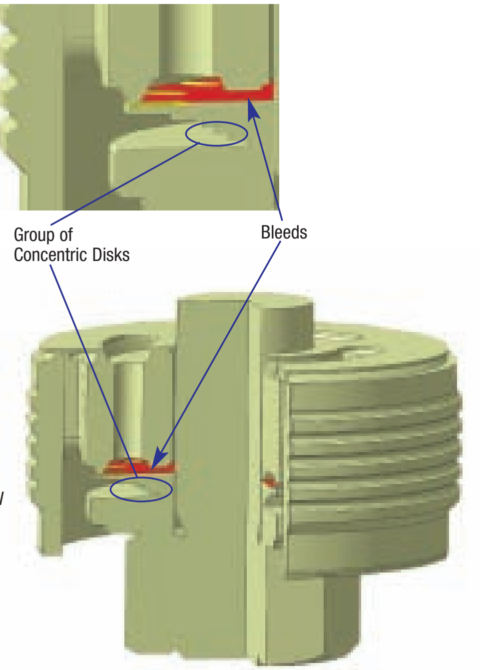
Decreasing the Shock in **Shock Absorbers**



Geometry of the shock absorber

Engineering simulation improves valve design for an automotive shock absorber.

By Marcelo Kruger, Geraldo Severi, Martin Kessler and Regis Ataides, ESSS, Florianópolis, Brazil
Sergio Vanucci, Pedro Barau and Robson Iezzo, Magneti Marelli Cofap, Campinas, Brazil

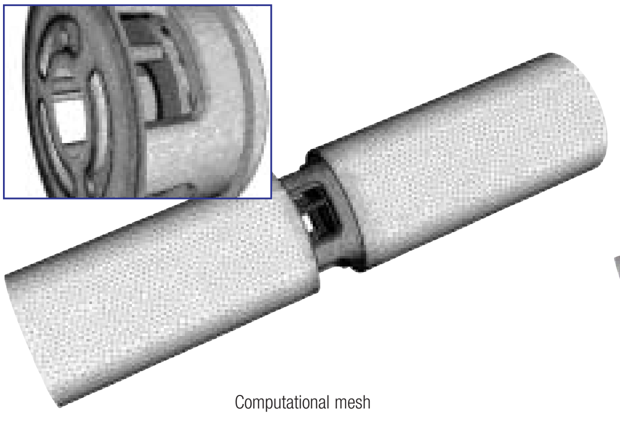
Automotive design requires consideration of a large number of factors including passenger comfort. Suspension design — more specifically, shock absorbers — requires special attention from engineers in order to improve ride quality. Without shock absorbers, a vehicle would have an uncomfortable jolting motion, as energy stored in the spring is released to the vehicle. Shock absorbers dampen spring vibrations by turning the kinetic energy from the springs into heat that is dissipated through a hydraulic fluid.

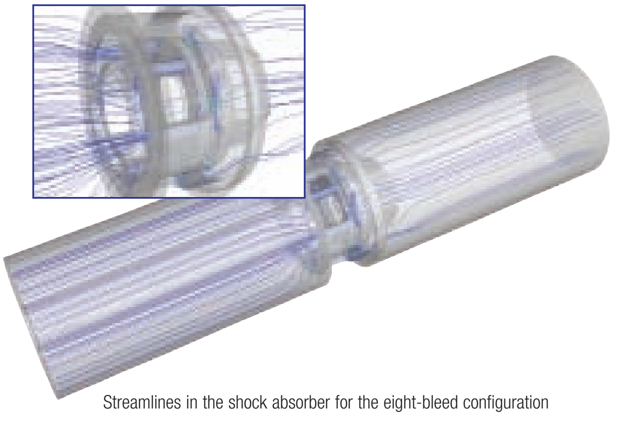
Engineers at Magneti Marelli Cofap in São Paulo, Brazil — a division of the international company that is committed to the design and production of high-tech systems and components for the automotive sector — have been developing a shock absorber system that is controlled mainly by valves employing circular disks. For low velocities, the fluid passes through small bleeds, or holes, on the disks, with the passage area defined by the number of bleeds. For high-velocity conditions, a different method is used.

In order to improve the shock absorber design and obtain a better understanding of hydraulic fluid flow, engineers from both Magneti Marelli Cofap and Engineering Simulation and Scientific Software (ESSS), an ANSYS channel partner in South America, collaborated on an engineering simulation analysis. The team built a computational model of the shock absorber for the low-velocity condition in order to understand the flow physics that occur in the valve, to evaluate dispersion of forces at the disks and to compare the results against experimental data.

To evaluate the flow pattern and measure the forces, the engineering team used ANSYS ICEM CFD meshing software to generate a computational mesh consisting of tetrahedral and prisms elements and containing 1 million nodes. Simulation of the steady-state flow was performed using the k-epsilon turbulence model in ANSYS CFX fluid flow software. The team considered three configurations of bleeds (two, eight and 16 bleeds) under three velocity conditions for each configuration at the inlet. They obtained the pressure field on the regions of interest at the valve using the CFX-Post post-processor and generated a plot of force (at the valve) versus velocity. The results showed very good agreement with experimental data.

The reliability of the results has given Magneti Marelli Cofap engineers increased confidence in the computational model and, by comparing the different configurations, has provided them with a better understanding of the complexity of the flow pattern. Simulation has imparted useful insights that the engineers now rely upon to make important decisions concerning the improvement of shock absorber efficiency. ■


Computational mesh


Streamlines in the shock absorber for the eight-bleed configuration