

A Modular Multifidelity Approach for Multiphysics Oleo-Pneumatic Shock Absorber Simulations

Ahmed A. Sheikh Al-Shabab*, Paulo A. S. F. Silva, Bojan Grenko,
Panagiotis Tsoutsanis, and Martin Skote

School of Aerospace, Transport and Manufacturing, Cranfield University, College Road,
Cranfield, Bedfordshire, MK43 0AL, UK

5 March 2024

Abstract

A numerical framework is developed and tested to simulate the internal dynamics of oleo-pneumatic shock absorbers. A modular approach is devised to address the multiphysics nature of the problem, starting with three dimensional scale resolving turbulence simulations and two dimensional axisymmetric multiphase URANS simulations. These simulations capture the main dominant aspects of energy dissipation through turbulence, and the multiphase mixing which can affect the working fluid properties. Internal flow simulations are run on a representative shock absorber geometry based on dimensions provided in the validation study together with sizing guidelines from the literature. A lower fidelity two-equation dynamic system solver is used to scan the design space and test the sensitivity towards various design parameters, in addition to identifying parameter combinations that would be of interest to investigate using higher fidelity methods.

1 Introduction

Oleo-pneumatic shock absorbers have been a relatively stable component on aircraft for several decades without witnessing progress comparable to that made in other areas, such as aeroengines or aerospace materials. This is partly due to the combined complexity of the internal flow dynamics and geometries involved, making experimental and computational work challenging and expensive. The multiphysics nature of the internal flow makes it difficult to identify the significant features of the flow field and the conditions under which specific phenomena become important in a given design.

A main difficulty in the investigation of oleo-pneumatic shock absorbers is the lack of widely researched standard geometries with known properties, similar to the NACA-series in the case of aerofoils. Therefore, a baseline geometry was generated by using the available information from the drop down experiment used for validation [1], while the missing parameters were estimated based on the general guidelines and sizing data available in the literature [2, 3].

The study of Milwitzky and Cook [1] was chosen as the reference experimental validation case in the present investigation, because it provides detailed drop test experimental measurements that can be used for both high-fidelity and low-fidelity model validation. The shock absorber used is of a small military trainer aircraft landing gear, with a relatively simple geometry, for which some key parameters are available. Therefore, a version of the full three-dimensional geometry can be approximately reconstructed based on these key parameters, together with the sizing guidelines in the literature [2, 3], for use in high-fidelity simulations. In addition, a comprehensive development of a dynamic system model of landing gear shock absorbers is presented by Milwitzky and Cook [1], including tire forces, inclination effects, friction forces and ground forces. This provides a useful basis for a low-fidelity simulation of the system.

The present investigation focuses on the development and validation of high and medium fidelity branches of a multifidelity framework developed to enable efficient generation of shock absorber performance databases and to facilitate design space exploration.

2 Methodology and Numerical Methods

The prediction of shock absorber performance requires accounting for a wide range of physical phenomena that are expensive to compute directly using fully resolved three-dimensional simulations. The main dominant features include the multiphase

*Paper presented at the Cambridge Unsteady Flow Symposium 4-5 March 2024. Corresponding author: Ahmed A. Sheikh Al-Shabab, email: a.a.sheikh-al-shabab@cranfield.ac.uk

interaction between hydraulic oil and gas inside the shock absorber, see Fig. 2a, in addition to the turbulent shear flow through the orifice. The multiphase interaction has an influence on the variation of working fluid properties as the stroke progresses, while the developing turbulent flow downstream of the orifice is central to the energy absorption within the shock absorber [2].

It is possible to conduct an integrated multi-physics shock absorber simulation, that accounts for most of the flow physics in a single simulation. However, it is more efficient (and computationally stable) to adopt a modular approach using specialised tools and concentrating on each of the flow physics in turn, particularly when a large number of simulations are planned to explore the design space. In this modular approach, understanding the coupling between the various aspects of flow physics requires careful consideration, and potentially a few integrated fully coupled multi-physics simulations. Nevertheless, the benefit of taking advantage of the most efficient and well developed tool in each area is attractive. The chosen numerical solvers have been validated on model problems that are representative of the flow physics of interest [4].

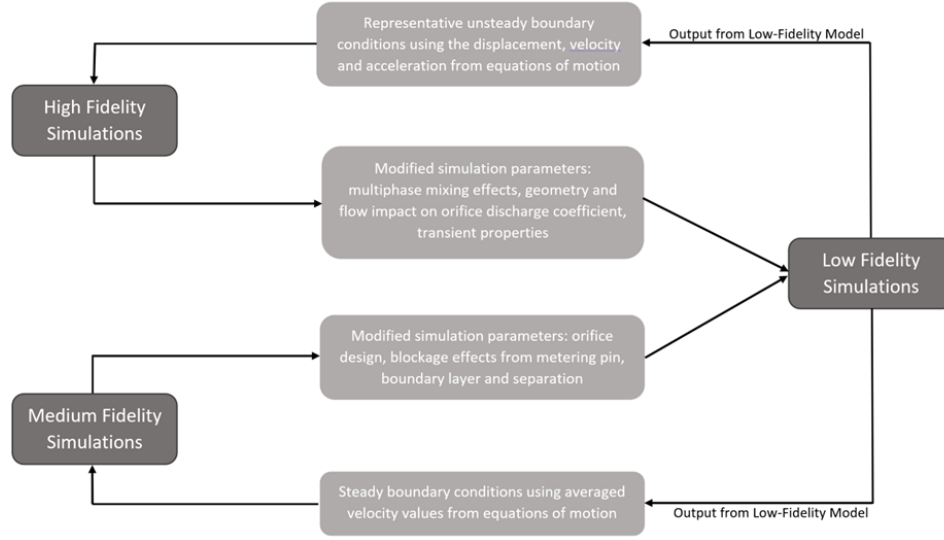


Figure 1: Multi-Fidelity interaction to refine discharge coefficient estimate and provide boundary conditions.

Adopting a multi-fidelity approach, Fig 1, increases the efficiency of design space exploration by using more expensive high-fidelity scale resolving simulations to provide fundamental flow physics understanding and to calibrate lower fidelity models in different flow regimes. Medium-fidelity models such as RANS simulations can also be utilised as an intermediate solution to conduct design refinement and fine tune low-fidelity models. RANS models have been shown to be effective at predicting free shear layer deflections in design studies [5, 6], which could be relevant here in the presence of different metering pin or valve flap designs. However, the two-equation dynamic system model low-fidelity solver [1, 4] will be tasked with carrying out the majority of simulations in design studies or database generation activities. It can also provide case specific boundary conditions for high-fidelity simulations from the relative piston displacement, velocity and acceleration. On the other hand, high-fidelity data can be used to calibrate critical low-fidelity solver parameters to improve prediction of the later stages of the shock absorber stroke.

2.1 Multiphase Simulation

Multiphase simulation cases are conducted using ANSYS Fluent 2019 R2 [7]. The cases are set up as 2D half sections with an axisymmetric boundary condition along the axis of symmetry. A Pressure based solver is used with the unsteady RANS $k - \omega SST$ [8] turbulence model. The ideal gas law is applied to account for gas compressibility. The domain is filled with two phases: hydraulic oil *MIL-H-5606*, and air, in accordance with the drop test [1]. A no-slip boundary condition is used on the side and top shock absorber walls, while an axisymmetric boundary condition is applied along the axis, and a variable inlet velocity is applied at the bottom wall. These boundary conditions are annotated on the computational domain and mesh visualization in Fig. 7. The time dependent inlet velocity is designed to replicate the shock absorber stroke rate, as an alternative to a moving mesh approach, because it reduces the computational effort and allows better control over mesh quality. The spatial discretization schemes are PRESTO (PREssure STaggering Option) scheme for pressure, modified HRIC for volume fraction, and second order upwind for the rest of the flow variables. The multiphase modelling method is an explicit formulation of the volume of fluid scheme with sharp interface modelling. A dual-time stepping approach is used with a maximum of 40 sub-iterations per physical time step. Variable time stepping method is applied with a CFL number constraint of 0.3, and a minimum allowable physical time step of $1e - 8$ seconds.

2.2 Turbulent Shear Layer Simulation

OpenFoam (v2006) [9, 10] was used to conduct single phase time-dependent scale resolving turbulent shear layer simulations. OpenFoam uses an unstructured finite volume approach to simulate fluid flow, which facilitates the handling of complex geometries that are expected in industrial type applications, as in the case of the shock absorbers investigated here. Field properties are stored at the centroid of each cell and then calculated at each cell surface "face" by interpolating the values at the centroids of the two cells sharing that face. Conservation laws are then applied based on these face values, in accordance with the chosen discretization scheme.

The PIMPLE algorithm [11] is selected in the present simulations, as it provides good stability and convergence control options. PIMPLE utilises an outer loop based on the PISO algorithm [12] to capture the field development in time, while an inner loop based on the SIMPLE algorithm [13] solves each field instance as a steady problem. The interpolation method used is a second order linear interpolation, which is a standard choice in many finite volume solvers. The convective term is discretized using a second order limited linear scheme, which provides a balance between accuracy and stability through limiting the nonlinear term. While the diffusive term is discretized using a second order linear scheme and temporal integration is conducted using a second order backward difference method. An adjustable time step is used to insure that the Courant–Friedrich–Lewy (CFL) number remains below 1.0 throughout, to aid in achieving a balance between efficiency and stability during these simulations.

After the discretization process, the resulting linear system for the pressure field is solved for using a Preconditioned Conjugate Gradient (PCG) method, while the velocity field system is found using a Preconditioned Bi-Conjugate Gradient method (PBiCG). The WALE subgrid scale model [14] is used to model the eddy viscosity of the subgrid scales in the Large Eddy Simulation, (LES) framework. The flow field is allowed to develop by allowing multiple through-flows across the entire domain, then time averaging is conducted until statistical convergence is observed for all turbulent properties of interest.

3 Computational Setup

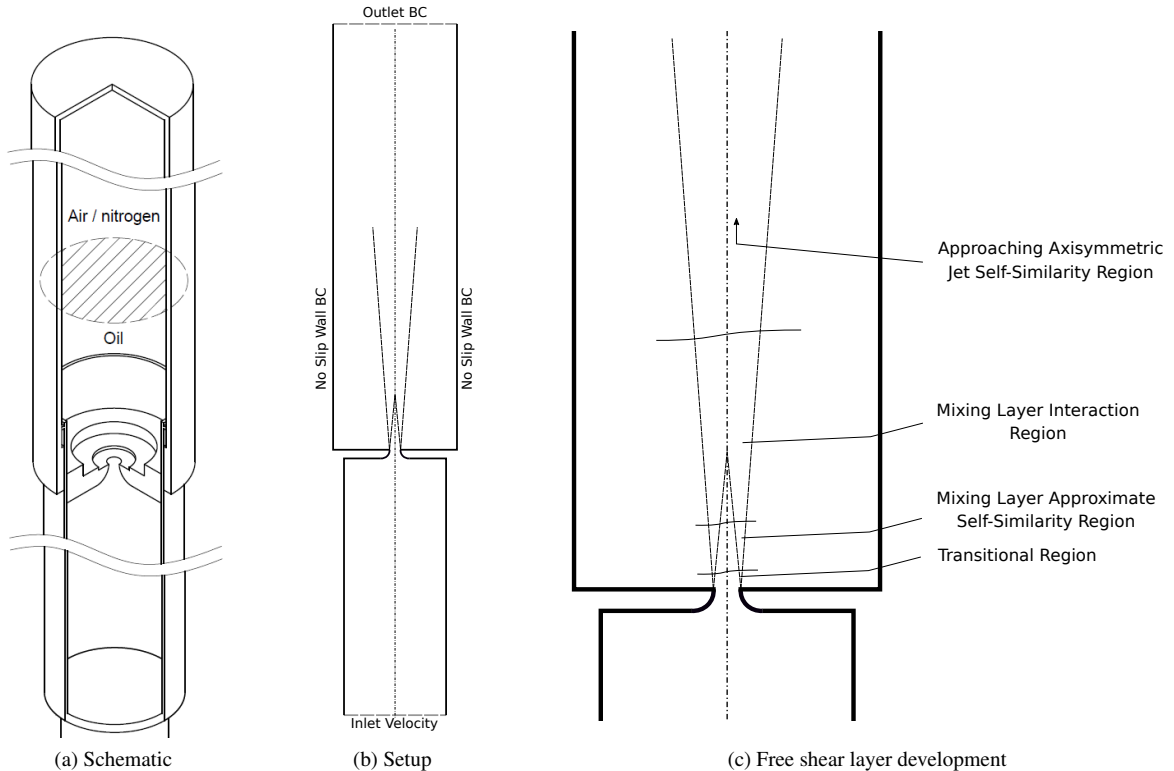


Figure 2: Shock absorber geometry and setup details, taken from Fig. 1 in Sheikh Al-Shabab *et. al.* [15]. (a) Schematic view of the simplified shock absorber geometry; (b) The computational domain with the applied boundary conditions; (c) Axisymmetric shear layer development regions, following Bradshaw *et al.* [16].

The baseline orifice geometry, illustrated in Fig. 2, has an orifice with semi-circular lips. The rounded outer (upstream) edge of the orifice helps avoid any boundary layer separation on the orifice's inner surface, while the accelerating flow under the influence of the favourable pressure gradient through the orifice allows relatively strong flow turning in the flow direction so that the flow can be tangential to the axial (streamwise) direction at the orifice exit. The three dimensional simulation is conducted on a simplified geometry of the shock absorber that focuses on the orifice nearfield region and the developing shear

Table 1: Scale resolving 3D axisymmetric free shear layer simulation.

Simulation Parameter	Setting
Working Fluid type	Hydraulic Oil (AN-VV-O-366B)
Hydraulic Oil Density, ρ_h (kg/m ³)	869
Hydraulic Oil Kinematic Viscosity, ν_h (m ² /s)	1.35×10^{-5}
Time step	$\approx 8 \times 10^{-7}$
Orifice Diameter, m	8.128×10^{-3}
Lower Chamber Diameter, m	7.462×10^{-2}
Upper Chamber Diameter, m	8.255×10^{-2}
Inlet Velocity, m/s	1.0
$Re = \frac{U_{orifice} D_{orifice}}{\nu_h}$	3×10^4
Total Grid Nodes	1.8×10^6

layer in its vicinity.

Grids of varying resolution were generated and tested to establish grid independence and optimise the setup, where possible. The resolution in the shear layer region was tested using a preliminary simulation on a grid with 185,000 nodes. This included investigating the estimated mixing layer thickness near the orifice tip to determine the grid resolution required there. The boundary conditions are illustrated in Fig. 2b, while further information about the case is provided in Table 1.

The inlet boundary condition, in the single phase 3D simulation, is taken to be a constant vertical flow through the shock absorber $V = 1.0$ m/s as a representative figure of an approximately time-averaged value over the stroke period from the Milwitzky and Cook [1] drop test. The stroke shape and value, corresponding to the drop test, can be seen in Fig. 3b, where this figure was obtained using a dynamic system model simulation of the drop test following the analysis in Milwitzky and Cook [1]. The initial delay in the response of the shock absorber, the flat region at time = 0.0 s, is due to the force required to deflect the tire and other internal effects (e.g., the gas pressure force), which leads to the shock absorber behaving as a rigid body during the early stages of impact; however, this is quickly overcome by the impact resulting in the subsequent stroke profile.

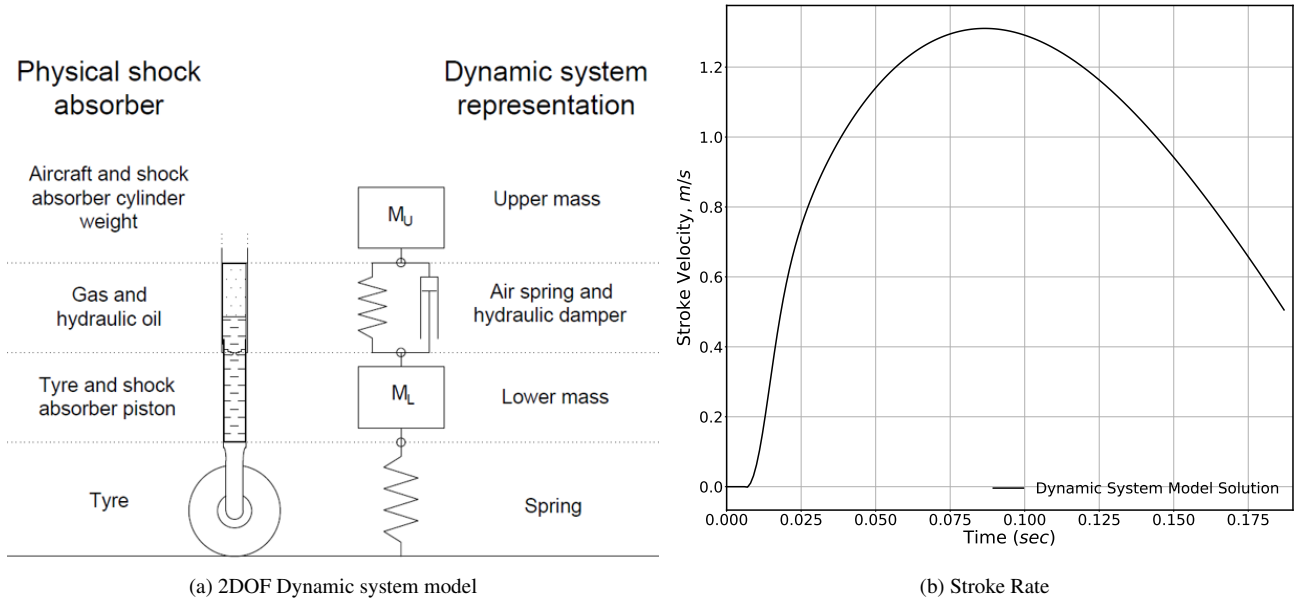


Figure 3: Shock absorber stroke velocity profile from a dynamic system model simulating a drop test, which is used to estimate the inlet velocity value in the LES study.

The flow in the present simulation is assumed to be incompressible, which is justified due to the low Mach number

prevalent throughout the computational domain ($M < 0.3$), and, since the focus of the study is on the free shear layer under a constant inlet velocity, we ignore the influence of the transient compression pressure waves associated with the movement of the piston in a real shock absorber. This assumption helps to increase the efficiency of the simulation by allowing the numerical stability to be determined by the hydrodynamic time-scale rather than the more restrictive acoustic time-scale.

The edge of the free shear layer for each profile was found locally based on the vorticity magnitude and streamwise velocity value. Then the high and low velocity values across the profile were defined, and their difference was used for normalisation of the velocity values across that specific profile. The self-similarity distance variable η is defined according to Equation E:SimCoord where the capital vertical coordinate Y has its origin at the orifice exit.

$$\eta = \frac{x - D_{orifice}/2}{Y}, \quad (1)$$

It should be noted that, in the 3D single phase simulation, the streamwise direction is along the vertical y-direction with the corresponding velocity component and turbulent (Reynolds) stress component, $v'v'$. For the purposes of analysing the near-field free shear layer development, a two-dimensional (2D) section (x - y plane at $z = 0$) is extracted from the three-dimensional domain, where the mixing layer is quasi two-dimensional within a distance equivalent to a few orifice diameters downstream of the orifice exit. In this region, the mixing layer is expected to exhibit standard behaviour that can be used to validate the simulation against similar mixing layer characteristics reported in experiments, which is a substitute for the lack of internal shock absorber data available in the literature.

4 Results and Discussion

4.1 Single Phase Turbulent Shear Layer

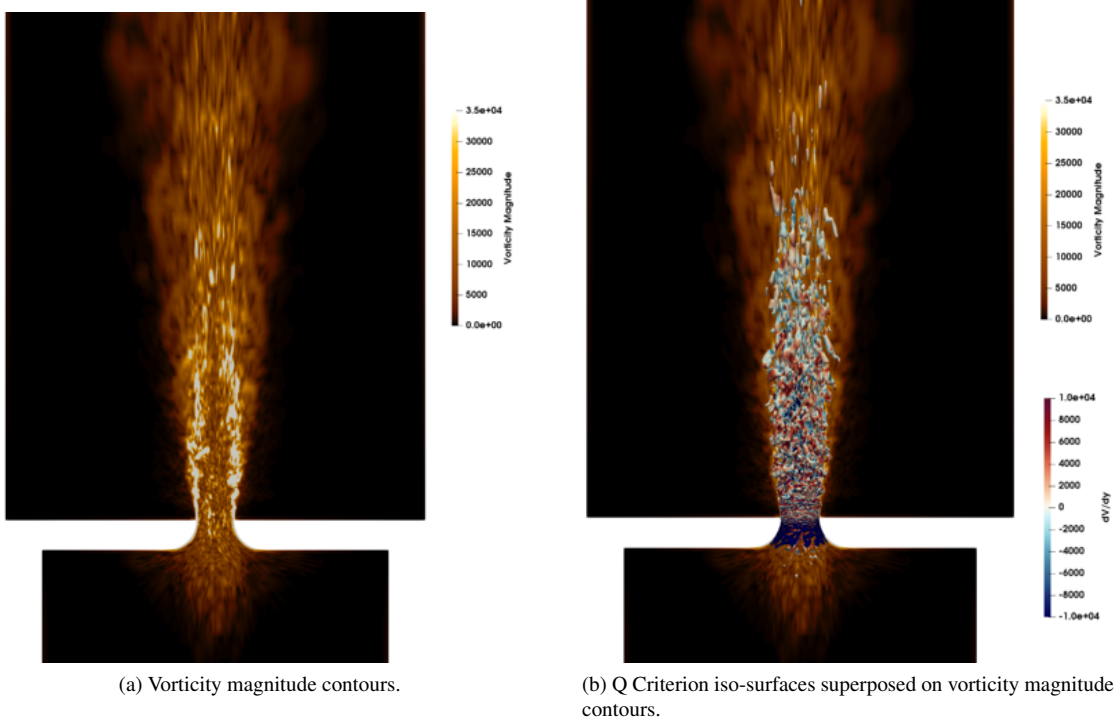


Figure 4: Flow development in the vicinity of the orifice.

Single phase simulations are used to focus on the orifice nearfield and the impact of modifications made in that region on the flow development. This includes the development of turbulent flow downstream, the pressure gradient around the orifice and its susceptibility to cavitation, as well as the overall hydraulic resistance and its likely impact on overall shock absorber performance.

The development of flow instabilities and turbulence in the free shear layers downstream of the orifice is visualised in Fig. 4, using the vorticity magnitude and Q-criterion, found by plotting iso-surfaces of the second invariant of the velocity strain rate tensor as shown in Fig. 4b. Driven by a strong primary instability, the development of increasingly complex flow structures in the mixing layer is observed as initially quasi two-dimensional instabilities, such as ring vortices, develop into

more complex structures and breakdown to progressively smaller scales, until the full turbulent cascade is established further downstream.

Vorticity magnitude contours are more clearly shown in Fig. 4a, where high vorticity magnitude values in the turbulent free shear layers downstream of the orifice are expected as a consequence of the strong primary instabilities and the turbulent flow in that region. However, it is also found that there is a region of significant vorticity magnitude upstream of the orifice. This is likely caused by the strong blockage effect of the orifice causing rapid flow radial convergence towards the axis of symmetry, resulting in a relatively *noisy* orifice jet core.

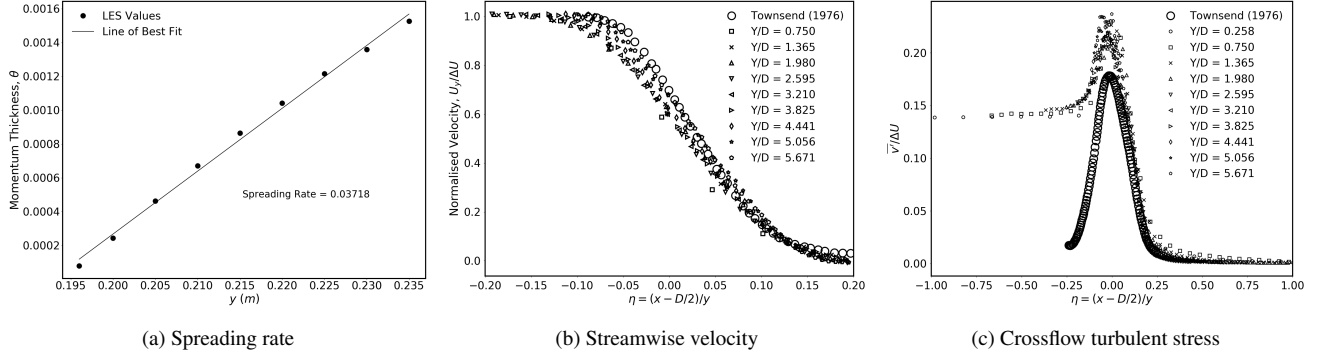


Figure 5: Free shear layer spreading rate and self-similarity validation. Plots modified from Sheikh Al-Shabab *et. al.* [15].

The development of the free shear layer downstream of the orifice can be validated against standard results for mixing layers from the literature [17, 18]. The spreading rate, shown in Fig. 5a, is calculated based on the growth rate of the shear layer's momentum thickness, where Jimenez [18] devised the following relation to fit the reported experimental measurements for a range of mixing layer stream ratios:

$$SR_\theta = \frac{d\theta}{dx} = 0.036\beta, \quad \beta = \frac{U_{High} - U_{Low}}{U_{High} + U_{Low}} \quad (2)$$

Noting that $\beta \approx 1$ for a single stream mixing layer, the calculated spreading rate value here of $SR_\theta = 0.037$ is within the expected range, particularly considering the spread in the mixing layer data [18] used to arrive at Eq. 2.

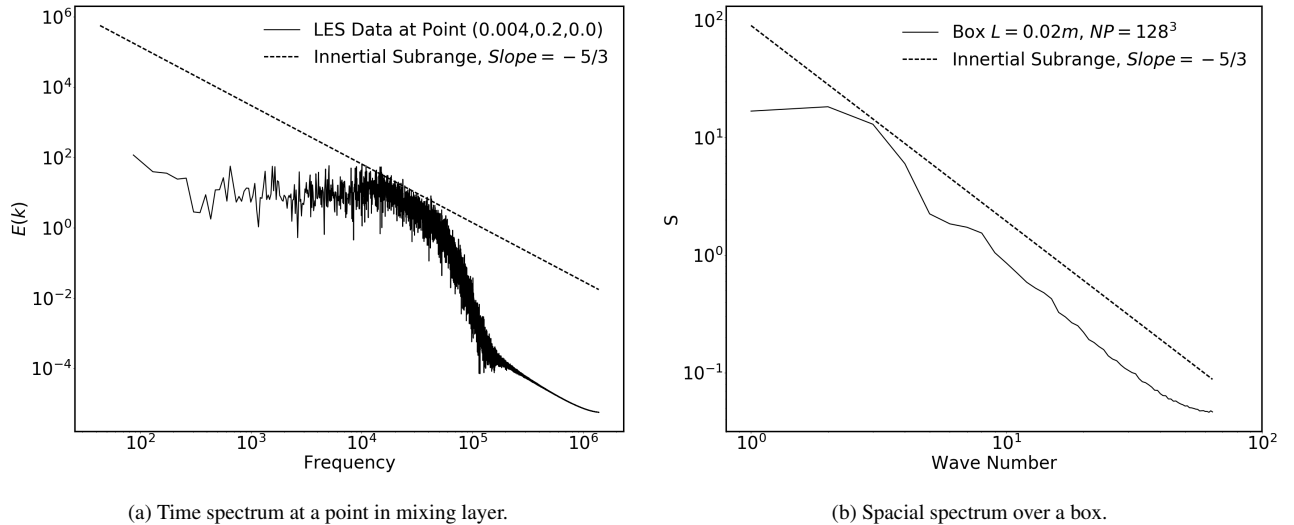


Figure 6: Spectra downstream of orifice.

The tendency of the mixing layer towards self-similarity is also demonstrated in Figs. 5b and 5c for the streamwise velocity and the the cross flow Root Mean Squared (RMS) component $\sqrt{u' u'}$ respectively. This state is reached once the free shear layer passes the initial transitional stage near the tip and enters the turbulent regime around one orifice diameter downstream of the exit, where the profiles of turbulent flow variables taken across the shear layer tend to collapse onto a single profile once they are normalised appropriately [17]. The first profile plotted at $Y/D_{orifice} = 0.75$ has a relatively larger deviation from

the self-similar profile while further downstream profiles exhibit increasingly better agreement with the self-similar one. The higher turbulent stress values on the inside of the jet through the orifice are consistent with the high unsteadiness observed in the core of the jet in Fig. 4a.

From a design prospective, temporal and spatial energy spectra, Fig. 6, are important to assess the critical regions of energy dissipation in different orifice designs and configurations. Turbulent kinetic energy spectra are calculated from the unsteady velocity fluctuations time history at a point in Fig. 6a, while in Fig. 6b a box spanning the central region downstream of the orifice including the shear layers is used following Tsoutsanis [19]. Both spectra plots show the presence of an inertial sub-range consistent with a developed turbulent cascade in their corresponding location. This will be further developed when applied to more complex designs, that could include metering pins, snubber chambers and other internal details.

4.2 Multiphase Simulation

Multiphase simulations are conducted on an unstructured two-dimensional axisymmetric domain, where the layout and mesh are shown in Fig. 7, with an unsteady velocity inlet profile corresponding to the piston stroke velocity given in Fig. 3b. The geometry used in these simulations, Fig. 2a, is simplified from the original to focus on the main multiphase interaction of interest, while reducing the geometrical complexity of orifice support tubes, cylinder telescoping bearings and snubber chamber details. Particularly since the details of these features are not publicly available in the literature.

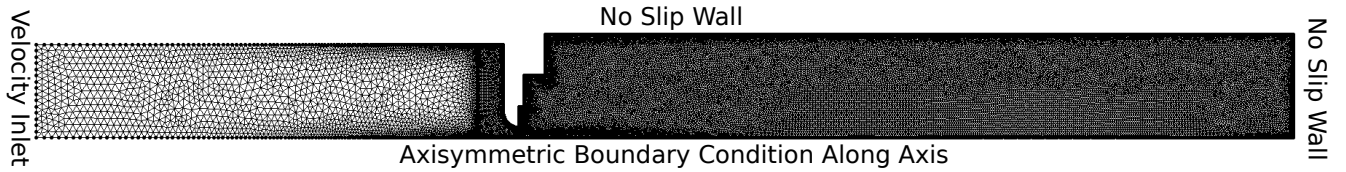


Figure 7: Simplified oleo pneumatic shock absorber 2D half domain mesh for validation case of Milwitzky and Cook [1].

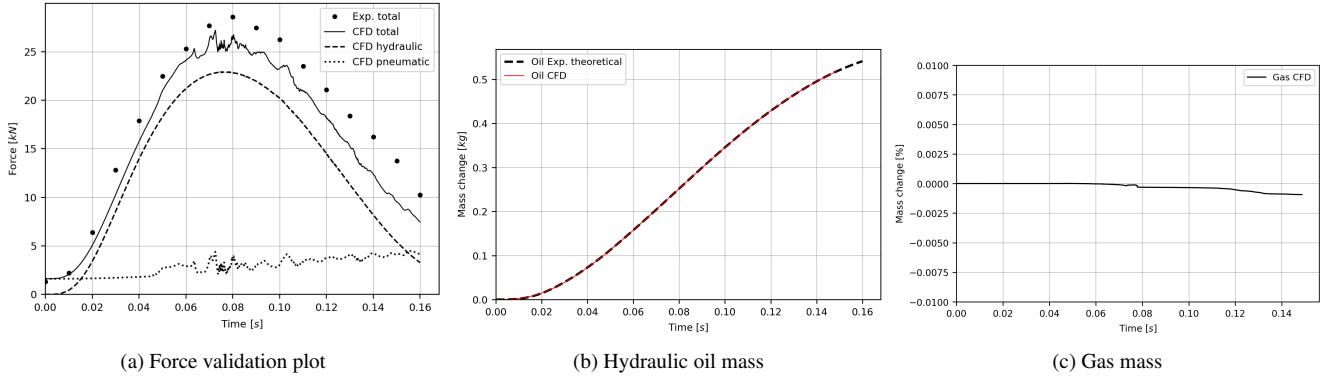


Figure 8: High-fidelity multiphase drop test case validation by comparison with force measurements and checking phase mass conservation.

The validation of the drop test case is performed against the experimental measurements of Milwitzky and Cook [1], which consist of the measured total force exerted by the shock absorber in the time span from impact at 0.0s to the end of the main stroke at 0.16s. The total force calculated from the present simulation is compared with the experimental measurements in Fig. 8a, where the agreement is found to be strong in the first half of the stroke until roughly the maximum force point, after which there is a nearly constant underprediction of the total force by the computational simulation relative to the experimental measurements. The observed discrepancy in the advanced stages of the stroke isn't surprising due to the increasing complexity of the flow physics and the likely impact of simplifying assumptions.

It is worth noting that the total force can't be extracted directly from the simulation in the manner an accelerometer can record the total force in the drop test. instead, the hydraulic and pneumatic forces are computed separately and then added to find the total force values. The individual curves of the pneumatic and hydraulic forces are also plotted in Fig. 8a to provide an insight into their contribution to the total and ensure their overall shape and values are reasonable. The pneumatic force is calculated using averaged pressure values over the top wall of the shock absorber, accounting for the fact that the jet through the orifice impacts the central portion of the upper wall and also accounting for the strong unsteadiness in the circulating flow reaching the top of the shock absorber. The pneumatic pressure plot follows the approximate shape of the theoretical pressure increase expected of an ideal gas under compression.

The hydraulic force, on the other hand, is less straightforward to calculate [20], as it cannot be directly measured from the available parameters in simulations but is rather calculated using Eq. 3.

$$F_h = \rho \frac{A_h^3}{2C_D^2 A_0^2} \dot{s} |\dot{s}|, \quad (3)$$

where A_h is the hydraulic area, C_D is the discharge coefficient, A_0 is the orifice area, and $|\dot{s}|$ is the stroke rate or inlet velocity in the case of the static mesh in the present simulation.

In addition, the discharge coefficient C_D is an influential parameter in the hydraulic force calculation. It is calculated using Eq. 4.

$$C_D = \frac{Q_{experimental}}{Q_{theoretical}} = \frac{\dot{m}}{\rho A_0 u_{max}}, \quad (4)$$

Where $\dot{m}$ is the mass flow through the orifice, and u_{max} is the maximum velocity at the point of venna contracta, downstream of the orifice. The maximum velocity is taken as maximum value from probes placed downstream of the orifice as the location of the venna contracta can show some dependency on the stroke rate.

Further validation of high-fidelity multiphase simulation was performed by monitoring the mass conservation of the oil and gas phases, as shown in Figs. 8b and 8c for oil and gas respectively. This is important to ensure that the mesh is fine enough to resolve the smaller bubbles when the violent multiphase mixing process takes place. The oil phase is checked against the calculated rise in mass due to the inflow implemented to simulate the stroke rate, where it is shown in Fig. 8b that there is a nearly perfect match between the calculated and actual oil mass present in the simulation. On the other hand, the gas phase does show a slight loss of mass towards the end of the stroke in Fig. 8c, when the mixing is most vigorous. However, the percentage decrease of gas mass is less than 0.001 %, and thus insignificant for the purposes of this study.

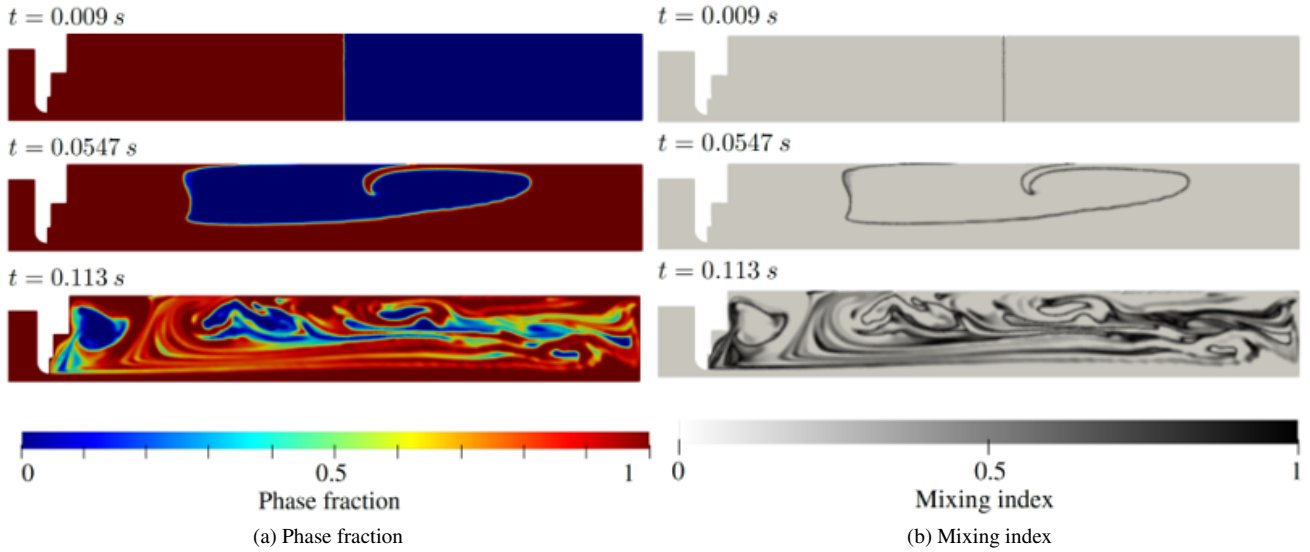


Figure 9: Multiphase contour plots.

The phase fraction contours in Fig. 9a showing the internal multiphase interaction at three different time values, clearly demonstrate the complexity of the interaction as the stroke progresses. The applied stroke rate causes the oil to rise through the orifice axis reaching the top wall. Then, The oil is deflected towards the side walls encapsulating the gas and creating a large recirculation zone extending along the full length of the upper chamber. The severity of the phase mixing increases as the oil breaks down the main gas mass into smaller bubbles and as those bubbles impinge on the orifice jet and the shear layers on its sides, causing the gas bubble to breakdown and disperse further as parts of it get convected with the jet. Some of the gas is pushed into a smaller recirculation region on the wall side of the orifice plate. To facilitate quantitative assessment of multiphase mixing, different forms of a mixing quality index have been developed [21, 22, 23]. It is defined here according to Eq. 5.

$$MI = \Sigma(1 - \sqrt{(2\alpha - 1)^2}) \quad (5)$$

The mixing index, Eq. 5 allows the interface between the two phases to be visualised at a given α value, as seen in Fig. 9b for $\alpha = 0.5$. In addition, a total interface area or length can be calculated by multiplying the mixing index value by a representative local cell area or length, as in Fig. 10. Which enables the assessment of multiphase mixing severity in different shock absorber designs systematically.

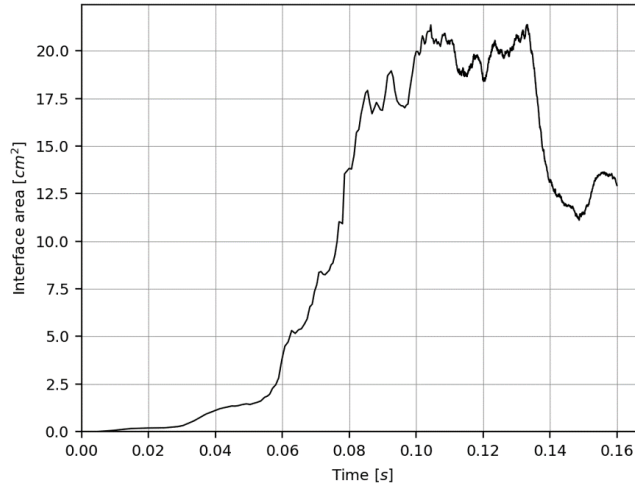


Figure 10: Multiphase mixing index time history.

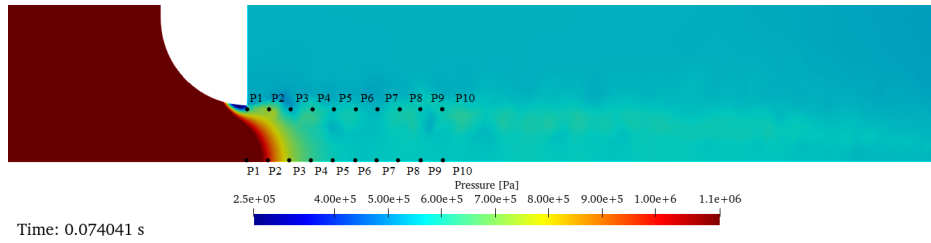


Figure 11: Multiphase drop test simulation – pressure field around the orifice with data probes position displayed, Fig. 8 in Sheikh Al-Shabab *et. al.* [24].

Finally, it is also possible to monitor low pressure regions and their extent, as illustrated in Fig. 11 to evaluate the susceptibility to cavitation in various operating conditions. Detecting cavitation is important because of the excessive forces often associated with the collapse of cavitation bubbles, and their potential to reduce the lifespan of internal components. While cavitation modelling is not included in the present multiphase simulation, regions of low pressure or severe pressure gradients in the domain could indicate the need for a dedicated cavitation simulation under these conditions. These simulations can then be run on a dedicated high order multiphysics solver [25] to investigate the full extent of cavitation impact on cases of interest.

5 Conclusions

The internal dynamics of oleo-pneumatic shock absorber were simulated using a geometry developed from the literature. A modular approach was used to tackle the complex multiphysics problem, starting with the most critical aspects of shock absorber flow physics; the turbulence development downstream of the main orifice and the multiphase mixing between oil and gas. The discussion focused on validating the simulations against drop test measurements and some standard physical results, in addition to assessing quantitative parameters that can be incorporated within an automated numerical framework. Low-fidelity simulations, based on a dynamic systems model of the shock absorber, were used to provide quick return simulations to explore the design space and provide case specific boundary conditions for higher fidelity simulations.

Future work will focus on adding additional areas of the relevant multiphysics affecting shock absorber operation, through simulations focusing on cavitation, heat transfer, compressibility etc. Furthermore, low-fidelity simulations will be calibrated to modify properties and parameters with stroke displacement, in order to optimise their performance over the entire stroke.

6 Acknowledgements

This research was funded by Innovate UK grant number 10002411.

7 Data Access

The underlying data can be accessed using the following DOI: 10.17862/cranfield.rd.24755505.

References

- [1] B. Milwitzky and F. E. Cook, "Analysis of landing-gear behavior," *NACA Technical Note*, no. TN-2755, 1952.
- [2] N. Currey, *Aircraft Landing Gear Design: Principles and Practices*. AIAA education series, American Institute of Aeronautics and Astronautics, 1988.
- [3] R. K. Schmidt, *The design of aircraft landing gear*. SAE International, 2021.
- [4] A. A. Sheikh Al-Shabab, D. Vitlaris, Z. Lin, B. Grenko, P. Tsoutsanis, A. F. Antoniadis, and M. Skote, "Numerical investigation of oleo-pneumatic shock absorber: Setup and validation," *WCCM-ECCOMAS2020*.
- [5] A. A. S. Al-Shabab and P. G. Tucker, "Toward active computational fluid dynamics role in open jet airfoil experiments design," *AIAA Journal*, vol. 56, no. 8, pp. 3205–3215, 2018.
- [6] A. S. Al-Shabab and P. Tucker, "Rans prediction of open jet aerofoil interaction and design metrics," *The Aeronautical Journal*, vol. 123, no. 1266, pp. 1275–1296, 2019.
- [7] ANSYS Fluent, Release 2019 R2, "Theory guide," 2019.
- [8] F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA journal*, vol. 32, no. 8, pp. 1598–1605, 1994.
- [9] H. G. Weller, G. Tabor, H. Jasak, and C. Fureby, "A tensorial approach to computational continuum mechanics using object-oriented techniques," *Computers in physics*, vol. 12, no. 6, pp. 620–631, 1998.
- [10] H. Jasak, *Error analysis and estimation for the finite volume method with applications to fluid flows*. PhD thesis, Imperial College London, 1996.
- [11] T. Holzmann, "Mathematics, numerics, derivations and openfoam@," *Loeben, Germany: Holzmann CFD*, 2016.
- [12] R. Issa, "Solution of the implicitly discretised fluid flow equations by operator-splitting," *Journal of Computational Physics*, vol. 62, no. 1, pp. 40–65, 1986.
- [13] S. Patankar and D. Spalding, "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows," *International Journal of Heat and Mass Transfer*, vol. 15, no. 10, pp. 1787–1806, 1972.
- [14] F. Nicoud and F. Ducros, "Subgrid-scale stress modelling based on the square of the velocity gradient tensor," *Flow, turbulence and Combustion*, vol. 62, no. 3, pp. 183–200, 1999.
- [15] A. A. Sheikh Al-Shabab, B. Grenko, D. Vitlaris, P. Tsoutsanis, A. F. Antoniadis, and M. Skote, "Numerical investigation of orifice nearfield flow development in oleo-pneumatic shock absorbers," *Fluids*, vol. 7, no. 2, p. 54, 2022.
- [16] P. Bradshaw, D. H. Ferriss, and R. Johnson, "Turbulence in the noise-producing region of a circular jet," *Journal of Fluid Mechanics*, vol. 19, no. 4, pp. 591–624, 1964.
- [17] A. Townsend, *The structure of turbulent shear flow*. Cambridge university press, 1980.
- [18] J. Jimenez, "Free shear flows," *Madrid: Universidad Politecnica de Madrid. Retrieved from torroja. dmt. upm. es*, 2013.
- [19] P. Tsoutsanis, X. Nogueira, and L. Fu, "A short note on a 3d spectral analysis for turbulent flows on unstructured meshes," *Journal of Computational Physics*, vol. 474, p. 111804, 2023.
- [20] Y. W. Ding, X. H. Wei, H. Nie, and Y. P. Li, "Discharge coefficient calculation method of landing gear shock absorber and its influence on drop dynamics," *Journal of Vibroengineering*, vol. 20, no. 7, pp. 2550–2562, 2018.
- [21] N. Solehati, J. Bae, and A. P. Sasmito, "Numerical investigation of mixing performance in microchannel t-junction with wavy structure," *Computers & Fluids*, vol. 96, pp. 10–19, 2014.
- [22] S. Wang, X. Huang, and C. Yang, "Mixing enhancement for high viscous fluids in a microfluidic chamber," *Lab on a Chip*, vol. 11, no. 12, pp. 2081–2087, 2011.
- [23] Q. Nguyen and D. V. Papavassiliou, "Quality measures of mixing in turbulent flow and effects of molecular diffusivity," *Fluids*, vol. 3, no. 3, p. 53, 2018.
- [24] A. Sheikh Al Shabab, B. Grenko, D. Vitlaris, P. Tsoutsanis, A. Antoniadis, and M. Skote, "Numerical investigation of oleo-pneumatic shock absorber: a multi-fidelity approach," in *ECCOMAS Congress 2022 - 8th European Congress on Computational Methods in Applied Sciences and Engineering*, ECCOMAS, 2022.
- [25] A. F. Antoniadis, D. Drikakis, P. S. Farmakis, L. Fu, I. Kokkinakis, X. Nogueira, P. A. Silva, M. Skote, V. Titarev, and P. Tsoutsanis, "Ucns3d: An open-source high-order finite-volume unstructured cfd solver," *Computer Physics Communications*, vol. 279, p. 108453, 2022.



DSPACE

<https://dspace.org/>

A modular multifidelity approach for multiphysics oleo-pneumatic shock absorber simulations

Sheikh Al-Shabab, Ahmed A.; Silva, Paulo A. S. F.; Grenko, Bojan; Tsoutsanis, Panagiotis; Skote, Martin

2024-12-03

Springer Nature

https://doi.org/10.1007/978-3-031-69035-8_8

Sheikh Al-Shabab AA, Silva PASF, Grenko B, et al., (2024) A modular multifidelity approach for multiphysics oleo-pneumatic shock absorber simulations. In: Proceedings of the Cambridge Unsteady Flow Symposium 2024, 4-5 March 2024, University of Cambridge, United Kingdom, pp. 137-151