Sigma Journal of Engineering and Natural Sciences
Web page info: https://sigma.yildiz.edu.tr
DOI: 10.14744/sigma.2026.2093

Review Article

Flow visualisation techniques for enhanced vehicle damper performance

Kamalakkannan KASI[1], Aadil Arshad FERHATH[1,*]

[1]*SRM Institute of Science and Technology, R2FV+6Q7, Potheri, SRM Nagar, Kattankulathur, Tamil Nadu, 603203, India*

ARTICLE INFO

Article history
Received: 15 August 2024
Revised: 03 October 2024
Accepted: 04 November 2024

Keywords:
Damper Performance; Flow Patterns; Fluid Flow Analysis; Visualisation Techniques

ABSTRACT

Vehicle dampers are fundamental components in vehicle engineering and are of great importance for ride comfort, handling and safety. Proper flow control within these dampers is vital to optimize vehicle performance and passenger comfort. Transparent damper models consisting of light transmitting materials facilitate direct visualisation of flow patterns of fluid. This is crucial for understanding how these systems work, under different conditions. The dye injection technique is based on the injection of coloured or fluorescent dyes in the damping fluid. When combined with high speed photography, this technology allows for detailed recording of transient events during compression and rebound cycles. Real-time response of the damper is captured through high-speed cameras that record the rapid changes in fluid dynamics, providing crucial data. The flow speed and turbulence inside the damper are determined and characterized by particle-tracking techniques.

Cite this article as: Kasi K, Ferhath AA. Flow visualisation techniques for enhanced vehicle damper performance. Sigma J Eng Nat Sci 2026;44(3):2249–2262.

INTRODUCTION

In automotive engineering, the pursuit of a smooth, safe and comfortable ride has been a major driver of progress and breakthroughs [1,2]. Dampers are essential for the proper control of the dynamic interaction between the vehicle suspension system and the road surface [3,4]. The main function of these devices is to absorb and dissipate efficiently the kinetic energy that is generated by the road's irregularities and the driver's actions [5]. Damping systems are used to reduce vibrations, oscillations and disturbances which can affect the driving comfort, the handling stability and the passenger safety [6,7]. These simple looking cylinders are operated on the basis of the complex field of fluid mechanics. Car dampers require accurate fluid control as a function of load and velocity to achieve damping characteristics. Understanding and improving the fluid flow is the fundamental basis for developing dampers with outstanding performance and ride quality [8].

Fluid flow analysis is the systematic study of the flow of liquids or gases in a given region. [9] It consists in analyzing the dynamics of the flow of the damping fluid in the damper during the compression and rebound cycles. The analysis is aimed at elucidating the fluid dynamics by identifying factors influencing damper performance, such as pressure distribution, heat generation, turbulence, and

***Corresponding author.**
*E-mail address: *aadilarshadsrmist*@gmail.com

This paper was recommended for publication in revised form by Editor-in-Chief Ahmet Selim Dalkilic

Published by Yıldız Technical University Press, İstanbul, Turkey

cavitation [10]. Flow patterns are the motion and interaction of the damping fluid with different internal parts like the piston, orifices and reservoir. The flow direction and the circulation of the fluid directly affect the ability of the damper to control the vibration and keep the stability of the vehicle [11]. The effectiveness of a damper is determined by several key factors. These include its ability to efficiently damp vibrations, control heat, resist cavitation and absorb energy from uneven road surfaces. Adequate fluid flow and heat transfer are crucial for the consistency of the damping behaviour and durability of the damper [12,13].

Automotive engineers and researchers have access to a range of experimental and computational techniques known as flow visualization methods to examine the complexities of fluid dynamics in key automotive components [14]. The range of visualisation techniques is vast, from transparent damper models and dye injection for qualitative observation to more advanced technologies such as high speed photography, particle tracking, pressure sensors, thermal imaging and computational fluid dynamics (CFD) for qualitative and quantitative analysis [15]. This study investigates in detail a number of techniques for analysing and visualising fluid flow, in order to gain better insight into car damper performance. The methods can be used to observe, quantify and analyse fluid flow patterns, turbulence, pressure variations and temperature distributions in dampers [16]. In this study, we will explore the importance of these approaches to shed light on the intricacies of car damper dynamics by analysing their practical implementation and evaluating their value. Moreover, we will explain how these observations can be translated into better damper designs which in turn will result in better cars in terms of driving experience as well as passenger well-being and satisfaction. Figure 1 illustrates the interaction between light waves and fluid dynamics.

Transparent Damper Models

Transparent damper models, also called see-through or cutaway damper models, are physical prototypes of vehicle dampers which are made from casing materials with transparency properties, e.g. acrylic or glass [17]. Specialized damper models are useful tools for research and education purposes. Engineers, students and hobbyists can directly see and understand the internal mechanisms of dampers with them. A careful attention is paid to the building of transparent damper models, which are supposed to faithfully imitate the external structure of real dampers [18]. These models are intended to include the transparent materials for the casing. Transparent materials facilitate the visualization of the interior constituents and fluid flow. These models are often used in educational settings, especially in automotive engineering programs and training workshops. Visual aids help students and technicians in understanding the working of dampers and basic principles of fluid dynamics. The transparent damper models are mainly to visualize the dynamic behavior of the damping fluid during the compression and rebound strokes. The dynamic behaviour of the fluid can be visually observed using a suitably coloured or fluorescent damping fluid, in terms of movement and circulation and the response to the change in the position of the damper [19]. Summary of flow visualisation techniques in vehicle damper analysis is shown in Table 1.

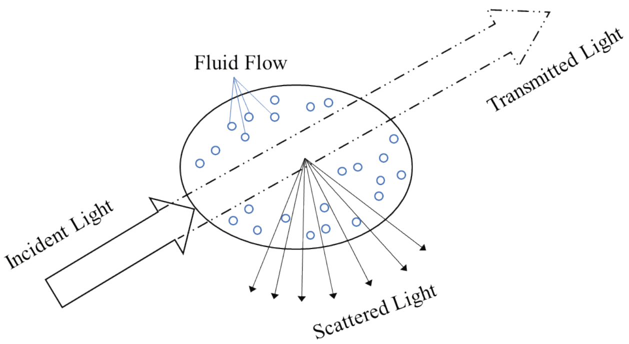

Figure 1. Interplay between light wave and fluid flow.

Transparent damper models serve as helpful teaching tools for explaining the concepts of damping rate, rebound damping and compression damping. These tools can be used by educators to show the effect of fluid flow on the ride quality and handling of a vehicle. Clear damper models are used by engineers and academics to study the fluid dynamics of dampers in detail. These models allow researchers to investigate the influence of design parameters and fluid variables on the damping performance. It is useful for the improvement of damper design and development of liquid formulations [24]. Transparent damper models are useful to identify and evaluate defects and irregularities of dampers. The behaviour of the damping fluid can be observed in order to identify difficulties such as cavitation, air entrainment or leakage [25]. The experimental configuration for evaluating passive dampers is depicted in Figure 2 (a), showing the layout of the experimental apparatus and the method utilised to simulate road conditions, as presented in Figure 2 (b).

The automotive industry also uses these models in public demonstrations and exhibits to highlight the technology and engineering behind vehicle suspension systems. Transparent damper models are often used in conjunction with computational simulations such as Computational Fluid Dynamics (CFD) to verify the fidelity of the computational models [26]. The comparison of empirical observations in the real world with simulated data validates the models' accuracy and representativeness in capturing dampers' behaviour. The transparent damper models are valuable tools for teaching, research and development in automotive engineering. Vehicle dampers provide a concrete approach to understand and improve the fluid dynamics inside them, which in turn improves the vehicle ride quality and performance. The automobile features a control mechanism that employs a magnetorheological (MR) damper Figure 3.

Table 1. Summary of flow visualisation techniques in vehicle damper analysis

Flow Visualisation Techniques	Advantages	Disadvantages
Transparent Damper Models	Visual investigation of the behaviour of fluid flow in real-time.	Restrictions apply only to qualitative observations
	Appropriate for use in instructional contexts	It may not accurately depict intricate interior details or intricate flow patterns.
Dye Injection	Real-time representation of the movement of fluids, formation of vapour bubbles, and blending of substances	It is restricted to subjective observations.
	It has useful and accurate information on fluid flow dynamics.	Disruptive and can change the smoothness of movement.
High-Speed Photography	Records instantaneous events and the flow patterns of liquids.	Restricted to the visible spectrum of light
	It gives numerical information on the speeds of the particles	Shadows and interference patterns may appear
Particle Tracking	Quantitative examination of particle speeds and paths.	It only allows tracking of particles, not fluid characteristics.
	Provides a deep understanding of the details of flow patterns	Difficult to apply to complex flow systems.
Pressure Sensors	Gives numerical information on distribution of pressure.	Limited to collecting pressure data only, without the ability to see flow.
	Shows Cavitation, Fluctuating Pressures and Overheating.	Lacks the ability to observe flow patterns directly.
Thermal Imaging	Displays the distribution of the temperature in space and zones of high temperature	Provides implicit information about the movement of the fluid.
	Identifies excessive heat and thermal issues.	Constrained by phenomena directly associated with temperature
Computational Fluid Dynamics (CFD)	Quantitative study of fluid motion and behavior.	Requires computational resources and specialised knowledge
	Predicts fluid flow, pressure distribution and temperature profile.	Depends on presuppositions and oversimplifications

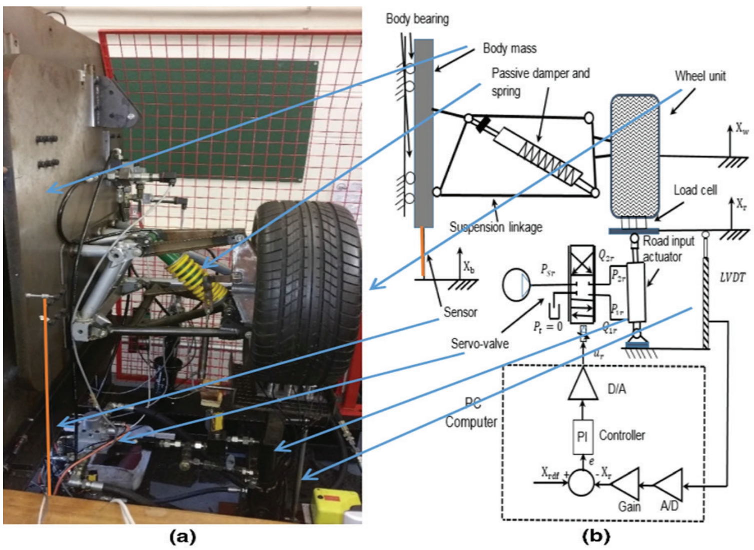

Figure 2(a). Experimental setup for testing passive dampers **(b).** Illustration showing the arrangement of the experimental equipment and the mechanism employed to replicate road conditions [From Al-Zughaibi [22], under the terms of the Creative Commons Attribution 4.0 International License]

Figure 3. The vehicle is equipped with a control system that utilises a magnetorheological (MR) damper. [From Grzesikiewicz et al. [23], under the terms of the Creative Commons Attribution (CC BY) license.]

Dye Injection

Dye injection technique is a widely used method for the investigation of fluid dynamics and engineering to study the fluid behaviour in different systems. This technique consists of the addition of pigmented or fluorescent dyes to the fluid under study which allows the visual observation and evaluation of the flow patterns, the turbulence and the properties of the fluid [27]. The primary purpose of dye injection is to improve the visibility and observability of fluid flow patterns. The movement, mixing and behaviour of fluid dynamics inside a particular system can be visualised by adding a dye to the fluid, allowing researchers to study and analyse [28]. The dye injection process involves the use of pigmented and fluorescent dyes. These are coloured dyes. They add pigment, changing the way the fluid looks . Fluorescent dyes are dyes that emit visible light when illuminated at certain wavelengths. Fluorescent dyes are especially useful when researchers want to study flow patterns in low light conditions or where high accuracy is required.

The use of dye injection is widespread in various fields such as fluid mechanics, aerodynamics, chemical engineering and environmental sciences. This method is concerned with the flow behaviour of liquids in pipes, tanks, channels and other systems containing liquids. In automotive engineering, dye injection is used as a technique to study and analyse the airflow patterns around cars or within specific components such as radiators and heat exchangers [30]. Once the dye is injected into the fluid, researchers can observe and record the flow patterns using a number of visualisation techniques. The characteristics of the boundary layer over (a) flat and (b) undulating surfaces are shown in Figure 4.

Under appropriate lighting conditions, a number of techniques can be employed including high speed photography, video recording or direct observation. Qualitative observation of fluid motion patterns can be performed by dye injection [31]. Through meticulous adjustment of dye concentration and the use of appropriate imaging apparatus, researchers can conduct quantitative analyses, including measuring flow velocities, tracking vortices, and evaluating turbulence levels. When performing dye experiments, it is important to adhere to safety measures to guarantee the use of non-toxic dyes that are compatible with the fluid being studied.

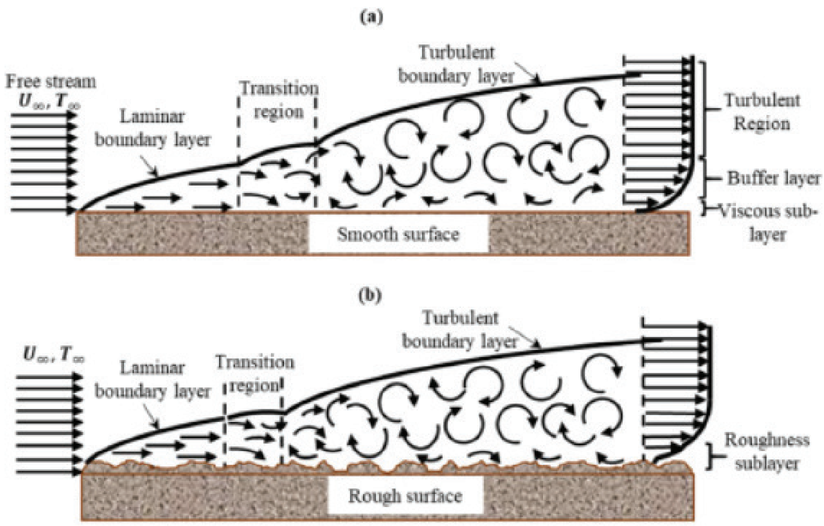

Figure 4. The behaviour of the boundary layer over (a) flat and (b) undulating surfaces. [From Kadivar et al. [29], under the terms of the Creative Commons Attribution 4.0 International License.]

In addition, it is important to avoid any form of contamination of the experimental environment . The use of dye injection for flow visualization has greatly increased with improvements in imaging technology and computer-assisted analysis [32]. Dye-injection experiments provide visual data from which detailed information is often extracted by digital image processing and analysis techniques. Dye injection is a useful methodology for academics and engineers wishing to gain an improved understanding of fluid flow behaviour. Dye injection is a useful tool for studying and estimating the complex dynamics of fluid motion in a variety of fields, such as industrial processes, environmental systems and engineering applications [33].

High-Speed Photography

High speed photography is a specialised technique which involves taking images at an extremely high rate, beyond the ability of normal cameras. This technique is used to record and analyse fast or transient events within a short time interval [34]. This method is applied in many areas, for example in science, engineering, sports analysis and art. High-speed photography is the process of capturing images at much higher frame rates than normal video or film cameras, typically in the range of several hundred to several thousand frames per second (fps) [35]. Modern systems are capable of recording at frame rates of millions of frames per second, capturing very fast phenomena. The application of this technique enables the fast investigation of dynamic physical phenomena, like chemical reactions, explosions, fluid dynamics and others [36]. It helps to study mechanical system dynamics, effect of forces and fluid flow behaviour in a range of practical situations. This instrument is used to assess athletes' performance including the analysis of sports actions such as golf swings and ball impacts in relation to their mechanics [37]. High-speed photography can capture extraordinary, visually captivating moments such as the dynamic motion of liquids dispersing on impact, the rapid rupture of balloons or the forceful collision of objects. High-speed photography is a practice that requires the use of specialised equipment, such as high-speed cameras, robust lighting arrangements and accurate triggering systems [38].

These components are designed to work together. In this way they can capture fast paced events with impressive accuracy and efficacy. For high speed photography, the presence of a suitable illumination is of the utmost importance owing to the very small exposure durations, leading to the use of highly concentrated light emissions [39]. Strobe lights are often used to produce controlled, even lighting. "Precise triggering is important to catch the precise time of interest. Various techniques such as manual activation, auditory sensors, motion-sensing devices, or laser-based activation mechanisms can be employed to attain this goal. After acquisition of high speed photographs, researchers and photographers often use specialist software for analysis and data processing [40]. This process can include quantification of distances, velocities, angles and sequences frame by frame to understand the dynamics of the phenomenon.

High-speed photography is essential in scientific investigations to observe phenomena that happen at speeds so fast that they cannot be seen with the naked eye [41]. This technique has been applied in several studies in fluid dynamics, ballistics, impact physics and related disciplines. High-speed photography has been used not just for scientific and technical purposes, but also by artists because it can capture unique and visually appealing moments [42]. Freezing a split-second in time can reveal intricate details that are often overlooked in real time. High speed photography is a very useful technique for the study and recording of transient events [43]. It's a portal to the dynamic world of fast events, whether that's for scientific exploration, engineering evaluation or artistic representation.

Particle Tracking

Particle tracking is a widely applicable methodology that is used in many different scientific, engineering, and research fields to observe and analyze the motion of particles or objects within a fluid or medium [44]. The method consists of following the spatial coordinates of individual particles over a certain period of time and is a powerful tool to gain information about fluid dynamics, flow characteristics and other phenomena. Particle tracking is used to obtain a detailed knowledge of the motion of particles in a fluid or medium [45]. This tool enables the investigation of several parameters such as velocity, trajectory, diffusion, and concentration of particles inside dynamic environments which allow the researchers to conduct comprehensive analyses [46]. Fluid dynamics studies employ techniques such as flow pattern studies, turbulent phenomena and mixing processes. Tracking of particles is a common technique in biology, where biologists use it to study movement patterns of cells, organelles and biomolecules in different biological systems [47]. This phenomenon gives a better understanding of chemical processes by showing reactions, diffusion and transport.

Tracking particles is a common technique used in environmental science. Researchers use it to study the spread of pollution, the transport of sediment and the quality of water. In many cases it is common practice [48] to add small particles or tracers, e.g. fluorescent microspheres or neutrally buoyant particles, to the fluid being studied. Particle tracking is a method that employs imaging equipment to record and evaluate particle movement. The main imaging methods are bright field microscopy, fluorescence microscopy and digital cameras . High-speed cameras [49] are often employed to capture the fast motion of particles. The analysis of images and the tracking of particles are supported by specialized software and algorithms. The algorithms can detect and track the exact locations of individual particles in successive frames automatically, enabling the collection of accurate and reliable data [50]. Particle tracking is a technique used

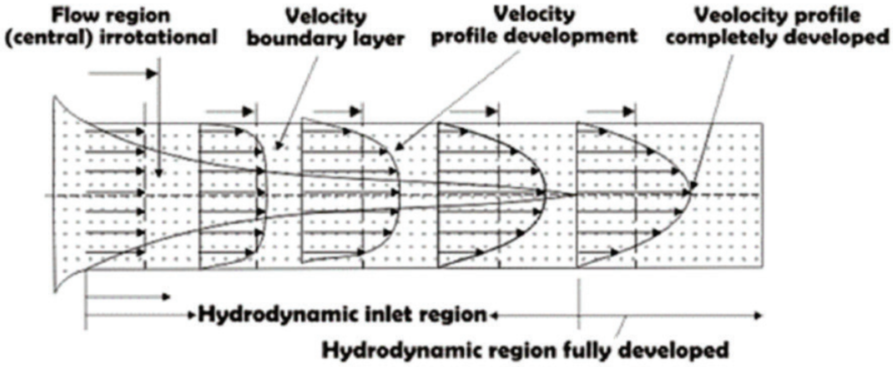

Figure 5. Variable input lengths and velocity profile evolution. [From Iturbide Jiménez F et al. [51], under the terms of the Creative Commons Attribution 4.0 International License.]

for quantitative investigation of fluid flow parameters such as particle velocities, accelerations and trajectories. The data can be used to calculate distributions of fluid velocity and to investigate properties of turbulence.

Particle tracking is crucial for understanding cellular and subcellular phenomena in biology and biophysics. This technique facilitates the quantification of cellular components and proteins, providing insights into fundamental biological systems. The task of particle tracking can be challenging in settings characterised by intricate flow patterns, high particle densities, or particles with poor contrast [52]. Resolution of these issues frequently necessitates using cutting-edge imaging methodologies and intricate tracking algorithms. Advances in image technology and computational skills have improved particle tracking capabilities. The utilisation of three-dimensional (3D) particle tracking and multi-particle tracking techniques has become increasingly prevalent in recent years, offering enhanced precision and a more comprehensive understanding across several fields of study. The study of particle dynamics in fluid and non-fluid systems can be significantly aided by particle tracking [53]. This phenomenon has implications in various scientific fields, improving our understanding of fluid dynamics, biological mechanisms, and environmental occurrences. Because of technological advances, the field of particle tracking is constantly expanding. As a result, researchers and engineers have access to more exact and comprehensive data. [54]. Figure 5 shows the evolution of variable input lengths and velocity profiles.

Pressure Sensors

The pressure sensors are paramount for viewing and analysing flow inside the vehicle dampers. These sensors enable the observation and comprehension of fluid dynamics within dampers by measuring pressure fluctuations at different points in the system. Pressure sensors are precisely positioned inside the damper such as intake and output ports, piston chamber and reservoir [55]. The sensors provide real-time data on the pressure distribution inside the damper during its compression and rebound strokes. Cavitation is formation and collapse of vapour bubbles in the damping fluid and its occurrence has a significant effect on the overall performance of dampers. Pressure sensors are used to detect changes in pressure that may indicate cavitation effects [56]. The provided information is sufficient for optimisation of damper design to minimize or eliminate cavitation. This can lead to reduced dampening effectiveness and structural damage.

By combining pressure sensors with various flow visualization methods, turbulence inside the damper can be analyzed. Turbulence may have negative effects on the damping efficiency of a system and deterioration of components. The pressure sensors are important for quantifying the pressure fluctuations that are associated with turbulence and hence for assessing the level of turbulence [57]. Validation and improvement of numerical models is possible through the use of experimental data from pressure sensors [58]. This way of working ensures that the simulation of the damping fluid with computational fluid dynamics (CFD) is a faithful reproduction of its real behaviour. Changes in pressure can be used to measure variations in fluid temperature, which is a significant factor in the study of heat generation and removal processes [59].

Monitoring thermal impacts is important to ensure the damper is operating within acceptable temperature limits. Through the examination of pressure data obtained from sensors, engineers can enhance the design of dampers to attain precise damping properties [60]. The orifice size, fluid properties and a number of parameters can be varied to optimize damping performance and improve overall ride comfort. Pressure sensors are very important for observing and studying the flow dynamics in car dampers and as such are valuable tools in this respect. Damping fluids are important for engineers and researchers to understand fluid dynamics. They are a good source of data to identify potential problems such as cavitation and turbulence. Moreover, these fluids contribute to the improvement of damper designs, resulting in enhanced performance and comfort in vehicles [61].

Thermal Imaging

Thermal imaging is a very useful tool for visualizing and analyzing flow patterns in automobile dampers. This methodology relies on an infrared (IR) camera to capture the temperature profile of surfaces and fluids, thereby providing engineers and researchers with important information about the thermal properties of dampers. The temperature fluctuations of the damper can be observed and analyzed by using thermal imaging [62]. The movement and interaction of the damping fluid with internal components during the compression and rebound cycles generates heat due to the friction and energy dissipation. Thermal imaging allows monitoring and quantification of temperature variations [63].

Cavitation is the formation and subsequent collapse of vapour bubbles in the damping fluid and can lead to localised temperature changes. Thermal imaging can be used to locate areas of the damper where cavitation is likely to occur. This is done by detecting temperature peaks associated with the collapse of bubbles. The thermal imaging technology produces a graphical representation of the spatial distribution of thermal energy in the damper [64]. This enables engineers to detect hot spots and temperature gradients, which helps them to see if the damper is operating within safe temperature limits. The increase in the temperature of damping fluids can reduce the efficiency of damping and deteriorate the components of dampers. [65]. Damper analysis key parameters and methods are illustrated in Table 2.

Thermal imaging is a useful technique used to detect areas of the damper that could be prone to overheating and to optimize the design in order to improve the heat dissipation [67]. The combination of thermal imaging with pressure sensors and other flow visualisation techniques allows full assessment of damper action. Engineers can then correlate thermal effects to fluid dynamics by combining temperature data with pressure and flow patterns. Thermal imaging can be used to evaluate the thermal efficiency of dampers in different operational conditions including different vehicle velocities, loads and road conditions [68]. The information provided is of great importance for the improvement of the design of dampers for practical applications. The thermal and flow field contours of the gaseous phase. The temperature and velocity profiles of the gas phase, without suspension injection (left) and with suspension injection (right), are depicted in Figure 6.

Table 2. Damper analysis key parameters and methods

Parameters	Techniques for Assessment
Fluid Flow Patterns	Transparent Damper Models Dye Injection High-Speed Photography Particle Tracking
Pressure Distribution	Pressure Sensors Computational Fluid Dynamics (CFD)
Temperature Profiles	Thermal Imaging Computational Fluid Dynamics (CFD)
Cavitation Detection	Pressure Sensors Transparent Damper Models
Heat Generation and Dissipation	Thermal Imaging Computational Fluid Dynamics (CFD)
Turbulence	High-Speed Photography Computational Fluid Dynamics (CFD)
Fluid-Structure Interaction	Computational Fluid Dynamics (CFD) Finite Element Analysis (FEA)
Impact Analysis	LS-DYNA

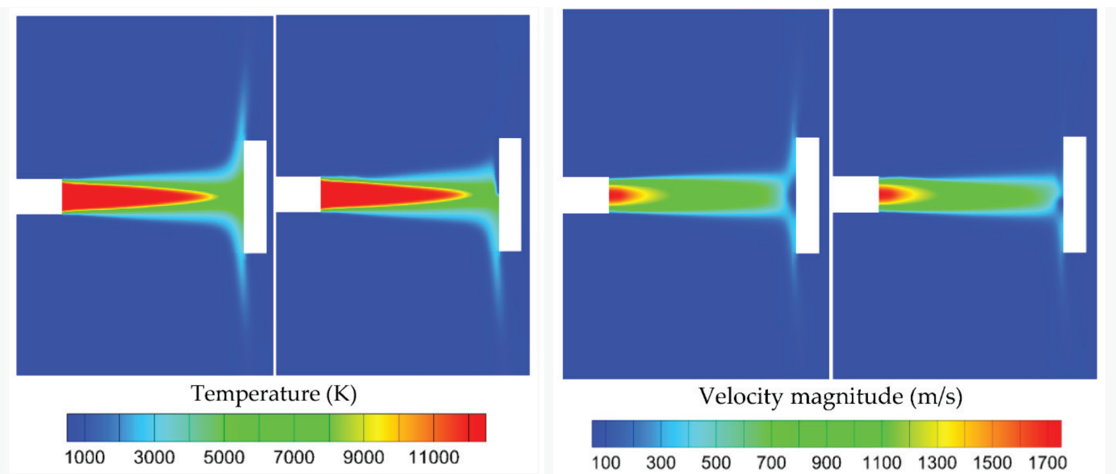

Figure 6. The temperature and velocity contours of the gas phase. without suspension injection (left) and with suspension injection involved (right). [From Jadidi M [66], under the terms of the Creative Commons Attribution 4.0 International License.]

In Computational Fluid Dynamics (CFD) simulations, it is common to integrate heat transfer models to forecast temperature changes in dampers [69]. Physical tests with thermal imaging data validate and complement numerical simulations, ensuring accuracy in the prediction of thermal behaviour. Thermal imaging is a useful technique for the view and analysis of the flow in vehicle dampers as it provides important information on the thermal characteristics of the damping fluid dynamics [70]. The data are important for the improvement of damper designs, for the identification of possible overheating and cavitation problems and for safe and efficient operation of car suspension systems.

Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is very useful tool to visualize and analyze fluid flow in automobile dampers. CFD simulations enable the modelling and analysis of damping fluids inside dampers providing valuable insight into fluid flow patterns, pressure distributions and many significant factors [72]. The initial step in carrying out the computational fluid dynamics (CFD) simulations is to build a three dimensional geometric model of the damper. This model includes both the external structure and internal parts of the damper, such as orifices, pistons, and fluid reservoirs. Accurate modelling of the geometry is essential to achieve meaningful simulation results [73]. The magnetic circuit's apex (the piston head) exhibits a significant concentration of magnetic flux, whereas the coil bobbins display a reduced flux concentration, with the predominant flux traversing the MR fluid area surrounding the magnetic poles of the piston. The input current is denoted as 1.0 A, whereas I is illustrated in Figure 7.

The geometric model is discretized into a mesh of cells or elements. Appropriate meshing techniques are of great importance for realistic representation in fluid dynamics. Engineers must find the optimal trade-off between the fine and coarse mesh resolutions [74] to achieve the computing efficiency with the accuracy. Explicit definitions of fluid parameters like density, viscosity and thermal conductivity are required for CFD simulations [75]. Car dampers usually use oil as damping fluid. The choice of the damping fluid plays an important role in the accuracy of simulations . The boundary conditions of the simulation, such as the input and output conditions, the movement of the piston, and other relevant characteristics are defined by engineers [76]. These are true representations of real-world scenarios and driving conditions. Software applications for flow visualisation of fluids in vehicle dampers is shown in Table 3.

A wide range of solvers and numerical methods are available for computational fluid dynamics (CFD) simulations. The choice depends on the damper arrangement and the level of precision desired [77]. The popular Computational Fluid Dynamics (CFD) solvers are finite volume, finite element and boundary element methods. Several simulation runs are carried out to evaluate a number of scenarios including variations in damper settings, piston speeds and fluid properties. The simulations provide useful insights into the effect of variations in these parameters on the fluid flow, pressure distribution and heat transfer in the damper [78]. Post-processing technologies analyse and visualise the outcomes of simulations. [79] Engineers can create contour plots, streamline visualisations and animations to understand the flow patterns, turbulence and pressure fluctuations that happen inside the damper.

Validation of computational fluid dynamics (CFD) simulations with experimental data is crucial for the evaluation of their accuracy and reliability [81]. The utilisation of an iterative process enhances damper design to achieve optimal performance. The comparison of simulation results with physical testing data is used to improve the confidence on the predictive capabilities of the computational fluid dynamics (CFD) model [82]. The application of Computational Fluid Dynamics (CFD) to simulate fluid dynamics has become a very effective way to study the flow inside vehicle dampers. These tools assist engineers in understanding complex fluid

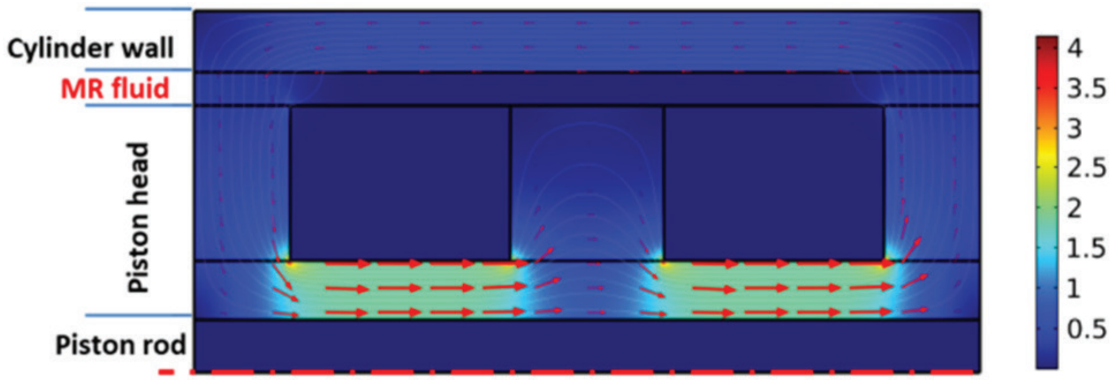

Figure 7. There is a high concentration of magnetic flux in the magnetic circuit's apex (the piston head), a lower flux concentration in the coil bobbins, and the majority of flux flowing over the MR fluid region around the piston's magnetic poles. 1.0 A is the input current, I. Measured in Tesla. [From Elsaady W et al [71], with permission from Elsevier.]

Table 3. Software applications for flow visualisation of fluids in vehicle dampers. [From Ferhath AA [80], with permission from Springer]

Softwares	Uses
ANSYS Fluent	Model and examine fluid movement, pressure distribution, and damper temperature profiles.
COMSOL Multiphysics	Investigate the dynamics of fluid-solid interactions and analyse the heat-transfer characteristics of dampers.
Open FOAM	Model fluid dynamics, encompassing intricate flow patterns, turbulence, and heat transfer within dampers.
SolidWorks Flow Simulation	Examine fluid flow and heat transfer in automobile dampers using the SolidWorks suite.
Autodesk CFD	Conduct a comprehensive examination of fluid movement, heat transfer, and pressure distribution in the various parts of the damper.
STAR-CCM+	Research the principles of fluid dynamics, heat transfer, and the vibrations caused by fluid flow in dampers.
Gambit	Generate grids and meshes explicitly designed for Computational Fluid Dynamics (CFD) simulations, with a focus on their applicability to damper analysis.
Abaqus	Investigate fluid-structure interactions in car dampers, with a specific focus on their behaviour during dynamic events.
LS-DYNA	Examine the effects of dampers and their interactions with other vehicle components across different settings.

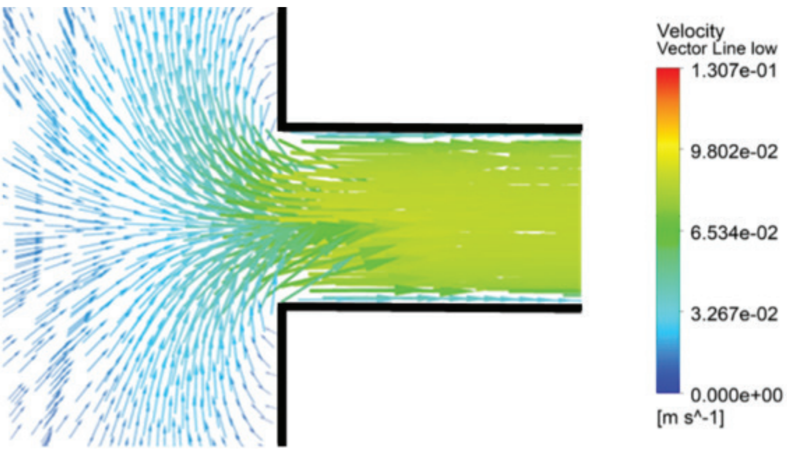

Figure 8. The velocity vectors of the MR fluid flow field in the MR damper at t=0.5 s represent the piston's highest velocity. The input current is 1.0 A, with a maximum magnetic field strength of 0.242 T at the magnetic poles. The fluid is stimulated by the movement of the piston walls at the entry of the throttling area (where the piston moves to the left while the fluid is restricted in the right direction). [From Elsaady W et al [71], with permission from Elsevier.]

dynamics, and improving damper designs for better vehicle performance and ride comfort [83]. Proper modelling techniques, mesh generation and validation procedures are important parts to ensure the accuracy and reliability of the computational fluid dynamics (CFD) simulations used in damper analysis [84].

Limitations of Flow Visualisation Techniques

Flow visualization methods for vehicle dampers are well known to be valuable tools for understanding and improving damper performance. However, it is important to acknowledge the limitations and problems of these techniques.

- ❖ Sometimes flow visualisation methods make real world situations simpler, so that experiments or simulations become more tractable. However, this over-simplification might lead to differences between the controlled lab or simulated environment and the real-world road environment, and potentially compromise the accuracy of the results.
- ❖ Automotive dampers are subjected to various external influences in the real world such as temperature variation, vibrations and road surface irregularities. These elements can be a source of noise and variability in the

experimental data and can make it difficult to isolate specific flow features.

- ❖ Full size damper action may not be fully reproduced in very transparent scale models. Differences in scale can influence the accuracy of the results of flow visualisation, especially for Reynolds-number and boundary-layer dependent phenomena.
- ❖ The visualization depth of transparent damper models and optical approaches is limited. Therefore, researchers may have limited ability to detect or quantify the flow characteristics occurring at significant depths within the damper and essential information might be omitted.
- ❖ Some flow imaging techniques (e.g., dye injection) can affect fluid dynamics by introducing external chemicals into the damper fluid. Such a modification may affect the precision of results especially when analyzing complex and delicate flow phenomena.
- ❖ High speed photography and particle tracking require sophisticated equipment and careful synchronisation. This leads to expensive and technically demanding experiments. Some issues may limit researchers' access to some techniques.
- ❖ Damping fluids are frequently characterized by complex, non-Newtonian and viscoelastic properties. This behaviour can be difficult to model and accurately visualise and may require specialist techniques.
- ❖ In optical methods, such as high speed photography, interference patterns and shadows can hide some details of the flow field and may therefore prevent a complete investigation of some particular flow phenomena.
- ❖ Significant processing resources and a high skill level are needed to execute computational fluid dynamics (CFD) simulations. Simulations at large scales can be very processing intensive and time consuming.
- ❖ It can be difficult to get the empirical data needed to validate the accuracy of computer models due to scale differences and boundary conditions. It is often difficult to get a perfect match between simulations and experiments.
- ❖ Potential researcher safety hazards exist in the research setting due to the use of high speed photography, pressure sensors and some chemicals used in dye injection. It is important to take proper safety precautions.

However, flow visualisation techniques are still indispensable to gain an insight into the behaviour of car dampers in the face of these restrictions. Researchers and engineers are continuously working on improving existing methodologies, addressing their inherent limitations and developing novel approaches to improve the accuracy and suitability of flow visualisation in damper analysis.

CONCLUSION

Techniques for flow visualisation are essential to understanding and improving the performance of vehicle dampers. The vital elements are very important to improve the comfort level during the rides, in maintaining the stability of the vehicle and in increasing the overall safety of the car and its passengers. The current discussion includes various flow visualisation techniques that reveal complex fluid dynamics within dampers and improve automobile engineering.

- ➢ A full understanding of damper behaviour is reached by employing a combination of different flow visualization methods which include transparent damper models, dye injection, high speed photography, particle tracking, pressure sensors, thermal imaging and computational fluid dynamics (CFD).
- ➢ Transparent damper models are used for educational purposes and as research prototypes for engineers and scientists. These tools allow us to observe directly the flow pattern and help us to understand better the principles of damping.
- ➢ Dye injection, high-speed photography and particle tracking techniques can be used to visualise and analyse fluid behaviour, turbulence and flow patterns within dampers. It makes easier the diagnosis of cavitation and other related problems.
- ➢ Pressure sensors and Computational Fluid Dynamics (CFD) simulations are used to obtain the numerical data of pressure distributions, temperature profiles and fluid dynamics. These data allow for the improvement of damper designs and the optimisation of their performance.
- ➢ Thermal imaging monitors temperatures and ensures dampers are within acceptable temperature thresholds. Integrated pressure sensors can detect overheating and cavitation, thus improving safety and efficiency.

Flow visualisation methods are important for validating computer models and optimizing damper designs. Engineers can anticipate and correct problems before they begin to impact vehicle operation in an adverse way. The knowledge that can be obtained from these results is of advantage both to automotive manufacturers and to the scientific and engineering domain in general. The development of flow visualisation techniques continues with the aim of improving vehicle safety, confort and performance. Engineers and researchers are ready to make data-driven decisions, improve designs and develop new solutions that enhance the overall driving experience. With the ever increasing advancement in technology and the increased understanding of fluid dynamics, the coming years present the possibility of more sophisticated flow visualisation methodologies. This is expected to lead to further improvements in vehicle dampers and overall vehicle dynamics.

ACKNOWLEDGEMENT

The authors express gratitude for the facilities provided by SRM Institute of Science and Technology at Kattankulathur, Chennai, Tamil Nadu 603203, India.

AUTHORSHIP CONTRIBUTIONS

Authors equally contributed to this work.

DATA AVAILABILITY STATEMENT

The authors confirm that the data that supports the findings of this study are available within the article. Raw data that support the finding of this study are available from the corresponding author, upon reasonable request.

CONFLICT OF INTEREST

The author declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

ETHICS

There are no ethical issues with the publication of this manuscript.

STATEMENT ON THE USE OF ARTIFICIAL INTELLIGENCE

Artificial intelligence was not used in the preparation of the article.

REFERENCES

[1] Vivas-Lopez CA, Hernández-Alcantara D, Morales-Menendez R, Ramírez-Mendoza RA, Ahuett-Garza H. Method for modeling electrorheological dampers using its dynamic characteristics. Math Probl Eng 2015:e905731. [CrossRef]

[2] Lak H, Zahrai SM, Mirhosseini SM, Zeighami E. Application of computational fluid dynamics in design of viscous dampers - CFD modeling and full-scale dynamic testing. Earthq Eng Eng Vib 2023. doi:10.1007/s11803-023-2209-5. [CrossRef]

[3] Alzughaibi A, Xue Y, Grosvenor R. A new insight into modelling passive suspension real test rig system with consideration of nonlinear friction forces. Proc Inst Mech Eng Part D J Automob Eng 2019;233:2257–2266. [CrossRef]

[4] Abd-Elsalam A, Gohary MA, El-Gamal HA. Modal analysis on tire concerning different parameters. Alexandria Eng J 2017;56:345–357. [CrossRef]

[5] Al-Yafeai D, Darabseh T, Mourad I. A state-of-the-art review of car suspension-based piezoelectric energy harvesting systems. Energies 2020;13:2336. [CrossRef]

[6] Yin Y, Rakheja S, Yang J, Boileau PE. Characterization of a hydro-pneumatic suspension strut with gas-oil emulsion. Mech Syst Signal Process 2018;106:319–333. [CrossRef]

[7] Omar M, El-Kassaby MM, Abdelghaffar W. A universal suspension test rig for electrohydraulic active and passive automotive suspension system. Alexandria Eng J 2017;56:359–370. [CrossRef]

[8] Vinuesa R, Brunton SL, McKeon BJ. The transformative potential of machine learning for experiments in fluid mechanics. Nat Rev Phys 2023;5:536–545. [CrossRef]

[9] Nagata T, Noguchi A, Nonomura T, Ohtani K, Asai K. Experimental investigation of transonic and supersonic flow over a sphere for Reynolds numbers of 103-105 by free-flight tests with schlieren visualization. Shock Waves 2020;30:139–151. [CrossRef]

[10] Procopio G, Giona M. Modal representation of inertial effects in fluid-particle interactions and the regularity of the memory kernels. Fluids 2023;8:84. [CrossRef]

[11] Suresha SP, Reddy GJ. Natural convective turbulent heat flow visualization around a vertical plate using low Reynolds number k-ε model. Waves Random Complex Media 2023;36:1–22. [CrossRef]

[12] Ferhath AA, Kasi K. The evolution of damper technology for enhanced ride comfort and vehicle handling in vehicle suspension system. Int J Dyn Control 2024;12:3908–3946. [CrossRef]

[13] Sancak ÖF, Güçlü R. Semi active control of fighter jet landing gear comparing with PID and fuzzy logic controller equipped magnetorheological damper. Sigma J Eng Nat Sci 2018;36:1215–1234.

[14] Raissi M, Yazdani A, Karniadakis GE. Hidden fluid mechanics: Learning velocity and pressure fields from flow visualizations. Science 2020;367:1026–1030. [CrossRef]

[15] Ivchenko AV. Flow visualization around cylinder under surface discharge action in the still atmosphere. J Phys Conf Ser 2021;2127:012010. [CrossRef]

[16] Gholinia M, Javadi H, Gatabi A, Khodabakhshi A, Ganji DD. Analytical study of a two-phase revolving system of nanofluid flow in the presence of a magnetic field to improve heat transfer. Sigma J Eng Nat Sci 2019;37:340–360.

[17] Taira K, Brunton SL, Dawson STM, Rowley CW, Colonius T, McKeon BJ, et al. Modal analysis of fluid flows: An overview. AIAA J 2017;55:4013–4041. [CrossRef]

[18] Badri Y, Syam T, Sassi S, Hussein M, Renno J, Ghani S. Investigating the characteristics of a magnetorheological fluid damper through CFD modeling. Mater Res Express 2021;8:055701. [CrossRef]

[19] Ahammed R, Hasan MZ. An FEA and CFD coupled numerical analysis approach for characterizing magneto-rheological (MR) grease damper. Sādhanā 2023;48:164. [CrossRef]

[20] Cai S, Wang Z, Fuest F, Jeon YJ, Gray C, Karniadakis GE. Flow over an espresso cup: Inferring 3-D velocity and pressure fields from tomographic background oriented Schlieren via physics-informed neural networks. J Fluid Mech 2021;915:A102. [CrossRef]

[21] Ferhath AA, Kasi K. Prediction of damping force in magnetorheological dampers using long short-term memory (LSTM) neural networks. Iran J Sci Technol Trans Mech Eng 2025;49:2387–2405. [CrossRef]
[22] Al-Zughaibi AIH. Experimental and analytical investigations of friction at lubricant bearings in passive suspension systems. Nonlinear Dyn 2018;94:1227–1242. [CrossRef]
[23] Grzesikiewicz W, Makowski M. Semi-active system of vehicle vibration damping. Appl Sci 2021;11:4577. [CrossRef]
[24] Caltagirone JP. The role of inertia in the onset of turbulence in a vortex filament. Fluids 2023;8:16. [CrossRef]
[25] Bailey SCC, Hultmark M, Monty JP, et al. Obtaining accurate mean velocity measurements in high Reynolds number turbulent boundary layers using Pitot tubes. J Fluid Mech 2013;715:642–670. [CrossRef]
[26] Vinuesa R, Brunton SL. Enhancing computational fluid dynamics with machine learning. Nat Comput Sci 2022;2:358–366. [CrossRef]
[27] Waris W, Lappa M. Patterning behavior of hybrid buoyancy-Marangoni convection in inclined layers heated from below. Fluids 2023;8:12. [CrossRef]
[28] M SK, S RB, M V, K NSR. Applying visualization techniques to study the fluid flow pattern and the particle distribution in the casting of metal matrix composites. J Manuf Process 2020;58:668–676. [CrossRef]
[29] Kadivar M, Tormey D, McGranaghan G. A review on turbulent flow over rough surfaces: Fundamentals and theories. Int J Thermofluids 2021;10:100077. [CrossRef]
[30] Liu Y, Lu Y, Wang Y, Sun D, Deng L, Wang F, Lei Y. A CNN-based shock detection method in flow visualization. Comput Fluids 2019;184:1–9. [CrossRef]
[31] Arafa N, Sullivan PE, Ekmekci A. Jet velocity and acoustic excitation characteristics of a synthetic jet actuator. Fluids 2022;7:387. [CrossRef]
[32] Chashechkin YD. Foundations of engineering mathematics applied for fluid flows. Axioms 2021;10:286. [CrossRef]
[33] Bhatti MM, Marin M, Zeeshan A, Abdelsalam SI. Editorial: Recent trends in computational fluid dynamics. Front Phys 2020;8. [CrossRef]
[34] Reinke P, Beckmann T, Ahlers C, Ahlrichs J, Hammou L, Schmidt M. High-speed digital photography of vapor cavitation in a narrow gap flow. Fluids 2023;8:44. [CrossRef]
[35] Mambro A, Congiu F, Galloni E, Lanni D. Blade drag resistance in windage operating of low pressure steam turbines. Fluids 2022;7:372. [CrossRef]
[36] Fuller PWW. Aspects of high speed photography-4-the study of fluid flow. J Photogr Sci 1994;42:133–139. [CrossRef]
[37] Brown RAS, Govier GW. High-speed photography in the study of two-phase flow. Can J Chem Eng 1961;39:159–164. [CrossRef]
[38] Eriksson I, Gren P, Powell J, Kaplan AF. New high-speed photography technique for observation of fluid flow in laser welding. Opt Eng 2010;49:1. [CrossRef]
[39] Heinz S. From two-equation turbulence models to minimal error resolving simulation methods for complex turbulent flows. Fluids 2022;7:368. [CrossRef]
[40] Swart B, Zhao Y, Khaku M, Che E, Maltby R, Chew YMJ, et al. In situ characterisation of size distribution and rise velocity of microbubbles by high-speed photography. Chem Eng Sci 2020;225:115836. [CrossRef]
[41] Xing HZ, Zhang QB, Braithwaite CH, Pan B, Zhao J. High-speed photography and digital optical measurement techniques for geomaterials: Fundamentals and applications. Rock Mech Rock Eng 2017;50:1611–1659. [CrossRef]
[42] Starodumov I, Nizovtseva I, Lezhnin S, Vikharev S, Svitich V, Mikushin P, et al. Measurement of mass transfer intensity in gas-liquid medium of bioreactor circuit using the thermometry method. Fluids 2022;7:366. [CrossRef]
[43] Liu Y, Wu W, Luo Y, Chu GW, Liu W, Sun BC, et al. CFD simulation and high-speed photography of liquid flow in the outer cavity zone of a rotating packed bed reactor. Ind Eng Chem Res 2019;58:5280–5290. [CrossRef]
[44] Fernández del Olmo P, Acién FG, Fernández-Sevilla JM. Productivity analysis in tubular photobioreactors using a dynamic photosynthesis model coupled to computational fluid dynamics particle tracking. Bioresour Technol 2022;344:126277. [CrossRef]
[45] Gedik E, Kurt H, Pala M, Alakour A. An experimental and artificial neural network investigation on the laminar flow of magnetorheological fluids through circular pipes. J Magn Magn Mater 2022;546:168893. [CrossRef]
[46] Wang H, Liu Y, Wang S. Dense velocity reconstruction from particle image velocimetry/particle tracking velocimetry using a physics-informed neural network. Phys Fluids 2022;34:017116. [CrossRef]
[47] Sommerfeld M, Cui Y, Schmalfuß S. Potential and constraints for the application of CFD combined with Lagrangian particle tracking to dry powder inhalers. Eur J Pharm Sci 2019;128:299–324. [CrossRef]
[48] McGlynn JA, Wu N, Schultz KM. Multiple particle tracking microrheological characterization: Fundamentals, emerging techniques and applications. J Appl Phys 2020;127:201101. [CrossRef]
[49] Gergely A, Néda Z. Computational fluid dynamics approach for oscillating and interacting convective flows. Fluids 2022;7:339. [CrossRef]
[50] Mallery K, Hong J. Regularized inverse holographic volume reconstruction for 3D particle tracking. Opt Express 2019;27:18069–18084. [CrossRef]

[51] Iturbide Jiménez F, Mendoza Jasso AJ, Antonio García A, Santiago Alvarado A. Design and construction of an apparatus to visualize incompressible fluid flow in several regimes. Rev Mex Fis E 2018;64:133−138. [CrossRef]
[52] Taira K, Hemati MS, Brunton SL, Sun Y, Duraisamy K, Bagheri S, et al. Modal analysis of fluid flows: Applications and outlook. AIAA J 2020;58:998−1022. [CrossRef]
[53] Serdar Genç M, Koca K, Açikel HH. Investigation of pre-stall flow control on wind turbine blade airfoil using roughness element. Energy 2019;176:320−334. [CrossRef]
[54] Dodds D, Sarhan AAR, Naser J. CFD investigation into the effects of surrounding particle location on the drag coefficient. Fluids 2022;7:331. [CrossRef]
[55] Wu X, Zhang X, Tian X, Li X, Lu W. A review on fluid dynamics of flapping foils. Ocean Eng 2020;195:106712. [CrossRef]
[56] Williams O, Samuell M, Sarwas ES, Robbins M, Ferrante A. Experimental study of a CFD validation test case for turbulent separated flows. AIAA Scitech 2020 Forum 6-10 January 2020 Orlando, FL, USA. [CrossRef]
[57] Guzmán E, Maestro A, Carbone C, Ortega F, Rubio RG. Dilational rheology of fluid/fluid interfaces: Foundations and tools. Fluids 2022;7:335. [CrossRef]
[58] Yu Z, Cai G, Liu X, Tang D. Pressure-based biosensor integrated with a flexible pressure sensor and an electrochromic device for visual detection. Anal Chem 2021;93:2916−2925. [CrossRef]
[59] Gücüyen E, Erdem RT, Gökkuş Ü. Numerical modelling of sudden contraction in pipe flow. Sigma J Eng Nat Sci 2020;37:903−916.
[60] Ma C, Xu D, Huang YC, Wang P, Huang J, Zhou J, et al. Robust flexible pressure sensors made from conductive micropyramids for manipulation tasks. ACS Nano 2020;14:12866−12876. [CrossRef]
[61] Guo Y, Gao S, Yue W, Zhang C, Li Y. Anodized aluminum oxide-assisted low-cost flexible capacitive pressure sensors based on double-sided nanopillars by a facile fabrication method. ACS Appl Mater Interfaces 2019;11:48594−48603. [CrossRef]
[62] Han X, Li W, Zhao H, Li J, Tang X, Wang L. Slippery damper of an overlay for arresting and manipulating droplets on nonwetting surfaces. Nat Commun 2021;12:3154. [CrossRef]
[63] Zhou H, Chen X, Zhang Y, Ai Y, Sun D. An analysis on the influence of air ingestion on vibration damping properties of squeeze film dampers. Tribol Int 2020;145:106168. [CrossRef]
[64] Mu Y, Liu M, Ma Z, Zhang J. Resistance characteristic analysis based study on a novel damper torque airflow sensor for VAV terminals. Build Environ 2020;175:106813. [CrossRef]
[65] Paramasivam B. Investigation on the effects of damping over the temperature distribution on internal turning bar using infrared fusion thermal imager analysis via SmartView software. Measurement 2020;162:107938. [CrossRef]
[66] Jadidi M, Moghtadernejad S, Hanson J. Numerical study of the effects of twin-fluid atomization on the suspension plasma spraying process. Fluids 2020;5:224. [CrossRef]
[67] Kingston TA, Weibel JA, Garimella SV. Time-resolved characterization of microchannel flow boiling during transient heating: Part 1 - Dynamic response to a single heat flux pulse. Int J Heat Mass Transf 2020;154:119643. [CrossRef]
[68] Yan W, Xuesong L, Yuhong L. CFD simulation of air ingestion in a squeeze film damper. 2019. ASME Turbo Expo 2019: Turbomachinery Technical Conference and Exposition June 17-21, 2019, Phoenix, Arizona, USA. [CrossRef]
[69] Lak H, Zahrai SM. Self-heating of viscous dampers under short- & long-duration loads: Experimental observations and numerical simulations. Structures 2023;48:275−287. [CrossRef]
[70] Czop P, Gniłka J. A quick-and-dirty method for assessing the risk of negative aeration effects of shock absorbers equipped with shim sliding base valves. Comput Assist Methods Eng Sci 2022;29:229−260.
[71] Elsaady W, Oyadiji SO, Nasser A. A one-way coupled numerical magnetic field and CFD simulation of viscoplastic compressible fluids in MR dampers. Int J Mech Sci 2020;167:105265. [CrossRef]
[72] Ferhath AA, Kasi K. Conventional flow analysis methods in vehicle damper performance evaluation. Discov Mech Eng 2025;4:8. [CrossRef]
[73] Yan Y, Tu J. Computational fluid dynamics. In: Yan Y, Tu J, editors. Bioaerosol Characterisation, Transportation and Transmission: Fundamental, Modelling and Application. Singapore: Springer Nature; 2023. p. 65−83. [CrossRef]
[74] Tharehalli Mata G, Kumar H, Mahalingam A. Performance analysis of a semi-active suspension system using coupled CFD-FEA based non-parametric modeling of low capacity shear mode monotube MR damper. Proc Inst Mech Eng Part D J Automob Eng 2019;233:1214−1231. [CrossRef]
[75] Demir S. CFD analysis of mini Venturi-type air flowmeters for lab-scale applications. Sigma J Eng Nat Sci 2021;38:603−612.
[76] Vishwakarma PN, Mishra P, Sharma SK. Characterization of a magnetorheological fluid damper: A review. Mater Today Proc 2022;56:2988−2994. [CrossRef]
[77] Nie W, He H, Sha L, Wang C, Du F. Computational fluid dynamics simulation and optimization of hydropneumatic spring damper valves for heavy vehicle applications. Machines 2023;11:680. [CrossRef]
[78] Xu P, Babanezhad M, Yarmand H, Marjani A. Flow visualization and analysis of thermal distribution for

the nanofluid by the integration of fuzzy c-means clustering ANFIS structure and CFD methods. J Vis 2020;23:97–110. [CrossRef]

[79] Akınbo BJ, Olajuwon BI. Heat and mass transfer in magnetohydrodynamics (MHD) flow over a moving vertical plate with convective boundary condition in the presence of thermal radiation. Sigma J Eng Nat Sci 2020;37:1031–1053.

[80] Ferhath AA, Kasi K. Orifice and fluid flow modifications for improved damping in vehicle suspensions: A comprehensive review. J Vib Eng Technol 2024;12:6777–6808. [CrossRef]

[81] Gurubasavaraju TM. Estimation of damping force of double ended magnetorheological damper through one way coupled CFD and FEA analysis. J Inst Eng India Ser C 2021;102:1145–1151. [CrossRef]

[82] Aziz MA, Aminossadati SM, Leonardi C. Performance analysis of magnetorheological plastomer dampers with different annular shear gaps. J Magn Magn Mater 2023;568:170396. [CrossRef]

[83] Dahlin Rodin C, de Alcantara Andrade FA, Hovenburg AR, Johansen TA. A survey of practical design considerations of optical imaging stabilization systems for small unmanned aerial systems. Sensors 2019;19:4800. [CrossRef]

[84] Mata GT. Characterization of dual annular duct MR damper with numerical and computational approach. J Vib Eng Technol 2023;12:3625–3640. [CrossRef]