Open-source High-fidelity Solvers Dedicated to Re-entry Analysis and Design for Demise

Vincent Casseau*, Daniel E. R. Espinoza and Thomas J. Scanlon
Centre for future air-space transportation technology (cFASTT), Aerospace Centre of Excellence (ACE) and James Weir Fluids Laboratory, Department of Mechanical and Aerospace Engineering
University of Strathclyde, 75 Montrose Street, G1 1XJ, Glasgow, UK
Emails: vincent.casseau@strath.ac.uk, daniel.espinoza@strath.ac.uk and tom.scanlon@strath.ac.uk

Richard E. Brown
Centre for future air-space transportation technology (cFASTT), Aerospace Centre of Excellence (ACE)
University of Strathclyde, 75 Montrose Street, G1 1XJ, Glasgow, UK
Email: richard.brown@strath.ac.uk

Summary

The success of any high-fidelity simulation in the fields of re-entry analysis and design for demise is dictated by the capacity the numerical codes have to be physically accurate and computationally efficient. Hypersonic hybrid hydrodynamic-molecular gas flow solvers should provide an answer to these two requirements. Whilst currently available hybrid solvers are mostly proprietary codes, the University of Strathclyde is developing an open-source CFD-DSMC hybrid code within the OpenFOAM framework that employs the existing application *dsmcFoam* and the new two-temperature CFD solver *hy2Foam*. The latest developments of the *hy2Foam* solver and the hybrid code are described in this paper.

Keywords: *hypersonics, open-source, CFD, two-temperature solver, DSMC, hybrid, re-entry analysis, design for demise.*

1 Introduction

The success of any high-fidelity simulation in the fields of re-entry analysis and design for demise is dictated by two essential requirements. The first one, computational efficiency, should allow the study of various design points within reasonable timescales to assess the aerothermodynamic performance of a body of interest along its trajectory path and to explore various coupling techniques with other hypervelocity numerical tools. The second feature, physical accuracy, is of vital importance whether the objective is to preserve the integrity of the body or not.

Within the open-source C++ fluid dynamic toolbox OpenFOAM,[1] the direct simulation Monte-Carlo (DSMC) application *dsmcFoam* has proven over the course of the past ten years to be mature enough for simulating a wide range of applications,[2–4] thus delivering trustworthy results in lower-density reacting environments. For altitudes below 90 km however, the computational expense of this approach becomes prohibitive. A new two-temperature conventional computational fluid dynamics (CFD) solver, *hy2Foam*, has been developed within the same OpenFOAM platform[5,6] to simulate hypersonic planetary atmospheric entries, hence addressing the re-entry problem for the lower range of altitudes corresponding to the continuum or continuum-transition regimes. Recently, these two approaches have been united to form a hypersonic hybrid hydrodynamic-molecular gas flow solver[7] in an attempt to ultimately combine the best of both worlds, thus satisfying the two initial prerequisites. This solver is thought to be the first open-source multi-temperature hypersonic hybrid code since most of currently available hybrid solvers reviewed[8] are essentially in-house codes.

The present work reviews the current progress made in the field of open-source high-fidelity hypersonic computations at the University of Strathclyde.

2 Methodology

2.1 *Open-source Navier-Stokes solver*

The *hy2Foam* two-temperature CFD solver computes the non-equilibrium Navier-Stokes-Fourier equations distinguishing the trans-rotational energy pool from multiple vibro-electronic energy pools. The different energy transfer processes are executed using well-established models such as the Landau-Teller equation with the Millikan-White correlation, or the vibrational-vibrational energy exchange model of Knab. The chemistry-vibration coupling is handled either by the Park TTv model or by the coupled-vibration-dissociation-vibration (CVDV) model.[9] One distinct novel feature of *hy2Foam* is the use of chemical rates derived from Quantum Kinetic theory[2] with the CVDV model which promotes a better consistency with the *dsmcFoam* code than in using Park's rates with the Park TTv model.

Species and mixture thermal properties are recovered using one of the several models implemented, for instance the Blottner and Eucken formulas and the Armaly and Sutton mixing rule. The diffusion fluxes are given by the generalisation of Fick's law for a mixture.

The numerics of the code relies on the central-upwind scheme of Kurganov and Tadmor[10] and the traditional first-order Smoluchowski temperature jump and Maxwell velocity slip are still available for wall boundary treatment.

2.2 *Hybrid CFD-DSMC code*

The hybrid code is using the *hy2Foam* and *dsmcFoam* solvers and makes the use of a single mesh. The first step consists in running a CFD computation with *hy2Foam* to obtain a first estimate of the entire flow-field. The calculation of the local gradient-length Knudsen number then enables the split of the computational domain into CFD and DSMC regions, thus creating an interface. The transfer of information across this interface is set to follow the state-based coupling approach. It requires the generation of a buffer zone on each side of the interface. The so-called DSMC buffer zone represents the extension of the DSMC region onto the CFD region, and the full DSMC region is used as the CFD buffer zone.

The DSMC equivalent particles allow the reconstruction of the CFD macroscopic fields in the CFD buffer zone. Conversely, new DSMC simulators are created into the DSMC buffer zone according to an appropriate distribution function. The Maxwellian, Chapman-Enskog and generalised Chapman-Enskog distributions have been implemented and tested for different gases.

3 Results

3.1 *Conventional CFD*

The implementation of *hy2Foam* has proven to be successful for various zero- and multi-dimensional case scenarios.[5,6] Among them is the Mach 20 flow around a circular cylinder illustrated in Figure 1. The fluid was composed of a binary reacting mixture of nitrogen molecules and atoms. The *hy2Foam* code has shown to provide a satisfactory estimation of the aerodynamic loads (less than 1.5 % difference vs. *dsmcFoam*) and thermal loads in reasonable agreement with *dsmcFoam* despite the presence of regions of thermal non-equilibrium within the flow-field.

The solver is currently being used to assess the loads during the demise of the satellite depicted in Figure 2.

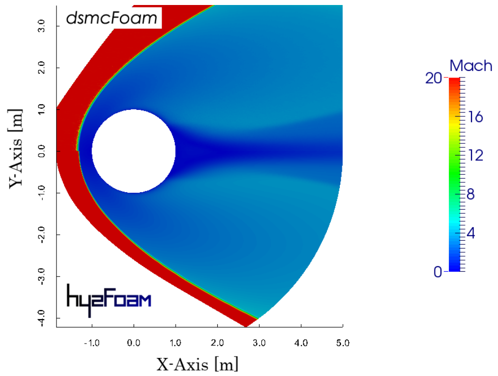


Figure 1: Flow past a Mach 20 cylinder (DSMC: top, CFD: bottom)

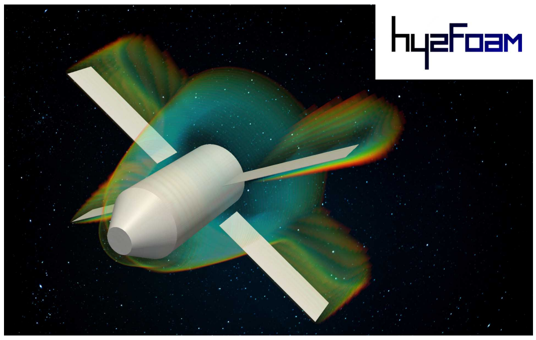
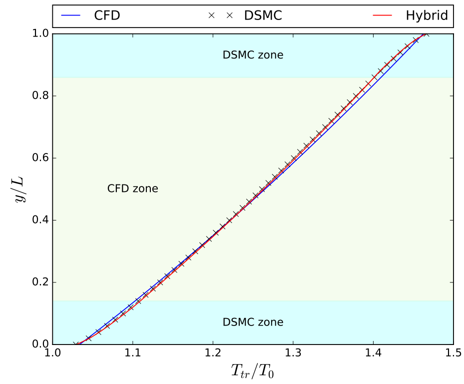


Figure 2: Mach 27 satellite demise at 90 km altitude

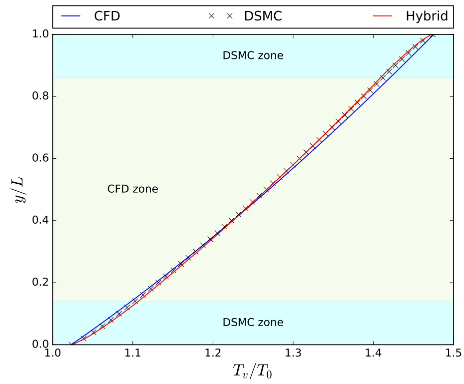
3.2 *Hybrid solver*

Using the solvers *hy2Foam* and *dsmcFoam*, the open-source hybrid code capabilities have been tested considering the one-dimensional case scenarios of a heat transfer flow and Couette flow.[7] The focus is set here on the Couette flow. The set-up consisted in two infinite flat plates separated by 1 m. The CFD and DSMC regions were fixed. The CFD region occupies the centre of the channel and spans from $y = 0.14$ m to $y = 0.86$ m. The temperature of the lower and upper walls are 2,000 K and 3,000 K, respectively. In addition, the lower wall is at rest while the upper wall is set to move at a velocity of 300 m s^{-1}. The fluid density is derived from a body-length Knudsen number equal to 0.05.

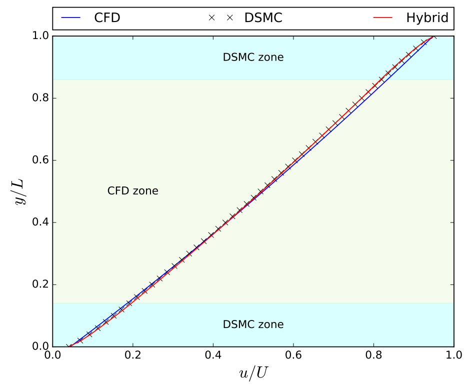
Figure 3 shows the CFD, DSMC and hybrid results for a fluid composed of nitrogen. Due to the relatively high temperature imposed at the walls, the nitrogen molecules will be vibrationally-excited. For this reason, the generalised Chapman-Enskog distribution is retained in the hybrid solver. It can be seen for both temperatures and velocity profiles that the CFD and DSMC solutions present some discrepancies in the near-wall region, discrepancies inherent to the Knudsen layer that can not be captured accurately with the CFD solver. The hybrid code overcomes this difficulty and produces very similar results to *dsmcFoam*.



(a) Trans-rotational temperature



(b) Vibrational temperature



(c) Velocity

Figure 3: CFD, DSMC and hybrid results for the Couette flow simulation.

4 Conclusions

The open-source platform OpenFOAM has been enhanced with two new solvers capable of dealing with the harsh conditions experienced during planetary atmospheric entry. *hy2Foam* is a two-temperature CFD solver that incorporates several state-of-the-art models and makes novel use of Quantum-Kinetic chemical rates. It has been verified and validated for multiple zero- and multi-dimensional cases, in particular for a binary reacting mixture around a Mach 20 cylinder. The hybrid CFD-DSMC solver combines *hy2Foam* and the existing *dsmcFoam* application following a state-based approach. The solver has been tested using a one-dimensional Couette flow and has proven to be in excellent agreement with the DSMC code. Future work will consist in assessing the performance of the hybrid solver and extending its capabilities to multi-dimensional and multi-species flows.

References

[1] OpenFOAM website. http://www.openfoam.org/, accessed: 2 August 2016.

[2] Scanlon, T. J. et al. Open source Direct Simulation Monte Carlo (DSMC) chemistry modelling for hypersonic flows. *AIAA Journal* **53**(6), 1670–1680 (2015).

[3] Palharini, R. C. *Atmospheric Reentry Modelling Using an Open-Source DSMC Code*. PhD thesis, University of Strathclyde, Glasgow, (2014).

[4] White, C. et al. Numerical and Experimental Capabilities for Studying Rocket Plume-Regolith Interactions. In *Proceedings of the 30th Rarefied Gas Dynamics Conference* (AIP, Victoria BC, Canada, July 10-15, 2016).

[5] Casseau, V., Palharini, R. C., Scanlon, T. J., and Brown, R. E. A Two-Temperature Open-Source CFD Model for Hypersonic Reacting Flows, Part One: Zero-dimensional Analysis. *Aerospace* **3**(4), 34 (2016).

[6] Casseau, V., Scanlon, T. J., John, B., Emerson, D. R., and Brown, R. E. Hypersonic Simulations using Open-Source CFD and DSMC Solvers. In *Proceedings of the 30th Rarefied Gas Dynamics Conference* (AIP, Victoria BC, Canada, July 10-15, 2016).

[7] Espinoza, D. E. R., Casseau, V., Scanlon, T. J., and Brown, R. E. An Open Source Hybrid CFD-DSMC Solver for High-Speed Flows. In *Proceedings of the 30th Rarefied Gas Dynamics Conference* (AIP, Victoria BC, Canada, July 10-15, 2016).

[8] Schwartzentruber, T. and Boyd, I. D. Progress and future prospects for particle-based simulation of hypersonic flow. *Progress in Aerospace Sciences* **72**, 66–79 (2014).

[9] Marrone, P. V. and Treanor, C. E. Chemical relaxation with preferential dissociation from excited vibrational levels. *Physics of Fluids* **6**(9), 1215–1221 (1963).

[10] Kurganov, A., Noelle, S., and Petrova, G. Semi-discrete central-upwind schemes for hypersonic conservation laws and Hamilton-Jacobi equations. *SIAM J. Sci. Comput.* **23**(3), 707–740 (2001).