Investigation of pressure variation & fluid flow behavior inside the shock absorber piston assembly using CFD simulation & Design of Experiments

Pritam P. Bhiungade
Project internship trainee
(at Gabriel India Limited, Pune)
MTech student, Mechanical Engineering Department,
SVPCET, Nagpur-441108
pritambhiungade@gmail.com

Swapnil S. Kulkarni
Assistant Manager
Gabriel India Limited
29th milestone, village-Kuruli
Pune-Nashik Highway,
Tal-Khed, Pune-410501
swapnil.kulkarni@gabriel.co.in

B. Ravi
Deputy General Manager
Gabriel India Limited
29th milestone, village-Kuruli
Pune-Nashik Highway,
Tal-Khed, Pune-410501
b.ravi@gabriel.co.in

G. R. BOOB
Assistant Professor
Mechanical Engineering Department,
SVPCET, Nagpur-441108
ghanshyamboob@rediffmail.com

Abstract
The Shock absorbers are the hydraulic dampers and fluid flow is governed through predefined passages. This fluid flow passages are responsible for variation in damping force with respect to velocity. This paper addresses the changing fluid flow domain accounts for change in fluid flow behavior. CFD simulation is carried out for shock absorber piston assembly. The actual effects of each subcomponent used in piston assembly were obtained through DOE study. In case of DOE study, 8 physical samples were developed & tested. Here, CFD simulation models were built by incorporating these sub components at their occurrence stage. This will reduce the overall computation time and efforts required for complete CFD simulation. The DOE study gives an understanding on the assembly behavior. Here, each component used in assembly has unique contribution on response variable (Damping characteristics) and these components contribute at given velocities. The full factorial design was used to understand the individual effects of these piston assembly sub components on response variable (damping characteristics). In this selected design, three factors plays vital role on response variable. The factors are namely Orifice, Deflection disk, and Piston fluid ports. Three main components like notches, deflection disks, and piston fluid ports affect the damping force at different velocity. The role of these piston assembly sub components on pressure variation & fluid flow behavior are considered for CFD simulation.

Introduction
Shock absorbers are important component of the suspension system. These are mainly hydraulic dampers. Shock absorbers are required in vehicles like passenger cars, commercial vehicles, two wheelers, three wheelers etc. The main functional of shock absorber is damping. Shock Absorber performance depends on the design feature of piston & base valve assembly during rebound and compression stroke respectively. Piston assembly consist piston, deflection disks, back up disk, orifice/ notched leaf valve etc. The comfort and control in rebound stroke of damper is achieved by making variation in design of piston assembly components. Experimentation and from historical data states that notches, deflection disk deflection & fluid flow ports in piston makes greater impact on rebound damping characteristics of shock absorber. By changing number of notches, variation in deflection disk deflection, number of holes & changing hole size will gives different pressure trend for low to high velocity damping. Hence, this paper addresses the characteristics of notch leaf valve, deflection disk, and fluid flow ports using DOE study. The effect on damping characteristics are studied with help of Design of Experiment [DOE] and verified by use of CFD simulation.
Process Methodology

In this study the fluid geometry of the problem is modeled by the use of CAD software, all geometry dimension are interconnected to ensure the stability of fluid ports geometry during the simulation update. Oil is taken as working fluid. This incompressible Newtonian oil has temperature independent density and viscosity, so the model is assumed to be isothermal in nature. CFD simulation is carried for two cases. In "Case-a" 6 fluid ports are considered each with the port diameter of 1.5mm & total flow area is 10.6mm². In "Case-b" 8 fluid ports are considered in which 4 fluid ports each with the port diameter of 1.5mm & another 4 fluid ports each with the port diameter of 1 mm & total flow area is 10.29mm², which is shown in Fig 1 & 2. In Fig 3, Notch leaf valve and disk deflection is modeled.

![Fig 1: Case-a](image1)
![Fig 2: Case-b](image2)
![Fig 3: Model of fluid flow domain](image3)

**Boundary Condition**

Different boundary conditions are used to ensure the flow to enter and exit from the fluid flow domain. As this problem contain the incompressible fluid flow hence, applying the boundary condition as velocity inflow at inlet boundary surface of the model and outflow at outlet boundary surface of the model. Boundary conditions given to the model are inflow, outflow, and wall.

![Fig 4: CFD meshing for Case-a](image4)
![Fig 5: CFD meshing for Case-b](image5)

**DOE study through physical testing- low velocity**

Eight physical samples were built and physically tested to ensure the effect of individual components on response variable.

![Graph1: Response at low velocity](image6)

Graph 1 shows that at low velocity notch leaf valve and deflection disk shows major effect on response variable (damping).
CFD model as per DOE study
The notch leaf valve is modelled in Acusolve as the major contribution of notch leaf valve at low velocity is obtained through DOE. The same low velocity magnitude is taken for simulation to understand the flow behavior through notch leaf valve.

Results & Discussions

Pressure variation for low velocity

Fig 6: Pressure distribution for Case-a
Fig 7: Pressure distribution for Case-b

Fig 6 and figure7 shows that as number of notches increases from 2 to 8, increase in notch fluid flow area will give reduction in pressure from Case-a to Case-b with respect to given velocity.

DOE study through physical testing – Transition velocity

Graph 2 shows at transition velocity, deflection disk shows major effect on response variable (damping force) but to understand the effect of disk deflection in chamber, disk deflection with notch leave valve are modeled as per deflection pattern. Disk deflection is very difficult to obtained using experimental work. This uniform deflection of disk is obtained by structural simulation where disk stiffness comes into picture. In case of structural simulation of disk deflection, uniform fluid loading is one approximation. It is directly taken for further CFD simulation. CFD simulation is carried out to understand the flow in shock absorber chamber & change in pressure at that location is observed.

Fig 8: Deflection at 0.4mm
Fig 9: Deflection at 0.7mm
Fig 10: Deflection at 1mm
**CFD model as per DOE study**

The different disk deflection geometries with notch leaf valve is modelled in Acusolve as the major contribution of deflection disk and notch leaf valve at transition velocity is obtained through DOE. The same transition velocity magnitude is taken for simulation to understand the flow behavior through deflection disk deflection and corresponding pressure differentials.

**Pressure variation for transition velocity**

CFD Results—2-Dimensional representation

- Normalized pressure: 1.23 MPA
- Normalized pressure: 1.17 MPA
- Normalized pressure: 1.14 MPA

Figure 11, 12, and 13 shows that at pressure seems to be reduced due to disk deflection at same (one particular) transition velocity.

Graph 3 shows that, pressure changes due to disk deflection. Pressure decreases at one particular velocity due to the increase in fluid flow area over deflection disk. This variation in pressure trend is mainly depends on disk stack thickness and preloading conditions considered in structural simulation.
Graph 4 shows that, increase in pressure trend at different transitional velocities. This is also depending on stiffness of disk stack in structural simulations.

**DOE study through physical testing – High velocity**

Graph 5 shows that at high velocity, piston ports and deflection disk contributes more on response variable. This is obtained by carrying out successive trials by changing the piston port size and number of ports in piston. The high velocity magnitude is checked to obtain the influence of piston port configuration.

The CFD simulation is carried out to understand the flow behavior in shock absorber chamber & change in pressure at that location is observed. In order to understand the effect of piston ports at high velocity, the experiments were performed as defined by Case-a piston & Case-b piston configurations. The result obtained from this experiments shows that the pressure variation for both the configurations. Case-a having more flow area as compared to Case-b. Hence Case-a will have less fluid pressure as compared to Case-b piston.

Case a: with 6 piston ports- each port diameter 1.5 mm (Flow area=10.6 mm$^2$)
Case b: with 8 piston ports – 4 ports diameter- 1mm and 4 ports diameter -1.5 mm (Flow area= 10.29 mm$^2$)

**CFD model as per DOE study**

The different piston port sizes and number of ports were modeled in Acusolve as port operating velocity is obtained from DOE study.
Pressure variation at high velocity

Normalized pressure: 4.02 MPA

Fig 14: Pressure distribution for Case-a

Normalized pressure: 4.13 MPA

Fig 15: Pressure distribution for Case-b

As velocity increases pressure increases but case-b piston has maximum pressure gain as compared to the case-a piston because case-b piston has less fluid flow area as compared to the case-a piston fluid flow area. Due to variation in fluid port area, resistance area inside the case-b piston domain is large as compared to resistance area inside the case-a piston domain. If the difference in flow area of mentioned configuration will be more then the pressure differential trend will differ more.

Benefits Summary
This CFD simulation illustrates most appropriate fluid flow behavior inside the shock absorber piston assembly. The accuracy can be enhanced by carrying out most appropriate fluid structure interaction simulation. This simulation helps to understand the effect of different fluid flow domain on fluid pressure behavior.

Conclusions
Physical DOE study ensures the most appropriate CFD model to reduce the time and cost involved in complex fluid structure interaction analysis. This physical DOE study addresses the operating conditions (specific velocity) at which any new valve component influences the performance. New valve components are incorporated in CFD to understand the flow behavior and pressure differential. Influence of each component at their occurrence stage is determined through DOE and modeled in CFD.

ACKNOWLEDGEMENTS

The authors express their thanks to Mr. Shridhar Nanal, Vice President Gabriel India Limited, R&D and Altair Engineering India for their technical support on this project.

REFERENCES