Bulletin of the *Transilvania* University of Braşov
Series I: Engineering Sciences • Vol. 6 (55) No. 2 - 2013

CFD PREDICTION OF THE FLOW INSIDE THE DAMPER BY THE USE OF LES MODEL

M. DIUDEA[1] V. HODOR[1] R. BĂLAN[1] M. BARA[1]

Abstract: *There are two sources that contribute to the definition of the excitation spectrum in the privacy of the dynamic behavior of the suspension level. One of the case concerns with the induced due to the asperities effect of the tread in contact with the tire. A second source of disturbance comes from the turbulence effect coming from the vehicle shell aerodynamic imperfection. The analysis refers to the behavior of fluid flow within the damper, at the interstitial area of the passage and the pilot floating pill. For this it was necessary that the area of interest to be treated in an unsteady regime, with moving wall and deformable mesh.*

Key words: *vehicle rolling, damper, flow, CFD, LES, deformable mash, spectral analyze.*

1. Introduction

Normally, vehicle suspension is used to attenuate unwanted vibrations from various road conditions. This can be accomplished by employing suspension system. So far, three types of suspension system have been proposed and successfully implemented; passive, active, and semiactive. The passive suspension system featuring oil damper provides design simplicity and cost-effectiveness. However, the performance limitations are inevitable owing to the lack of controllability.

The active suspension system provides high-control performance in wide frequency range. However, this type may require high-power sources, many sensors, servovalves and sophisticated control logic. One way to resolve these requirements of the active suspension system is to use the semiactive suspension system. The semiactive suspension system offers a desirable performance generally enhanced in the active mode without requiring large power sources and expensive hardware [1].

Vehicle shock absorber design involves an analysis of the flow through the telescope accurately by CFD prediction. When driving vehicles, there are two sources that contribute to the definition of the excitation spectrum in the privacy of the dynamic behavior of the suspension level. One of the case concerns with the induced due to the asperities effect of the tread in contact with the tire.

A second source of disturbance comes from the turbulence effect coming from the vehicle shell aerodynamic imperfection. This accurate analysis of the flow conditions inside damper, is imposed by

[1] Dept. of Mechanisms, Precision Mechanics and Mechatronics, Technical University of Cluj-Napoca.

the mixed effect coming from the compressibility changes in the fluid flow and the reaction of the spring spacer. This spring runoff opposing fluctuating demands, continuous modify the position of a pilot pill floating in relative positions more or less close (eights) to the holes manufactured to in the piston to pass/join the two damper chambers.

Accuracy in pressure reading, imply to identify specific spectrum at monitoring points distributed in the critical flowing path (constrained at: compressibility changes and dynamics of the flow reversal), was possible only with the use of LES model.

In this paper, it is shown only the research on specific CFD approach, following that in the second, to present the technique of processing pressure values/ strengths, monitored in critical areas. Shock absorbers play a role in the suspension of the car, the main function is to reduce vibration damper body and the wheel, thereby maintaining a firm and constant contact between wheel and road.

Working principle of the hydraulic damper is based on the transformation of mechanical energy into thermal energy of oscillation. The state of art in hydraulic damper is a major issue in the Vehicle System Dynamics, as it is to be seen in papers like: Valve performance can be predicted by coupling the valve deflection with CFD pressure results.

This technique involves sequential geometry and simulation updating, while varying both the geometry and flow-rate. The valve deflection is calculated by post-processing the pressure distribution" from “Investigation of Damper Valve Dynamics Using Parametric Numerical Methods” F.G. Guzzomi, P.L. O’Neill and A.C.R. Tavner, 16th Australasian Fluid Mechanics Conference Crown Plaza, Gold Coast, Australia 2-7 December 2007 [3].

Herr et al. made a CFD component analysis showing unique features of flow pattern, discharge coefficients, and pressure distribution for various shock absorber components on the “A Shock Absorber Model Using CFD Analysis and Easy5” Herr, F., Mallin, T., Lane, J., and Roth, S., SAE Technical Paper 1999-01-1322, 1999 [4].

Present studies, involves the monitoring of the continuous variation in terms of pressure (force resisting) related to the relative positioning of the piston (depending on the specific down force). Shock absorbers play a role in the suspension of the car, the main function is to reduce vibration damper body and the wheel, thereby maintaining a firm and constant contact between wheel and road.

Working principle of the hydraulic damper is based on the transformation of mechanical energy into thermal energy of oscillation. Most dampers are double acting, working in both directions near the front wheel, the body opposes less resistance, and that the removal of bodywork wheels oppose more resistance, to cushion vibrations. At the same time this is accomplished by maintaining a contact with the ground.

In building a car, correct dimensioning of package spring-Shock has a great importance. When passing over a bump, the energy resulting from motion compression is stored in the spring. It will try to release by extension, the energy stored in compression. This phenomenon would produce body movements, movements that would lead to the destabilization of the vehicle.

Driving will become unsafe and uncomfortable. Behavior of the spring-damper package is designed depending on several parameters: the weight of the vehicle, driving conditions, gauge, axle spacing, the comfort that you need to meet this class of vehicle. In order to increase safety in running vehicles, many researches have been conducted in the field, and also there were developed many applications that allow determining the response of the suspension to a specific external stimulus.

2. Description of Principle of Operation of the Damper

Behavior of the spring-damper package is designed depending on several parameters: the weight of the vehicle, driving conditions, gauge, axle spacing, the comfort that you need to meet this class of vehicle. In order to increase safety in running vehicles, many researches have been conducted in the field, and also there were developed many applications that allow determining the response of the suspension to a specific external stimulus.

In Figure 1 is represented schematic diagram of the hydraulic damper telescopic-pipe. In the course of detente, the liquid from the top of the piston is compressed and sent the rebound valve on the bottom.

The value generated by the piston at the bottom is greater than the volume of liquid pushed down, the rod coming out of the tube volume. The difference is added to the liquid in the compression chamber (the space between the tube and the inner tube tank) that penetrates through the intake valve. This occurs due to the depression created in piston and air cushion in the upper chamber of compensation.

In the compression stroke, under reciprocating fluid communication passes through the valve at the top of the tube. A part of the liquid (equal to the volume rod inserted into the tube) through the compression valve in compensation chamber. The tube is used to protect the piston rod and the ring to seal the piston.

Calibrated orifice that controls the viscous resistance of the liquid to drain its slits are made in the form of disks or directly in piston valves.

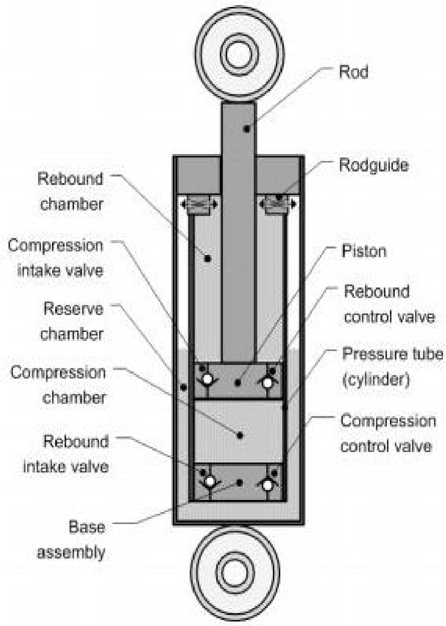

Fig. 1. *Conceptual design for a double chamber damper* [2]

To simplify the operation of the damper can define four modes: low speed and high speed piston plunger for each stage compression or rebound. In low speed flow regime due to slow movement of the piston, the fluid flow is not large enough to force the valve open clearing. Thus, at the relaxation phase, all the fluid flows through the piston, and through the valve openings compensation.

At the stage of compression due to the shutter plate located on the piston, some of the holes are blocked, leading to reduced surface hat route. The second case is when high-speed, where the compression phase of the compensation valve is open. The relaxation phase, it allows the passage of large quantities of fluid, due to the increased surface area of passage.

3. Method of Study

The four operating modes of shock, varies depending on the remoteness or proximity to the body of the wheel, and the compression of the damper or constructive relaxation natural elements. Professional code ANSYS CFX is useful for studies in which it wants to capture the effect of changing the damping coefficient to change the direction of travel of the piston. Consider the case of a low speed travel of the piston, when the critical speed is not exceeded, so the compression valve passes only the amount of liquid required volume compensation piston rod. This study being necessary for a properly sized hole in the piston or rings calibrated. For the simulation model was used silencer in 3D, which is represented in Figure 2.

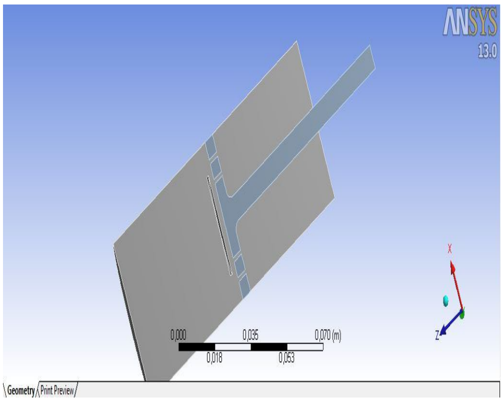

Fig. 2. *Definition of the interest*

Present studies, involves the monitoring of the continuous variation in terms of pressure (i.e. force resisting) related to the relative positioning of the piston (depending on the specific down force). This sequential analysis of flow conditions (at the certain changing interstitial fluid compressibility) is required by the combined effect with the behavior of the spring spacer. This spring runoff opposing fluctuating demands, continuous modify the position of a pilot pill floating in relative position more or less close (eights) to the holes manufactured to pass the front face of the piston. To simplify the simulation, the piston was considered fixed and floating ring is considered mobile, which is pressed by a spring piston with an elasticity coefficient k_a.

In sectioning plane of the piston are present only four 1 mm diameter holes, two of which are closed by floating ring, where it is in contact with the piston. The mash was performed following areas of interest, i.e. the interstitial space between the holes in the piston and floating ring to highlight the fluid dynamics and pressure of the field and hit on him, forcing the piston on the distance. All this is happening because the floating ring moves relative to the piston, and it was necessary to use a dynamic mesh witch are represented in Figure 3 and Figure 4.

The dynamic mesh model uses the ANSYS FLUENT solver to move boundaries and/or objects, and to adjust the mesh accordingly. The dynamic mesh model is used when boundaries move rigidly (linear or rotating) with respect to each other With respect to dynamic meshes, the integral form of the conservation equation for a general scalar, ϕ, on an arbitrary control volume V, whose boundary is moving can be written as:

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_V \rho\phi\,\mathrm{d}V + \int_{\partial V}\rho\phi(\vec{u}-\overrightarrow{u_g})\,\mathrm{d}\vec{A} = \int_{\partial V}\Gamma\nabla\phi\,\mathrm{d}\vec{A} + \int_V S_\phi\,\mathrm{d}V\,, \tag{1}$$

where ρ is the fluid density, $\vec{v}$ - the flow velocity vector, $\overrightarrow{ug}$ - the mesh velocity of the moving mesh, Γ - the diffusion coefficient, S_ϕ - the source term of ϕ.

Here ∂V is used to represent the boundary of the control volume V.

The time derivative term in Equation (1) can be written, using a first-order backward difference formula, as follows:

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_V \rho\phi \,\mathrm{d}V = \frac{(\rho\phi V)^{n+1} - (\rho\phi V)^n}{\Delta t}, \quad (2)$$

where n and n+1 denote the respective quantity at the current and next time level.

The (n + 1) the time level volume V^{n+1} is computed from:

$$V^{n+1} = V^n + \frac{\mathrm{d}V}{\mathrm{d}t}\Delta t, \quad (3)$$

where $\frac{\mathrm{d}V}{\mathrm{d}t}$ is the volume time derivative of the control volume.

In order to satisfy the mesh conservation law, the volume time derivative of the control volume is computed:

$$\frac{\mathrm{d}V}{\mathrm{d}t} = \int_{\partial V} \overrightarrow{u_g} \cdot \mathrm{d}\vec{A} = \sum_j^{nf} \overrightarrow{u_{gj}} \cdot \overrightarrow{A_j}, \quad (4)$$

where nf is the number of faces on the control volume and $\overrightarrow{A_j}$ is the j face area vector.

The dot product $\overrightarrow{ug \cdot j} \cdot \overrightarrow{A_j}$ on each control volume face is calculated from:

$$\overrightarrow{u_{gj}} \cdot \overrightarrow{A_j} = \frac{\rho V_j}{\Delta t}, \quad (5)$$

where ρV_j is the volume swept out by the control volume face j over the time step Δt [5].

In the case of the sliding mesh, the motion of moving zones is tracked relative to the stationary frame. Therefore, no moving reference frames are attached to the computational domain, simplifying the flux transfers across the interfaces. In the sliding mesh formulation, the control volume remains constant, therefore from Equation (3), $\frac{dV}{dt}$ and $V^{n+1} = V^n$.

Equation (2) can now be expressed as follows:

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_V \rho\phi \,\mathrm{d}V = \frac{[(\rho\phi)^{n+1} - (\rho\phi)^n]V}{\Delta t}. \quad (6)$$

4. Problem Description

In this problem it has been used - modeling a moving wall on a tank. The assumption that the wall motion is known will be made and supplied to the CFX in a comma separated value (CSV) format *Set-up*. The geometry was generated with two bodies combined in one part. The one domain, hereafter called the port, is the domain where the mesh is going to be deformed because of the moving wall.

The other domain is the tank to which the fluid is being ejected. The mesh in this region will not be deformed. So we move along to opening the mesh file in CFX and we begin by changing it over to a transient run. The next step would be to opening the Default Domain and in the panel change the Mesh Deformation option to Regions of Motion Specified.

The next step is to create a sub-domain for the port region under the Default Domain. In the sub-domain panel, select the port region for the location and move over to the Mesh Motion tab. It was used a specified mesh motion using ccl. In the current case the motion is in the z-direction, specifying a name of the ccl expression Mesh Motion which was define next. A key point is going to be used is that we want to compress the mesh in the entire domain evenly to maintain the best quality mesh we can.

Defining a temporal functions from CSV file

Assuming that it is known the movement of the wall, we are going to read it in using a CSV file.

Then the data should be a function of time. We import it as a spatial variable. We have changed it over when we define our cell expressions.

The next step is to change the spatial function into a temporal one. We have done this by creating an expression called Mesh Deformation. We have then defined this as Specified Motion. Displacement (t * 1 m / 1 s) * Stroke Distance. Note we will define Stroke Distance later.

First we generated a function that should be used to make sure we compress the entire sub-domain evenly. We do this by generating a user-function we will call Interpolation Location. We put unit of [m] in the Argument Units and for the Resulting Units. For the one-dimensional function we will supply the data pairs 0, 0 and 4, 1. We do this because the port mesh at 0 [m] will not be deformed and the port mesh at 4 m will deform the full amount we will specify (my port is 4 m long).

Next we create our Mesh Motion expression. For this we define it at Mesh Deformation * Interpolation Location (z-Total Mesh Displacement Z). Note the Interpolation Location is the function we just defined and Total Mesh Displacement Z is the predefined expression that outputs the total mesh displacement in the z-direction relative to the initial mesh. We defined the Mesh Deformation expression earlier.

The final expression we need to define is the Stroke Distance. We simply define this through a ccl expression to be –4 m. The negative sign indicates that displacement will be in the –Z direction.

In order to perform simulations there were fixed the conditions of motion of the piston (i.e. input and output constrains) as an alternating movements of the fluid being sinusoidal velocity variation with a π phase shift between input and output (see Figure 5).

In the simulations were performed several sets of initial data to see the variation of the flow rate in areas of interest and shutter blade movement under the action of fluid pressure (see Figure 6).

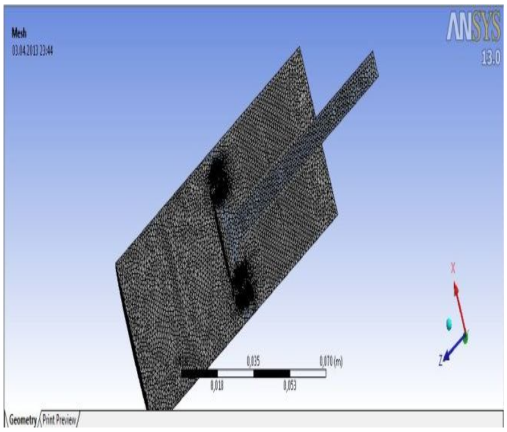

Fig. 3. *Generic mesh*

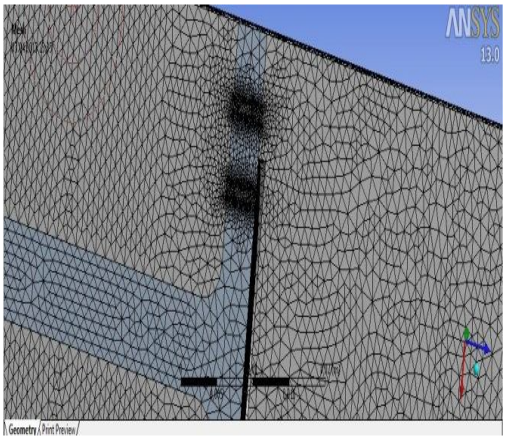

Fig. 4. *Mesh detail*

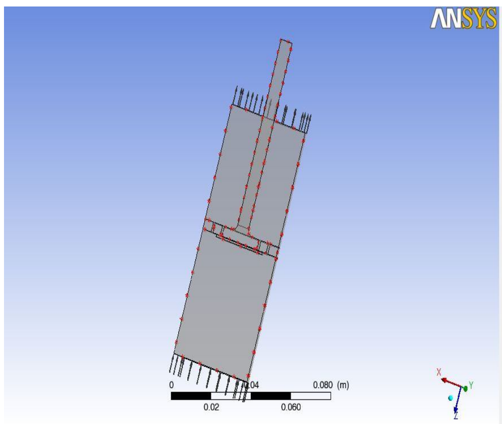

Fig. 5. *Simulation settings*

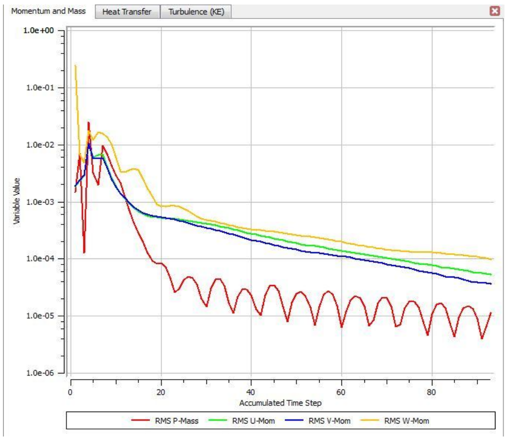

Fig. 6. *Root mean square*

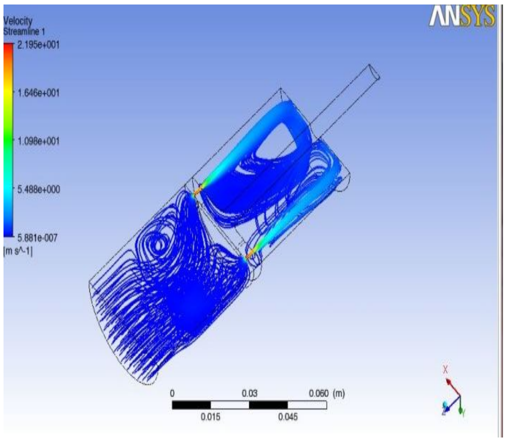

Fig. 7. *Speed spectrum*

5. Conclusions

The design of the suspension should be taken into account the need to maintain contact with the road surface. Damper must face the relatively high frequency excitations and relatively small amplitude, excitations from the type/quality of runway roughness. Also suspension must face MORE promptly to such stress in steps (impact). On the other hand, the quality of the suspension is appreciated through the way in which attenuates vibrations from body aerodynamics effect when running at high speeds.

After analyzing the results it was observed that for small displacements of the piston and due viscosity working fluid, solid shutter delay moves to the piston design which imposes a fine passage of surface area in relation to surface area is covered by slide the compression phase (see Figures 7-9). This directly implies the significant improvement of the ride comfort and the steering stability of the vehicle system.

The results presented in this work are quite self-explanatory justifying that the proposed control methodology is very effective for vibration isolation of the suspension system subjected to parameter variations and external disturbances.

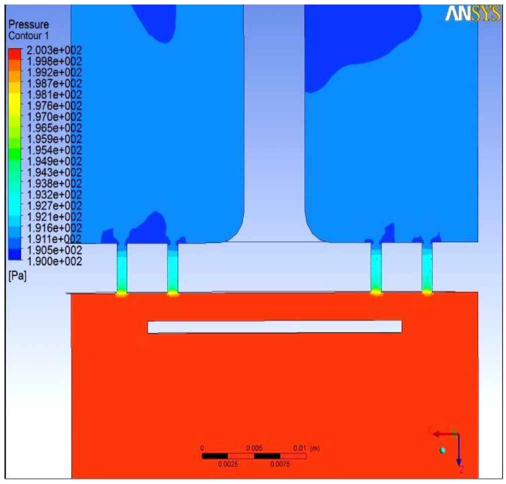

Fig. 8. *Pressures spectrum*

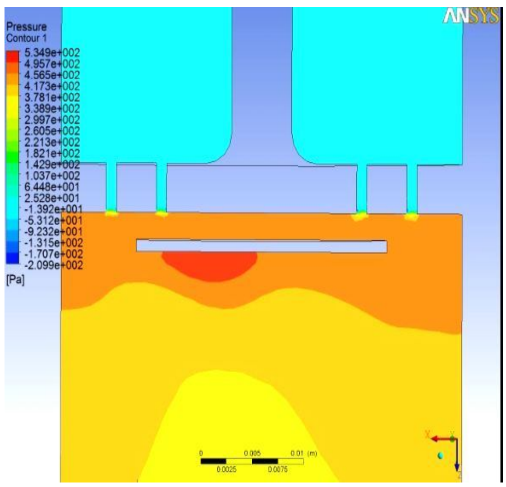

Fig. 9. *Pressures spectrum*

Acknowledgments

Authors gratefully acknowledge the financial support of the National PN-II-PT-PCCA 2011-3.2.0512, 2012, 2015 for the "EQUATOR" "Advanced strategies for high performance indoor Environmental Quality in Operating Rooms".

References

1. Canale, M., Milanese, M., et al.: *Semictive Suspension Control using 'Fast' Model Predictive Control.* In: American Control Conference, Portland, OR, USA, June 8-10, 2005.
2. Czop, P. Sliwa, P.: *A Computational Fluid Flow Analysis of a Disc Valve System.* In: Journal of KONES Power Train and Transport **18** (2011) No. 1, p. 118.
3. Guzzomi, F.G., O'Neill, P.L., Tavner, A.C.R.: *Investigation of Damper Valve Dynamics Using Parametric Numerical Methods*. In: 16th Australasian Fluid Mechanics Conference Crown Plaza, Gold Coast, Australia, 2-7 December, 2007, p. 1123.
4. Herr, F., Mallin, T., Lane, J., Roth, S.A.: *Shock Absorber Model Using CFD Analysis and Easy 5.* In: SAE Technical Paper 1999-01-1322, 1999.
5. *Manualul* (*tutoriale CFD*) *Fluent*, Ansys CFX.