23rd Australasian Fluid Mechanics Conference - 23AFMC
Sydney, Australia
4 – 8 December 2022

Paper No: AFMC2022-440

Condensation Modelling of Expanding Cold Gas Jets during Hypersonic Retro-Propulsion Manoeuvres within the RETPRO Project

Tamas Bykerk[1*], Stefan Fechter[1], Daniel Kirchheck[2] and Sebastian Karl [1]

[1] The German Aerospace Center (DLR), Bunsenstr. 10, Göttingen, Lower Saxony, 37073, Germany
[2] The German Aerospace Center (DLR), Linder Höhe, Cologne, North Rhine Westphalia, 51147, Germany
*mailto: tamas.bykerk@dlr.de

Abstract

The RETPRO project (Validation of Wind Tunnel Test and CFD Techniques for Retropropulsion), as part of ESA's Future Launchers Preparatory Programme, aims at preparing the tools, necessary for a reliable design and simulation of future rocket launchers or spacecraft. A particular focus is assigned to vertical take-off and landing configurations using retro propulsion as part of their control concept for entry, descent, and landing manoeuvres. Wind tunnel tests and computational fluid dynamics are used to generate a comprehensive aerodynamic database, which is required for flight dynamics simulations, enabling mission and performance analyses of possible future launcher designs. Wind tunnel tests are conducted in the DLR Cologne H2K facility, with room temperature dry air ejected through selected nozzles to simulate the exhaust plume. Condensation effects might occur in the plume due to the low static freestream pressure at Mach 7, combined with the expanding flow in the nozzle. This paper presents results from numerical investigations including a vapour-equilibirum model which evaluate the potential influence of plume condensation on measured data in the wind tunnel. A qualitative comparison between experimental and numerical results is presented through Schlieren photographs. Condensation was observed in the numerical results, causing the flow path in and around the plume to be altered. Surface pressure coefficients in the condensation case were observed to be approximately 5% lower than when using the standard ideal gas model. Finally, the shock stand off distance was reduced, but not significantly. The comparison with tunnel data was therefore more-or-less the same as with the ideal gas model and the use of the condensation model was not deemed necessary for subsequent computations.

1 Introduction

The recent success of several commercial launchers in landing, recovering and relaunching complete main stages has renewed the interest in the study of re-usable space transportation concepts. As shown by several systems, a restriction to the recovery of the first stage and the application of retro-propulsion for landing appears to be a promising concept for low-cost, robust and flexible vehicles. Commercial launch vehicles, such as the SpaceX Falcon 9, has inspired the development of the vertical take-off and landing (VTVL) re-usable launch vehicle (RLV). The recovery of the first stage is achieved through strategic retro-propulsion burns, control surface deflections and deployable landing legs to accomplish a precision landing. The nature of this trajectory introduces complex flow topologies through a large range of Mach numbers, requiring a strong understanding of the vehicle aerodynamics, plume behaviour and vehicle stability (Ecker *et al.* 2020).

The recovery of the first stage booster is a complex task which relies heavily on the successful implementation of a control system. This requires engineers to have intimate knowledge of vehicle aerodynamics, the effect of control surface deflections and the use of propulsion systems for vehicle deceleration and attitude control. This data is typically consolidated in an aerodynamic database, where derivatives are compiled according to Mach number, Angle of Attack, thrust setting and control surface inclination angle. Due to the complexity and cost of gathering this data from full scale flight tests, results from high-fidelity CFD predictions play a large role in the vehicle design. These datasets are complemented by ground test results which further serve as a basis for CFD accuracy assessment and validation. A close interaction of wind tunnel based and CFD investigations enables the reduction of prediction uncertainties and extrapolation of ground test data to flight scales.

While there is substantial experience in load predictions for hypersonic capsule or winged type re-entry concepts, retro-propulsion maneuvers are a particularly challenging problem for both CFD and ground based testing. This is primarily due to the complex fluid mechanical and thermo-chemical interaction between the exhaust plumes and the free stream. During retro-propulsion phases, large parts of the vehicle are immersed in its exhaust gases. Large recirculation regions and zones of flow reversal significantly affect the heat load distribution and aerodynamic characteristics. Furthermore, the divergence of the exhaust jets at low ambient pressures gives rise to strong plume-plume interactions in the vicinity of the launcher base.

In this context, the RETPRO project aims at the further development and validation of experimental and numerical simulation techniques which are needed for the aerodynamic and aerothermal design of re-usable launch systems with retro-propulsion (Kirchheck *et al.* 2019). RETPRO is carried out under a programme of and funded by the European Space Agency - through the Future Launchers Preparatory Programme. RETPRO is one of a few studies within Europe looking at better understanding the requirements for the design of VTVL RLVs (Laureti & Karl 2022, Klevanski *et al.* 2018).

This paper focuses on the presentation and discussion of steady-state numerical results which reconstruct a test case at Mach 7 with one active nozzle conducted in DLR's Hypersonic Wind Tunnel Cologne (H2K). This serves as a precursor study for the work presented in refs. (Bykerk *et al.* 2022, Kirchheck *et al.* 2022). The main goal of these investigations is to evaluate the potential for condensation in the cold gas plume and any related effects this may have on the experimental data. First an overview of the RETPRO vehicle is presented. This is followed by an introduction to the CFD methodologies employed to recreate the selected wind tunnel test. Finally the results are discussed and conclusions are drawn on the correlations seen between experimental and numerical datasets.

2 RETPRO Vehicle

The RETPRO vehicle is of a two stage design with a re-usable first stage. Four grid fins are used for attitude control and braking, while four retractable legs are intended for the vertical landing. The shape is presented in Figure 1.

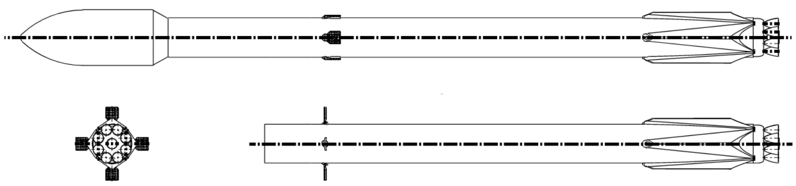


Figure 1. Overview of the RETPRO vehicle

An overview of the full sized vehicle geometric properties is summarised below in Table 1. The wind tunnel model has been scaled by a factor of 80.

Table 1. Vehicle geometric details

Property	Stage 1	Stage 2
Height (m)	47	23
Diameter (m)	3.66	3.66/5.20
Number of Engines	9	1

3 Methodology

3.1 Baseline Numerical Model

The TAU code is a second order finite-volume solver for the Euler and Navier-Stokes equations which includes a comprehensive range of RANS-based or scale resolving turbulence models. It uses unstructured computational grids to facilitate the analysis of complex geometries and is highly optimized for the application on massively parallel HPC systems (Langer *et al.* 2014). TAU has been successfully applied to a wide range of sub-to-hypersonic flow problems, both in scientific and industrial applications, including the analysis of re-usable launcher configurations.

The baseline set of numerical models which have been applied for the present investigations provide accurate and robust treatment for the flow conditions at Mach 7. The calculation of the inviscid fluxes in the finite volume framework is based on the application of the AUSMDV flux vector splitting scheme (Wada & Liou 1994) together with MUSCL gradient reconstruction (Van Leer 1979) to achieve second order spatial accuracy. Viscous fluxes are treated with a low-dissipation central discretization scheme.

Turbulence was modelled with a Spalart-Allmaras one-equation RANS model in low Reynolds formulation. This model provides a good compromise between numerical efficiency and accuracy and is particularly applicable to flows with strong compression shocks. The model completely resolves the structure of the turbulent boundary layer including the laminar sub-layer. Thus, an adequate setup of the numerical grid is required which is achieved by using prismatic sub-layers close to the wall with a first dimensionless wall spacing of y+ in the order of one and a wall normal stretching ratio of grid cells of less than 1.3. The baseline model uses an ideal gas assumption with a ratio of specific heats equal to 1.4.

3.2 Vapour-Liquid Equilibrium Model

To address the concern of potential condensation, an addition to the baseline configuration of the TAU code was created. Due to the principle of energy conservation, increasing the flow kinetic energy must be offset by a reduction in the thermal energy. Therefore, the acceleration of the room temperature air through an expansion in a nozzle will result in a decrease in temperature, which has already been shown to result in severe condensation in the plume during retro-propulsion pre-tests in the H2K facility (Gutsche *et al.* 2021). Depending on the expansion ratio, this temperature drop may be severe enough to reach the saturation temperature of the oxygen or nitrogen present in the air. The baseline thermodynamic model of a calorically perfect gas is not able to capture these effects and will give potentially non-physical temperatures in the flowfield, i.e. gas temperatures below the saturation limits for which no physical transport properties are available. This could have a significant effect on the temperature dependent variables such as Mach number and the ratio of specific heats.

A vapor-liquid equilibrium (VLE) model treats air as a tabulated fluid according to the Open Source Library CoolProp (Lemmon *et al.* 2021). For performance issues the equation of state (EOS) is tabulated within the CFD solver using an adaptive 2D quadtree approach covering the whole thermodynamic range of interest. The tabulation is done in a preprocessing step and the tabulated EOS is used in the CFD solver. The advanced EOS model of Lemmon et al. extends the applicability range of the fluid model to low temperatures, including liquid and gaseous states. The transition between gaseous and liquid states is computed using thermodynamic VLE routines. This provides a consistent transition of the fluid properties in the transition region between gaseous and liquid states using the saturation states. Within the fluid solver the vapour-liquid mixture is used as an integral model for the capturing of condensation effects at low temperatures. An additional quantity, the liquid mass fraction, is a measure for the amount of liquid inside the cell. This quantity is used later on to identify the zones in which condensation might occur. It is important to note that the assumption of chemical equilibirium is a conservative approach and may lead to an overestimation of condensation effects. This is due to the fact that no model is used for the onset of condensation. Thus, even though zero primer particles are available in the flow, condensation effects occur.

3.3 Internal Flow

The internal flow within the thrust nozzle was modelled separately to the external flow. The outflow conditions at the nozzle exits were interpolated to the external flow grid to simulate the active jet. To simplify the workflow and reduce computational expense, this was completed using a 2D axisymmetric setup of a single nozzle. The grid used for these investigations are provided below in Figure 2. Reservoir boundary conditions were taken from the wind tunnel test case, with a pressure and temperature of 31.66 bar and 294 K respectively.

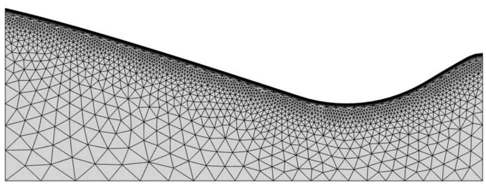


Figure 2. Single nozzle grid used for internal flow path

3.4 2D Axisymmetric External Flow

The first step in these investigations was to perform a precursor study on possible condensation effects on a 2D grid. Using a 2D axisymmetric approach allowed high grid densities relative to what would be possible on a 3D mesh for the same point count. As a result, the degree of condensation could be determined quickly for a grid with high resolution of the plume. An example of a close-up of the vehicle in a 2D grid is provided below in Figure 3. Note that some simplifications have been made to the geometry because the outer nozzles cannot be represented axisymmetrically. For this reason the peripheral nozzles were removed and all other elements have been treated as a body of revolution, i.e. central nozzle, landing legs, gridfins and the gridfin holders. Despite this, enough of the geometry has been retained to allow representative computations of the 3D case. Freestream boundary conditions were taken from the wind tunnel test case, with a stagnation pressure and temperature of 16.03 bar and 583 K respectively.

4 Results

Figure 4 presents a comparison between the baseline TAU solution and the VLE model for a Mach 7 test case. The liquid mass fraction presented is the fraction of gas which has condensed to a liquid. Results show a maximum value of 26.5% found in outer plume where flow expansion is high, while condensation in the core is around 5 to 15%. Naturally no condensation is computed in the baseline code version as the ideal gas thermodynamic model does not include condensation effects. Given the

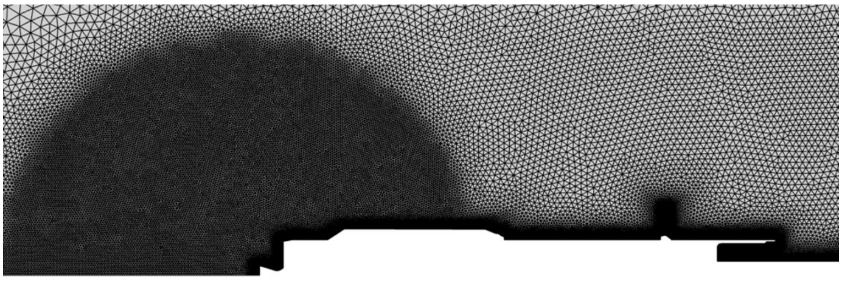


Figure 3. Example of a 2D axisymmetric grid of the RETPRO model

oxygen component of air is approximately 21%, the values above this in the plume extremity suggest that some nitrogen has also liquified. It is however important to mention again that this VLE method assumes an equilibirium state, which is a conservative estimate of condensation present in the plume.

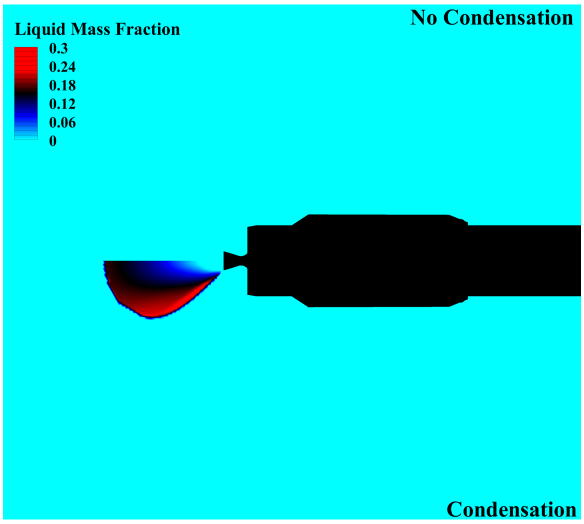


Figure 4. Comparison of the liquid mass fractions for condensation test case with baseline TAU perfect gas model

Figure 5 presents a comparison of the pressure field between the two solutions. The numbers have been included to identify areas of interest and they correlate to the list below.

1. Slight reduction in the shock stand-off distance seen with the consideration of condensation
2. Pressure in core plume varies slightly in regions where condensation is present
3. The size and shape of the plume varies, with the no condensation case showing a smooth, rounded plume extremity, where the condensation model appears to have a flattened shape. The location of maximum plume width is shifted aft in the condensation case
4. Due to the flatness of the plume, a slightly different pressure structure is observed

Heat is released when the phase change from gas to liquid occurs (heat of condensation), increasing the plume temperature compared to the case without condensation, thus unphysical temperatures in the plume are not encountered and all fluid temperatures are above the triple point of air. Slightly lower total temperature in the strong shock is observed and is attributed to the more accurate modelling of air, where the ratio of specific heats is not constant and equal to 1.4. These trends are visibile in Figure 6.

A lower Mach number is observed in the plume due to a higher temperature when condensation is present. As a result, a slight difference in the shape and location of the recirculation zone behind the plume exists and is attributed to a slight difference in plume shape seen in Figure 7.

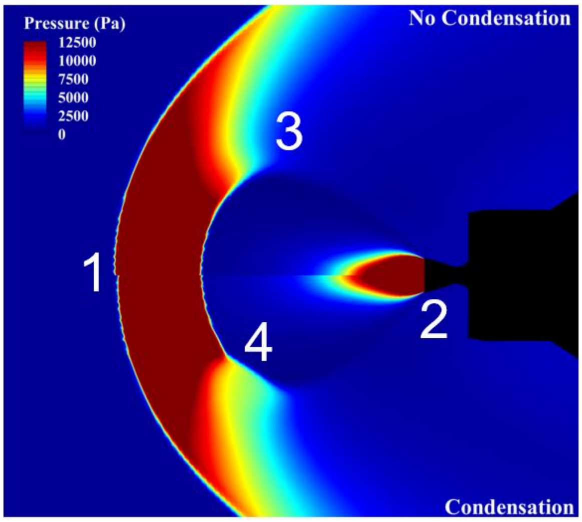
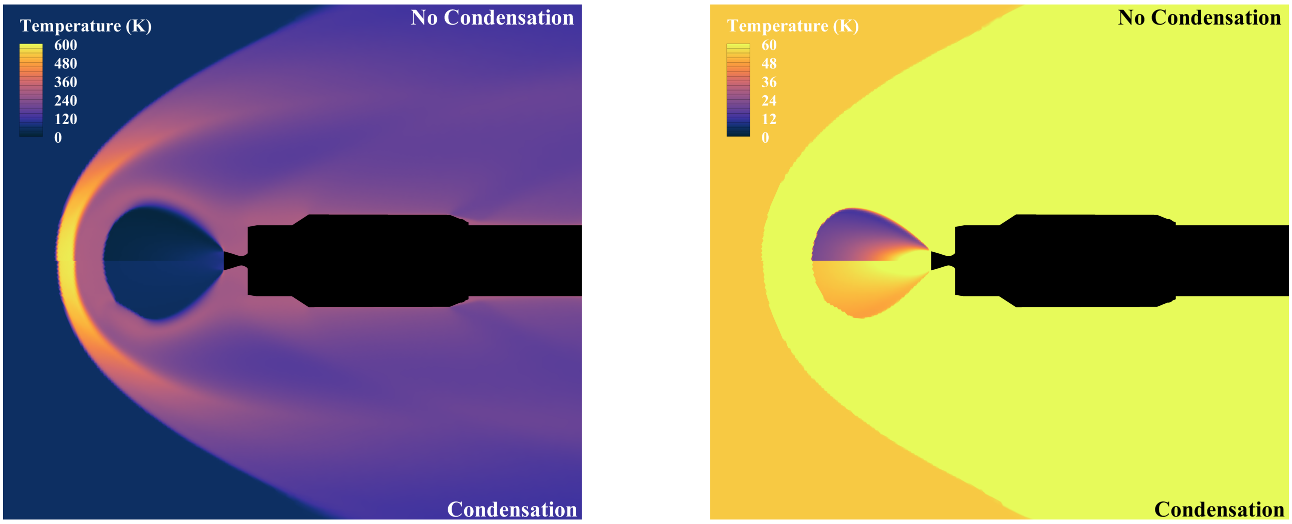


Figure 5. Comparison of the plume structure for condensation test case with baseline TAU perfect gas model

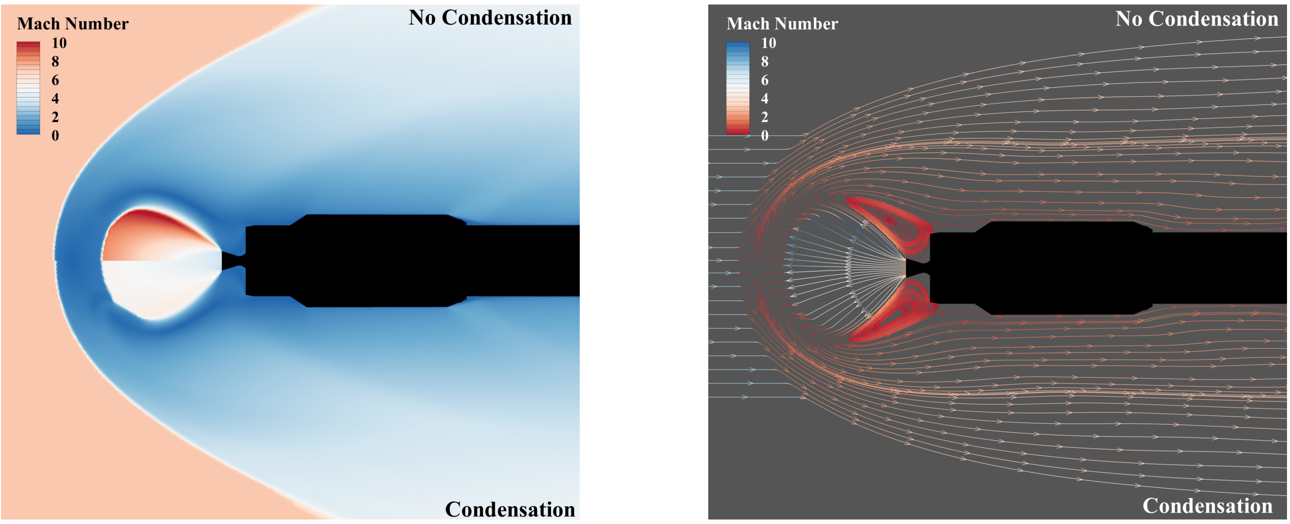


(a) **(b)**

Figure 6. Comparison of the temperature for condensation test case with baseline TAU perfect gas model showing (a) global field and (b) plume only

Figure 8 presents a comparison between the two pressure coefficient distributions along the surface of the model. Here an increased pressure in the base region and landing legs of the rocket without condensation is observed and is attributed to the change in the pressure flowfield from the plume shape. Overall an X force increase of approximately 5% was recorded for the case without condensation.

Finally a comparison between CFD and Schlieren photographs taken from the wind tunnel was made. Figure 9 presents two images, with the bottom sections showing the wind tunnel images and the top halves presenting the CFD results. Both cases show a slight overestimation of the shock stand-off distance when comparing with the experimental results. While the barrel shock from the nozzles are slightly different, there is no significant difference between the two cases which suggests that while condensation may be present, it does not play a significant role in obtaining an accurate comparison between CFD and WTTs.



(a) **(b)**

Figure 7. Comparison of the Mach number for condensation test case with baseline TAU perfect gas model showing (a) contour field and (b) coloured streamlines

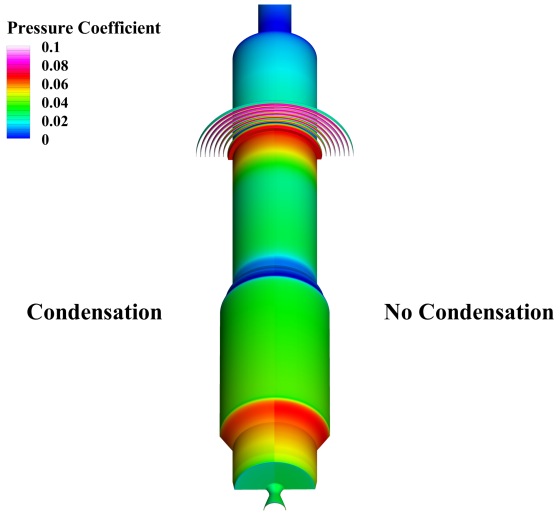
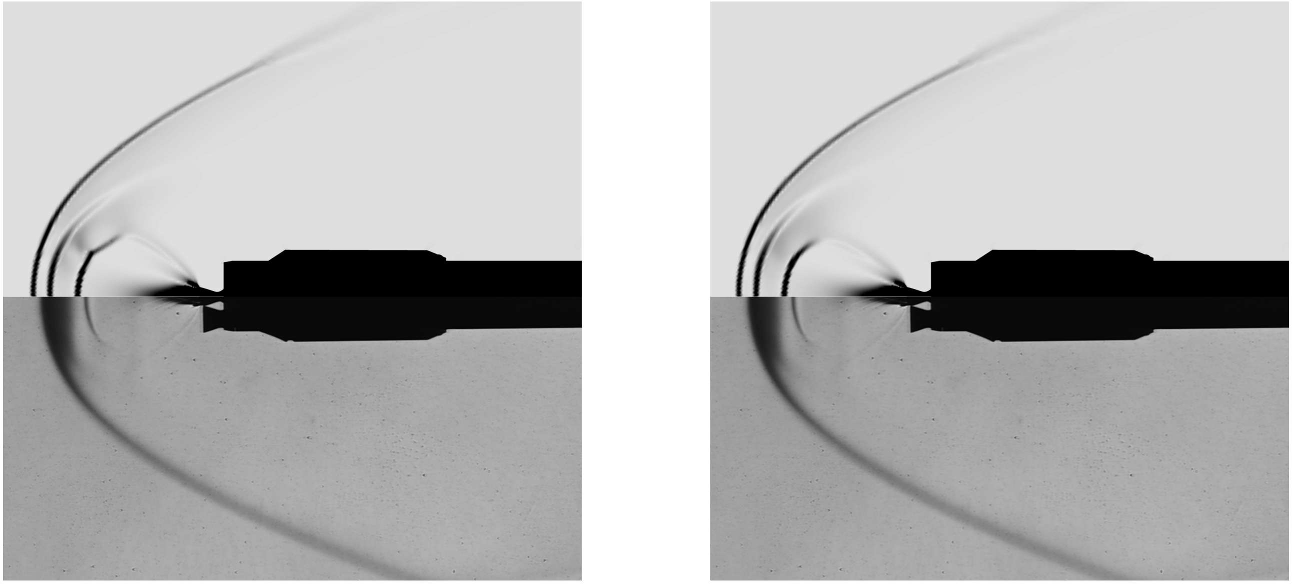


Figure 8. Comparison of the surface pressure coefficient for condensation test case with baseline TAU perfect gas model

5 Conclusions

This paper has numerically investigated the potential presence of condensation in the nozzle plumes for a cold gas test case at Mach 7. This was completed using a vapour liquid equilibrium model. Condensation was observed in the outer region of the plume and resulted in some variation to the plume size and shape. Subsequently the flow path in and around the plume was altered, causing minor changes in the pressure field. Because of this, surface pressure coefficients in the condensation case were observed to be approximately 5% lower than when using the baseline TAU code. Finally, the shock stand off distance was reduced, but not significantly. The described influences, which become visible in the CFD comparison, must be considered against the background of the conservative modeling of the condensation. Due to the equilibrium modeling, the deviations are also to be considered conservative. The comparison with tunnel data was therefore more-or-less the same as with the ideal gas model and further use of the condensation model was not deemed necessary.



(a) **(b)**

Figure 9. Comparison of the Schlieren images from wind tunnel with (a) condensation test case and (b) baseline TAU perfect gas model

References

Bykerk, T. and Kirchheck, D. and Karl, S. 2022, Reconstruction of Wind Tunnel Tests using CFD for a Reusable First Stage during Rocket Retro-Propulsion, *International Conference on Flight Vehicles, 9th European Conference for Aeronautics and Space Sciences (EUCASS)*.

Ecker, T. and Karl, S. and Dumont, E. and Stappert, S. and Krause, D. 2020, Numerical Study on the Thermal Loads During a Supersonic Rocket Retropropulsion Maneuver, *Journal of Spacecraft and Rockets*,**57** (1), 131–146.

Kirchheck, D. and Marwege, A. and Klevanski, J. and Riehmer, J. and Gülhan, A. and Karl, S. and Gloth, O. 2019, Validation of Wind Tunnel Test and CFD Techniques for Retro-propulsion (RETPRO): Overview on a Project within the Future Launchers Preparatory Programme (FLPP), *International Conference on Flight Vehicles, Aerothermodynamics and Re-entry Missions and Engineering (FAR)*.

Gutsche, K. and Marwege, A. and Gülhan, A., 2021, Similarity and Key Parameters of Retropropulsion Assisted Deceleration in Hypersonic Wind Tunnels , *Journal of Spacecraft and Rockets*,**58** (4), 984–996.

Kirchheck, D. and Marwege, A. and Klevanski, J. and Gülhan, A. 2022, Hypersonic Retrograde Propulsion Experiments - A Basis for Validation of CFD within RETPRO, *International Conference on Flight Vehicles, 9th European Conference for Aeronautics and Space Sciences (EUCASS)*.

Klevanski, J. and Ecker, T. and Riehmer, J. and Reimann, B. and Dumont, E. and Chavagnac, C. 2018, Aerodynamic Studies in Preparation for CALLISTO - Reusable VTVL Launcher First Stage Demonstrator, *69th International Astronautical Congress*.

Langer, S. and Schwoeppe, A. and Kroll, N. 2014, The DLR Flow Solver TAU - Status and recent Algorithmic Developments, *52nd Aerospace Sciences Meeting*.

Laureti, M. and Karl, S. (2022) Aerothermal databases and load predictions for Retro Propulsion-Assisted Launch Vehicles (RETALT), *CEAS Space Journal*.

Lemmon, E. and Jacobsen, R. and Penoncello, S. and Friend, D., 2000, Thermodynamic Properties of Air and Mixtures of Nitrogen, Argon, and Oxygen from 60 to 2000 K at Pressures to 2000 MPa, *J. Phys. Chem. Ref. Data*,**29** (3), 331–385.

Van Leer, B., 1979, Towards the ultimate conservative difference scheme. V. A second-order sequel to Godunov's method, *J. Comp. Phys.*,**32** (1), 101–136.

Wada, Y. and Liou, M.-S., 1994, A Flux Splitting Scheme with High-Resolution and Robustness for Discontinuities, *32nd Aerospace Sciences Meeting and Exhibit*