23rd Australasian Fluid Mechanics Conference - 23AFMC
Sydney, Australia
4 – 8 December 2022
Paper No: AFMC2022-92

Computational flow modelling of the T4 free-piston driven reflected shock tunnel

Tamara Sopek[1*], Fabian Zander[1], Wilson Y.K. Chan[2] and Peter A. Jacobs[2]

[1]University of Southern Queensland, [2]The University of Queensland
*mailto: tamara.sopek@usq.edu.au

Abstract

Free-piston driven shock tunnels, such as the T4 tunnel that is the focus of this study, are a class of impulse facilities able to cover a wide range of flight Mach numbers, which have been used for decades to study hypervelocity flows at true total enthalpies and flow speeds. Their test gas is generated in a succession of flow processes involving complex and unsteady wave processes which are modified by viscous and high-temperature gas effects. A complete transient and spatial characterisation of the test gas properties is crucial for accurate interpretation of the experimental data obtained. However, experimental characterisation of these highly energetic shock tunnel flows is arduous and typically only limited measurements are performed. Routine experimental data analysis is generally based on quasi-steady state calculations coupled with basic pressure measurements to deduce test gas properties. This simplified methodology introduces unquantified errors between these estimated and actual test gas properties, and thus propagated errors in experimental data analysis. The present paper outlines the use of transient one-dimensional and axisymmetric numerical models for the full facility flow reconstruction to obtain more accurate test gas properties. The quasi-one-dimensional model is used to calculate the transient flow history in the driver tube, and serves as an inflow to the axisymmetric calculations of the downstream region. These simulations are validated through comparison with experimental data obtained from the T4 facility. The comparison shows that the shock speeds for two out of three locations were simulated with sufficient accuracy leading to the conclusion that the numerical modelling is adequately capturing relevant fluid processes for most of the shock tube length. The present work is the initial part of a planned investigation of a full facility flow reconstruction where instead of the one-dimensional modelling of the driver section, a higher fidelity moving mesh approach will be used to capture the piston motion.

1 Introduction

Free-piston driven shock tunnels are high enthalpy facilities capable of reproducing flight conditions relevant for Earth re-entry or scramjet engine operation. As such, these facilities provide a unique environment for the development of next generation space vehicles and scramjet technology. Full and accurate characterisation of the test flow in these facilities is crucial for any experimental investigation to obtain high quality measurement data, and for the numerical modelling community for validation of their computational models. However, the harshness of the flows produced by these facilities makes it difficult for physical measurements to be taken. Instead, simplified state-to-state fluid computations are typically used in conjunction with basic pressure measurements to estimate the flow properties supplied by these facilities. This simplified method of flow reconstruction results in an unknown level of discrepancies compared to the actual properties seen in the facility.

Full facility modelling is important as it enables us to capture and study a variety of mechanisms that occur during the flow development along the length of the facility, such as driver gas contamination, complex interactions of the waves, nozzle flow, etc. Previous simulations of similar facilities have been achieved, however, they either lack modelling of the piston motion (Goozée 2006), or modelled a smaller (McGilvray 2013) or different type of facility (Gildfind 2018a, Gildfind 2018). Additionally, according to authors' knowledge, this work is the first time that the axisymmetric simulations of the shock tube for T4 are initialised from the compression tube, whereas previous works had axisymmetric simulations starting at some point downstream of the primary diaphragm.

This study attempts to correctly model all the transient processes by using a hybrid analysis which includes coupled one-dimensional and axisymmetric numerical models (Gildfind 2018a). Such a hybrid method simplifies and reduces the cost of calculations compared to a full facility modelling using viscous and turbulent axisymmetric numerical analysis of the chemically reacting flow. Instead, the hybrid method has the computationally cheap quasi-one-dimensional Lagrangian code L1d as the first step. This code is able to capture free-piston motion and other longitudinal wave processes which are of essence for both the driver and driven tubes of the shock tunnel facility. However, as a quasi-one-dimensional code, L1d is not able to capture detailed fluid phenomena and instead it relies on empirical loss factors, hence it is used as the first step only. The result of this analysis is a computed flow history near the end of the compression tube, which is subsequently used as an inflow to the axisymmetric analysis of the facility. Our analysis showed the need to include the part of the compression tube in the axisymmetric simulations to have a better representation of the flow processes in the facility.

The simulations are validated through comparison with experimental measurements obtained at several locations along the facility length. Numerical results of the full facility axisymmetric simulations are then analysed and compared with that from the quasi-one-dimensional calculation. This comparison thus illustrates whether the simplified methodology can be used on a regular basis.

2 Experimental facility

The experimental facility that is the object of this study is the T4 free-piston driven reflected shock tunnel of the Centre for Hypersonics at The University of Queensland. As shown in Fig. 1, T4 consists of five main sections: the reservoir, the compression tube, the shock tube, a convergent–divergent nozzle, and the test section. The experimental data used for comparison in this work was obtained with a Mach 7 nozzle in the investigation of using a nitrogen driver for low-enthalpy conditions (Chan 2021).

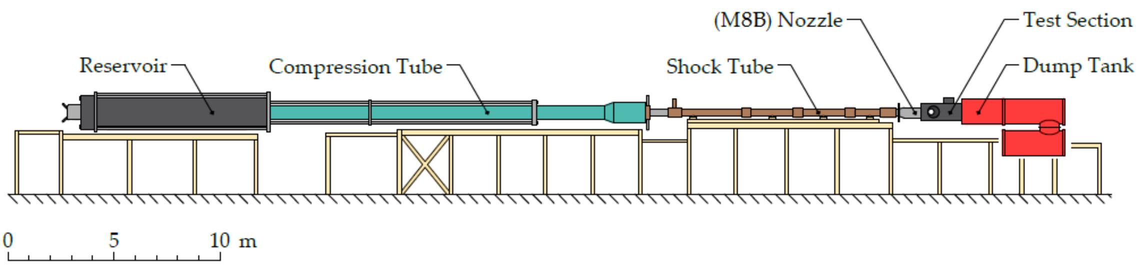

Figure 1. T4 free-piston driven shock tunnel. Reproduced from Doherty (2013).

The operation of the T4 free-piston shock tunnel consists of several significant processes. Prior to a test, the compression and shock tube are filled with driver and test gas respectively, while the reservoir is filled with high-pressure air. The nozzle, test section and dump tank are evacuated to a 130 Pa vacuum pressure. On firing, the high-pressure reservoir gas drives the piston through the compression tube. The piston gains kinetic energy as it travels along the compression tube, subsequently passing it to the driver gas as it slows down towards the end of the compression tube. This results in a nearly adiabatic compression of the driver gas until the burst pressure of the primary diaphragm is reached. Following the diaphragm opening, the large pressure differential between the driver gas and test gas drives a shock wave through the shock tube. This shock wave compresses and accelerates the test gas towards the nozzle, and upon reaching the end of the shock tube it reflects back upstream and stagnates the test gas. The result is a region of hot, pressurised gas which then expands through a convergent-divergent nozzle to the desired test conditions.

During the experiments, measurements of compression tube pressure, stagnation pressure, nozzle exit Pitot pressure and incident shock speed are typically obtained. The Mach 7 nozzle in T4 was used in the referenced experimental study, and Pitot pressure measurements were obtained within the uniform core flow region, approximately 130 mm downstream of the nozzle exit. More information on the experimental procedure can be found in Chan (2021).

3 Overview of simulations

Complete modelling of the free-piston driven shock tunnel facility demands time-accurate calculations to capture transient wave processes. This results in extensive calculations increasing quickly with facility size due to the increase in grid sizes and run times needed to complete the simulations. The hybrid analysis breaks up the process into several smaller simulations, each focussing on distinct flow processes, such as piston dynamics or nozzle flow development. The process is separated into following calculations: 1. simulation of the free piston compression and primary diaphragm rupture using a quasi-1-D code L1d; 2. simulation of the last part (0.16m) of the compression tube post primary diaphragm opening, shock tube flow and simulation of the flow expansion through the nozzle using the Eilmer code. This hybrid methodology is illustrated in Fig. 2. The simulations were performed using the facility geometry, and the initial fill and boundary conditions taken from the study in Chan (2021).

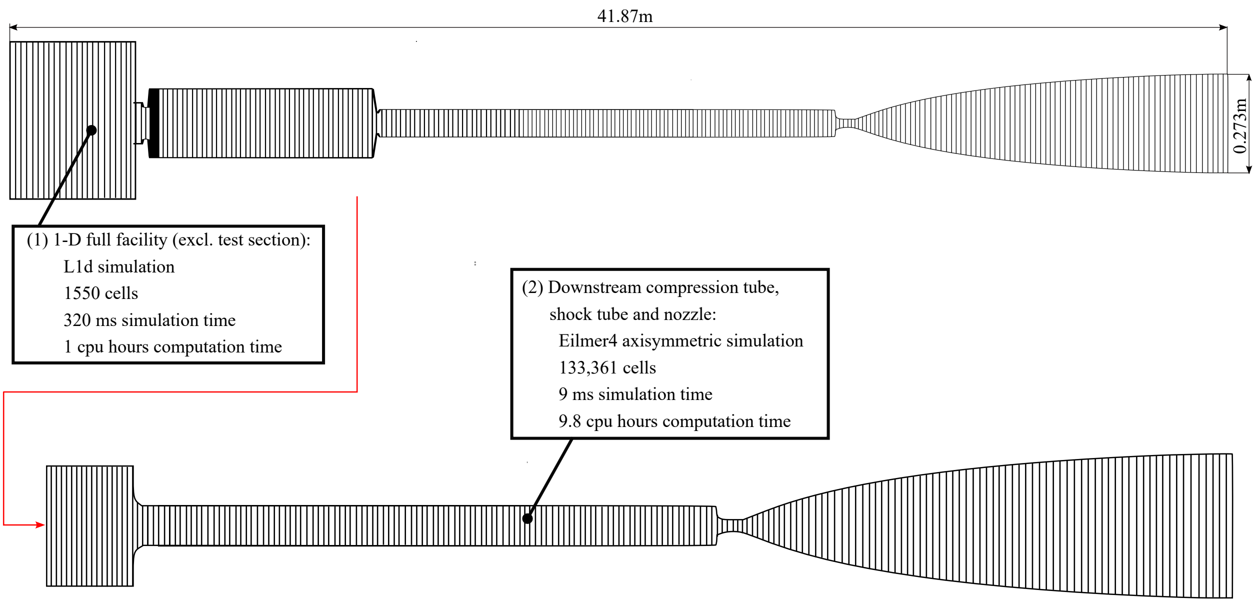

Figure 2. Hybrid computational scheme. Not to scale.

4 L1d

The first stage is the complete facility simulation using a quasi-one-dimensional Lagrangian solver L1d (Jacobs, 1994) at a significantly lower computational cost compared to a full axisymmetric simulation [1]. This 1-D model, which accurately captures longitudinal wave processes and includes the piston dynamics, is used to calculate the transient flow history from piston launch to primary diaphragm rupture. The solution is then used as the inflow to a higher-fidelity axisymmetric calculation of the downstream section. Thus, accuracy of the 1-D model directly impacts the accuracy of the final test flow. The model was tuned to match the experimental measurements as closely as possible. The simulations of the reservoir (500 cells), compression tube (500 cells), shock tube (500 cells) and test section (50 cells) were performed with viscous effects. Loss regions were applied to launcher slots, primary diaphragm, shock tube and nozzle contraction and throat with minor-loss coefficients set as 25.0, 0.8, 1.5 and 0.5, respectively. Burst pressures were set at 25.6 MPa for the primary (steel) diaphragm and 1.0 MPa for the secondary (Mylar) diaphragm. Conditions of a history point set at 0.16 m upstream from the end of the compression tube was used as the inflow for the Eilmer simulations. This location was chosen based on the location of the piston after the opening of the primary diaphragm in the relation to the time at which the incident shock is passing the shock timing station.

As mentioned previously, the computed flow history at the location at the end of the compression tube at x=25.84 m is used as an inflow to the axisymmetric calculation. In the quasi-one-dimensional L1d calculations, minor-loss coefficients at the primary diaphragm and along the shock tube are applied. However, the axisymmetric simulation does not use such coefficients, but, instead, includes more directly the flow mechanics of oblique shocks and boundary layers. Also, the relevant physics are not adequately captured if initialising these calculations after the primary diaphragm. In particular, the initial process of ramming the driver gas through a large contraction and then through a poorly-opening steel diaphragm and into the shock tube causes a shock train in the upstream-end of the shock tube. This results in total pressure losses downstream of the contraction section. Therefore, the last part of the compression tube and the primary diaphragm station were included in the axisymmetric simulation to improve the modelling.

5 Eilmer 2D/axisymmetric tube calculations

The second stage of the hybrid method is the axisymmetric numerical calculation performed using the Eilmer4 Navier–Stokes compressible flow code (Jacobs 2016, Gibbons 2022). Time accurate flow properties extracted from the L1d simulation is used here as an inflow to the axisymmetric simulation of the downstream section. The facility sections were modelled employing structured grids refined towards the wall to resolve the boundary layer. Simulations assumed ideal gas behaviour for the driver gas and equilibrium chemistry for the test gas. The initial temperature of gas in both tubes was set to 296 K.

The results presented here were obtained using the grid with 70 radial cells for compression tube and 40 radial cells for the shock tube, clustered towards the tubes' walls. The nozzle had a grid resolution of 300 cells axially, 40 cells radially, clustered toward the wall and throat.

Turbulence was assumed for the entire simulation and the turbulence model used was the k-ω turbulence model of Wilcox (2006). Results were recorded at measurement locations along the facility length, including shock timing stations, nozzle-supply region and nozzle exit, to enable comparison with the experimentally measured data.

[1] The L1d and Eilmer codes are part of the Gas Dynamics Toolkit and freely available https://gdtk.uqcloud.net

6 Results and analysis

Simulations in this study were performed using conditions of T4 shot 11316 where the facility was operated with a 100% argon driver. Reservoir and compression tube fill pressures were 1.65 MPa and 27.4 kPa, respectively. The shock tube was filled with air at 270 kPa pressure resulting in total enthalpy of approximately 1.6 MJ/kg. This condition results in a "a severely under-tailored condition" as stated in Chan (2021). Figure 3 shows *x-t* (distance–time) diagrams from the L1d4 simulations for this experiment. Logarithmic pressure is mapped into the diagram to highlight the wave processes.

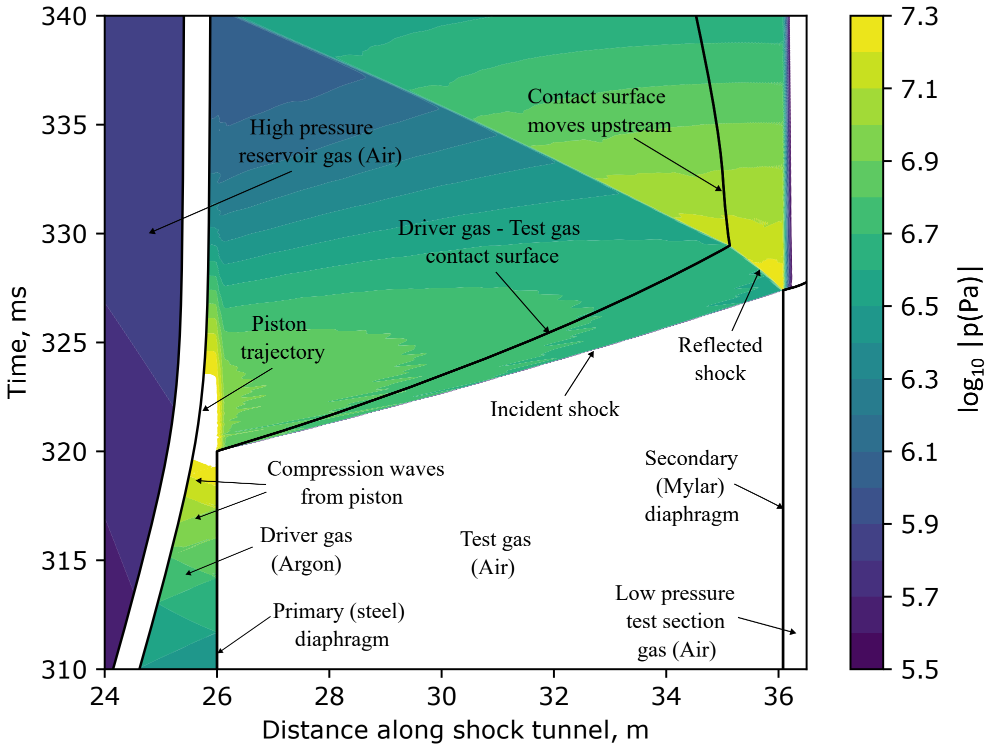

Figure 3. T4 *x-t* diagram showing wave processes for the pure argon driver test (shot 11316).

Fig. 4 illustrates the shock train occurring after the primary diaphragm opening.

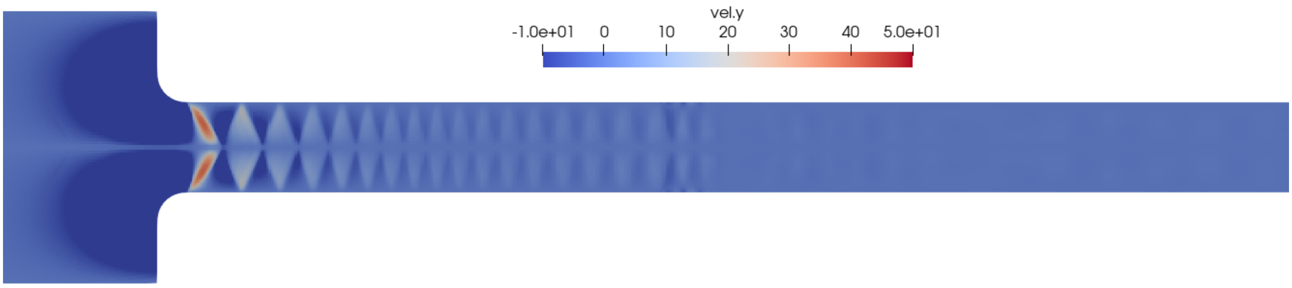

Figure 4. Upstream shock tube shock train.

Numerical results of both L1d and Eilmer codes are validated through comparison with the experimental measurements at the same spatial locations, as shown in Fig 5 for the shock timing stations. The shock speeds for all three sets of data are plotted at mid-positions between the shock timing stations, and between the final shock timing station and the nozzle-supply region location for the last shock speed. The initial conditions of the simulations were defined to match the filling conditions used in the experiments. The comparison shows that the L1d is not adequately simulating the incident shock speeds. There is a significant discrepancy between the L1d and experimental measurements. The results obtained with the Eilmer code, however, show a very good match with the experimentally measured values for the first two shock speeds, indicating that the relevant flow processes are modelled with sufficient accuracy. These results demonstrate that it is important for simulations to include the last length of the compression tube.

The reason for the discrepancy between axisymmetric and experimental results at x=35.09 m in Fig 5 is likely due to turbulence modelling and, maybe, lack of grid resolution. Imperfections in turbulence modelling may result in a boundary layer developed on the shock tube walls which is thinner than the actual boundary layer. This might translate into higher shock speed of numerical results towards the end of the shock tube. Also, the piston dynamics was not included in the reported simulations, which contributes to the discrepancy.

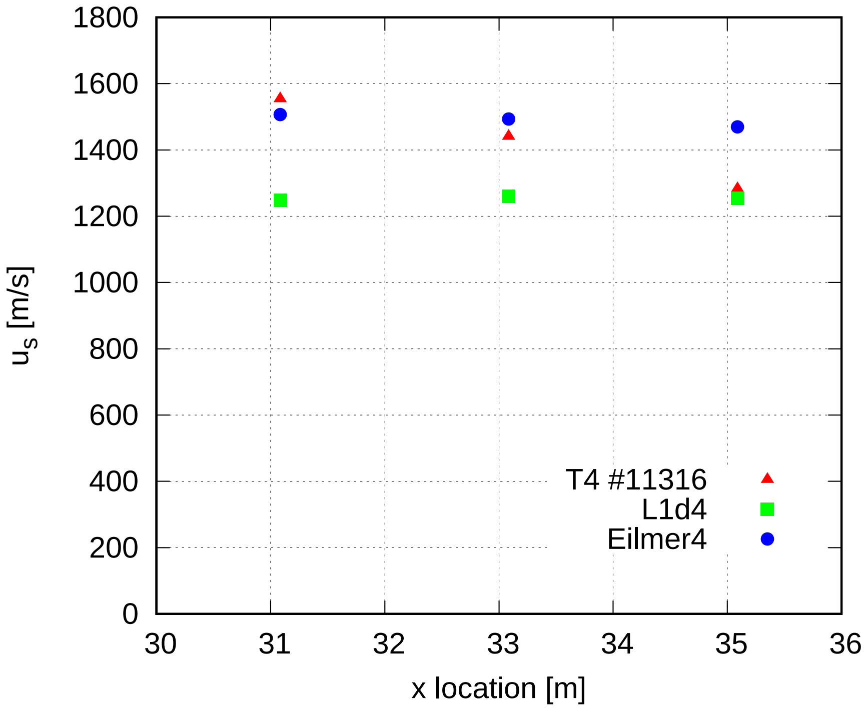

Figure 5. Comparison between the experimental and numerically obtained shock speeds in the shock tube.

The reason for the lower accuracy of L1d code is likely be attributed to the method used to account for total pressure losses that is based on using minor-loss coefficients. While this method results in reasonable accuracy of matching the experimentally recorded shock station data, higher fidelity Eilmer simulations show that a better agreement can be achieved toward the upstream-end of the shock tube. This leads to the belief that Eilmer results can be used for improvements in the method and/or the values of loss coefficients used in L1d. These results also indicate that, at the present moment, the axisymmetric calculations are worth pursuing considering the improvement in agreement with the experimentally measured values compared to the quasi-one-dimensional calculations.

The last step in completing the test flow reconstruction was an axisymmetric calculation of the T4 Mach 7 nozzle. The nozzle flow development is presented in Fig. 6, in terms of contours of Mach number (top half) and driver gas mass fraction (bottom half) for several different times, starting from t=0 (shock arrival at the throat inlet plane) to t=2.5 ms, and in 0.5 ms time increments. At t=0.5 ms, the test gas has entered the nozzle where regions of high Mach number are seen, while the shock tube shows arrival of the driver gas toward the nozzle supply region. Between t=0.5 and t=2.5 ms, the flow is being established within the nozzle, while simultaneously driver gas is being entrained into the test gas. The last time frame shows the established nozzle flow at targeted Mach 7 condition.

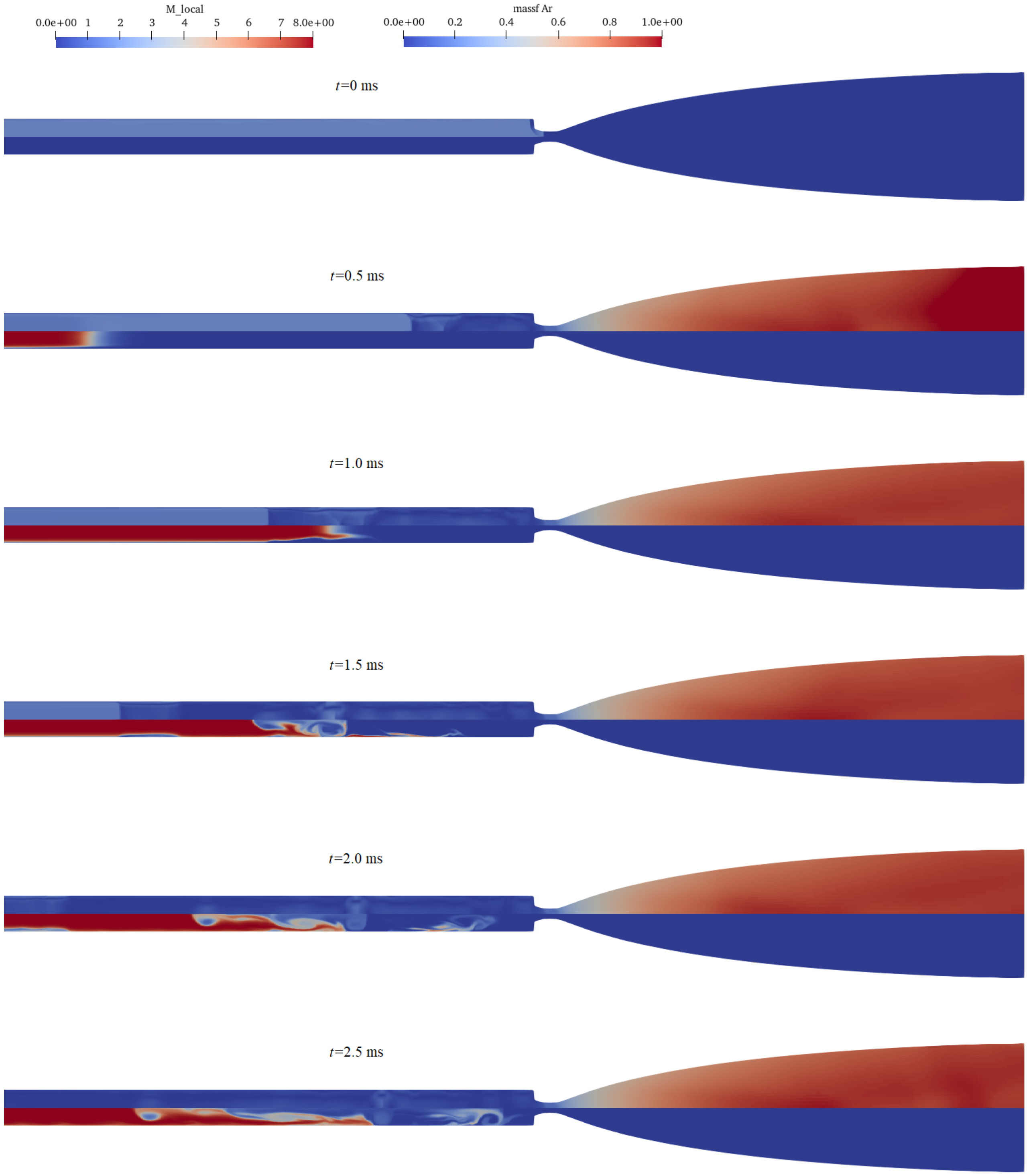

Figure 6. Eilmer4 2-D axisymmetric calculation of flow development in the T4 M7 nozzle. Upper half of each plot shows Mach number; lower half of each plot shows mass fraction of driver gas, Ar. Time frames from the instant of secondary diaphragm opening to *t*=2.5 ms.

7 Conclusions

This study examined a hybrid numerical modelling methodology for reconstructing free-piston driven shock tunnel test flows. This involved first simulating the full facility with the quasi-1-D Lagrangian L1d code. The result was used as an inflow to a higher-fidelity 2-D axisymmetric Eilmer4 model of the downstream section. By simulating the nozzle and test section in the final stage, the final test flow was obtained. These numerical results were validated through comparison with experimental measurements at several locations along the facility length.

The comparison shows that the shock speeds for two out of three locations were simulated with sufficient accuracy leading to the conclusion that the numerical modelling is adequately capturing relevant fluid processes for the most of the shock tube length. The comparison of the last shock speed value revealed discrepancy when reaching the nozzle-supply region of the shock tube. It is an ongoing work to improve modelling of that region as well. The results will be used for improvement of loss factors incorporated into L1d in effort to improve the agreement between sets of data. Nozzle simulations have shown that the Mach 7 flow has been established in the nozzle. These simulations have been able to reproduce driver gas entrainment in the test gas as the slug of test gas is leaving the shock tube.

8 Acknowledgements

Fabian Zander is funded by the Australian Research Council through the Discovery Early Career Researcher Award number DE200101674. This work was supported through computational resources of the University of Southern Queensland High Performance Computing (HPC) Fawkes.

References

Chan, W. Y. K., Whitside, R. W., Smart, M. K., Gildfind, D. E., Jacobs, P. A. and Sopek, T. 2021, Nitrogen driver for low-enthalpy testing in free-piston-driven shock tunnels, *Shock Waves Journal*, **31**, 541–550.

Doherty, L. J. 2013, *Experimental Investigation of an Airframe Integrated 3-D Scramjet at a Mach 10 Flight Condition*, The University of Queensland.

Gibbons, N. N., Damm, K. A., Gollan, R. J. and Jacobs, P. A. 2022, Eilmer: an Open-Source Multi-Physics Hypersonic Flow Solver, *Submitted to Computer Physics Communications*.

Gildfind, D. E., Jacobs, P. A., Morgan, R. G., Chan, W. Y. K. and Gollan, R. J. 2018, Scramjet test flow reconstruction for a large-scale expansion tube, Part 1: quasi-one-dimensional modelling, *Shock Waves Journal*, **28**, 877–897.

Gildfind, D. E., Jacobs, P. A., Morgan, R. G., Chan, W. Y. K. and Gollan, R. J. 2018, Scramjet test flow reconstruction for a large-scale expansion tube, Part 2: axisymmetric CFD analysis, *Shock Waves Journal*, **28**, 899–918.

Goozée, R. J., Jacobs, P. A., and Buttsworth, D. R. 2006, Simulation of a complete reflected shock tunnel showing a vortex mechanism for flow contamination, *Shock Waves Journal*, **15(3-4)**, 165–176.

Jacobs, P. A. 1994, Quasi-one-dimensional modeling of a free-piston shock tunnel, *AIAA Journal*, **32(1)**, 137–145.

Jacobs, P. A. and Gollan, R. J. 2016, Implementation of a Compressible-Flow Simulation Code in the D Programming Language, *Applied Mechanics and Materials*, **846**, 54–60.

McGilvray, M., Dann, A. G. and Jacobs, P. A. 2013, Modelling the complete operation of a free-piston shock tunnel for a low enthalpy condition, *Shock Waves Journal*, **23**, 399–406.

Wilcox, D. C. 2006, *Turbulence Modelling for CFD*, DCW Industries, Inc.