.

ACKNOWLEDGEMENT–The support of Indamin Shock Absorber is gratefully acknowledged.

REFERENCES

Baracat, D. E. (1993). A proposal for mathematical design of shock absorbers. *SAE Paper No.* 931691.

Dixon, J. C. (1999). *The Shock Absorber Handbook.* SAE R-176. Society of Automotive Engineers. Warrendale, Pa.

Duym, S. (2000). Simulation tools, modelling and identification, for an automotive shock absorber in the context of vehicle dynamics. *Vehicle System Dynamics* **33**, **4**, 261–289.

Duym, S. and Reybrouck, K. (1998). Physical Characterization of nonlinear shock absorber dynamics. *European J. Mechanical Engineering* **43**, **4**, 181–188.

FLUENT User Guide (1998). Fluent Incorporated, Centerra Resources Park, 10 Cavendish Court, Lebanon, NH 03766.

Herr, F., Mallin, T., Lane, J. and Roth, S. (1999). A shock absorber model using CFD analysis and easy 5. *SAE Steering and Suspension Technology Symp.*, Detroit, Michigan, 267–281.

Krakov, M. S. (1999). Influence of rheological properties of magnetic fluid on damping ability of magnetic fluid shock absorber. *J. Magnetism and Magnetic Materials,* **201**, 368–371.

Lee, C. T. and Moon, B. Y. (2005). Study of the simulation model of a displacement-sensitive shock absorber of a vehicle by considering the fluid force. *J. Automobile Engineering,* **219**, 965–975.

Lee, C. T. and Moon, B. Y. (2006). Simulation and experimental validation of vehicle dynamic characteristics for displacement-sensitive shock absorber using fluid-flow modelling, *Mechanical Systems and Signal Processing*, **20**, 373–388.

Morinaga, H., Kuboto, M. and Kume, H. (1997). Mechanism analysis of shock absorber rattling noise. *J. SAE Review* **18**, **2**, 198–198.

Purdy, D. J. (2000). Theoretical and experimental investigation into an adjustable automotive damper. *Proc. Institution of Mechanical Engineers, Part D, J. Automobile Engineering* **214**, **3**, 265–283.

Reybrouck, K. (1994). A nonlinear parametric model of an automotive shock absorber. *SAE Paper No.* 9400869.

Reybrouck, K. and Duym, S. (1998). A physical and parametric model for nonlinear dynamic and temperature-dependant behaviour of automotive shock absorbers. *Proc. 11th ADAMS Users Conf.*, Frankfurt.

Surace, C., Worden, K. and Tomlinson, G. R. (1992). An improved nonlinear model for an automotive shock absorber. *Nonlinear Dynamics* **3**, **6**, 413–429.

Tallec, P. L. and Mouro, J. (2001). Fluid structure interaction with large structural displacements. *Computer Methods in Applied Mechanics and Engineering*, **190**, 3039–3067.

Yakhot, V. and Orszag, S. A. (1986). Renormalization group analysis of turbulence I; Basic Theory. *J. Sci. Comput.*, **1**, 1–51.