# The Impact of CFD Techniques on the Fidelity of Fixed Wing Rear Aspect Signature Prediction

**Dr Anthony Smith, Miss Yvonne Paulley**
S & C Thermofluids Ltd

UNITED KINDGOM

tony.smith@thermofluids.co.uk

**Keywords:** *Future Combat Aircraft, Modelling, Observability, CFD*

## *ABSTRACT*

*This paper is a summary and discussion of the conclusions drawn from work undertaken by AVT-232 and the following group AVT-281. The partner nations have investigated a large number of test cases to assess the fidelity of vehicle infrared signature prediction, drawn from multiple platform types, and the cases relevant to future combat aircraft are presented here.*

*Test cases were identified to challenge both the flowfield prediction using CFD and then the calculation of the infrared radiance. Partner nations used their own prediction tools and the results were compared against the experimental measurements, where available. Axisymmetric and rectangular gas turbine exhausts were modelled at sea level static and at Mach 0.3.*

*In general, the gas radiance models showed good agreement to the test data. The greatest variation was seen in the CFD predictions of the exhaust flowfield. These variations were due to a significant number of variables including mesh refinement and topology, turbulence models, effect of the surrounding flight stream, fidelity of the boundary conditions and nozzle geometry.*

*The MULDICON UAV was also modelled at cruise. The vehicle geometry and complex nozzle were included in the flowfield calculation. The region where the plume impinges on the aft deck was identified as a key infrared emission source.*

*The paper continues with a discussion of design techniques to minimise the sources of infrared emission in key aspect imaging. This includes novel means of rapidly mixing the exhaust to reduce the temperature and a discussion on alternative fuels.*

## 1.0 INTRODUCTION

The threat to air platforms from infrared (IR) guided missiles remains high, despite the continued development of advanced countermeasures. This is due to the tempo of developments of potent counter-countermeasure (CCM) techniques for air-to-air and surface-to-air missiles; particularly for recent man portable air defence systems (MANPADS).

The ability to accurately predict the signature of aircraft provides the basis for developing techniques to deny threats the capability to acquire aircraft at range. In addition, by shifting the balance between software modelling and extensive trialling during development, the economic burden on costly measurement trials can be reduced. Although trials will still be required as proof of principle when introducing survivability measures, improved understanding of the signature of both target and weapons demonstrated by accurate modelling (especially from first principles) ensures that contributing factors at the time of trials are understood and appropriately captured.

Previous NATO task groups in the 1990's investigated and produced software models to conduct IR signature predictions of air targets, including aircraft and missiles. This included a NATO standard prediction code NIRATAM and Computational Fluid Dynamics (CFD) models NPLUME and REP. With the advent of improved gas radiation line atlases and the widespread use of commercial CFD codes, a number of nations recognised a joint interest in revisiting the legacy data and prediction techniques to ensure that prediction techniques were fit-for-purpose, appropriately validated and included the benefits of recent computational improvements. The Joint Exercise on IR Signature Prediction, AVT-232, ran from 2014-2016 [1].

The participating nations within AVT-232 had a variety of codes and techniques available for the prediction of the IR signature of aircraft and missiles. Given the investment by each individual nation, and the variety of codes and process used, it was not considered appropriate to start a joint code development activity. Therefore code comparison through validation exercises was employed to ensure that the 'best of breed' techniques could be considered and some independence in establishing validation status through international Subject Matter Expert (SME) peer review. Many of these agencies represented already had mature and advanced signature prediction capabilities, but identified the potential benefit from improved validation and verification processes to identify existing gaps and uncertainties in their respective capability.

Of particular interest to the Specialists' Meeting on Multi-disciplinary design approaches and performance assessment of future combat aircraft, AVT-324, are the results of predicting gas turbine exhaust flows from representative nozzle shapes in both static and flight stream conditions. This test case highlighted the difficulties in accurately calculating the thermal distribution of the gaseous flowfield for air breathing engines, and the resultant impact on the predicted signature.

The work from AVT-232 continued in "Cross Domain Platform EO Signature Prediction", AVT-281, which operated from 2017-2019. This group extended the challenge of vehicle signature prediction to consider exhausts and vehicles from the land, sea and air domains. A theoretical UAV, MULDICON, was assessed as a full vehicle. No experimental data was available but modelling the complete airframe and exhaust provided significant challenges.

## 2.0 GAS TURBINE EXHAUST MODELLING

Using the Gnome Engine Test Laboratory[1] (GETL) it was possible to conduct a well-controlled experiment where a flight stream was provided at a variety of Mach numbers over two different AGARD nozzle configurations (axisymmetric and rectangular) [2]. These flows simulated ideally expanded non-reacting plumes (equivalent to air breathing engines). The nozzle geometry selected enabled a variety of pressure ratios and exhaust temperatures to be tested. Species concentrations measurements also contributed to a full data package of boundary conditions for computational fluid dynamics (CFD) and IR predictions.

Pressures and temperatures in the plume were measured using a traverse probe and infrared measurements were gathered at a wide range of engine and flight stream conditions to allow comparison with modelling.

Four cases were selected by the AVT-232 group for CFD and infrared modelling, these were:

- Case 1 – round nozzle, NPR 2.3, flight stream off
- Case 2 – round nozzle, NPR 2.3, flight stream Mach 0.3
- Case 3 – rectangular nozzle, NPR 3.7, flight stream off

---

[1] A UK engine test facility operated by S&C Thermofluids.

- Case 4 – rectangular nozzle, NPR 3.7, flight stream Mach 0.3

The round nozzle has a very small expansion ratio of 1.02 and so the NPR shown above represents operating fully expanded for cases 1 and 2.

## 2.1 Axisymmetric gas turbine exhaust

An axisymmetric nozzle of exit diameter 65mm was encased within an aft body geometry specified in [2]. This nozzle was mounted from a combustor rig at the Gnome Engine Test Laboratory (GETL)- see Figure 2-1. Note that the mixing length down the nozzle is longer than in a real world application to allow a flight stream nozzle to be installed.

**Figure 1: Axisymmetric AGARD nozzle and freestream exit.**

The resulting plume was traversed to capture total pressure and temperature across the nozzle exit and down the plume centreline.

The members of AVT-232 predicted the flowfield of the nozzle at sea level static conditions using the following CFD solvers:

- Fluent
- REP
- Hightail
- PLUMES

Some of these codes have the capability to alter the turbulence model used and the associated parameters.

### 2.1.1 Sea level static calculation

Despite the simplicity of this nozzle shape the group found challenges in accurately recreating the experimental data. The majority of the CFD solutions overpredicted the core length of the exhaust, as shown in Figure 2-2. This results in a larger hot core region and overpredicts the plume radiance. The turbulence model has a significant effect – REP uses k-W as standard but this is was developed for rocket motor plumes and is best suited to shear flows at Mach numbers around 2. Fluent uses k-ω SST as standard and this generated a long core and less rapid mixing than displayed in the experimental data.

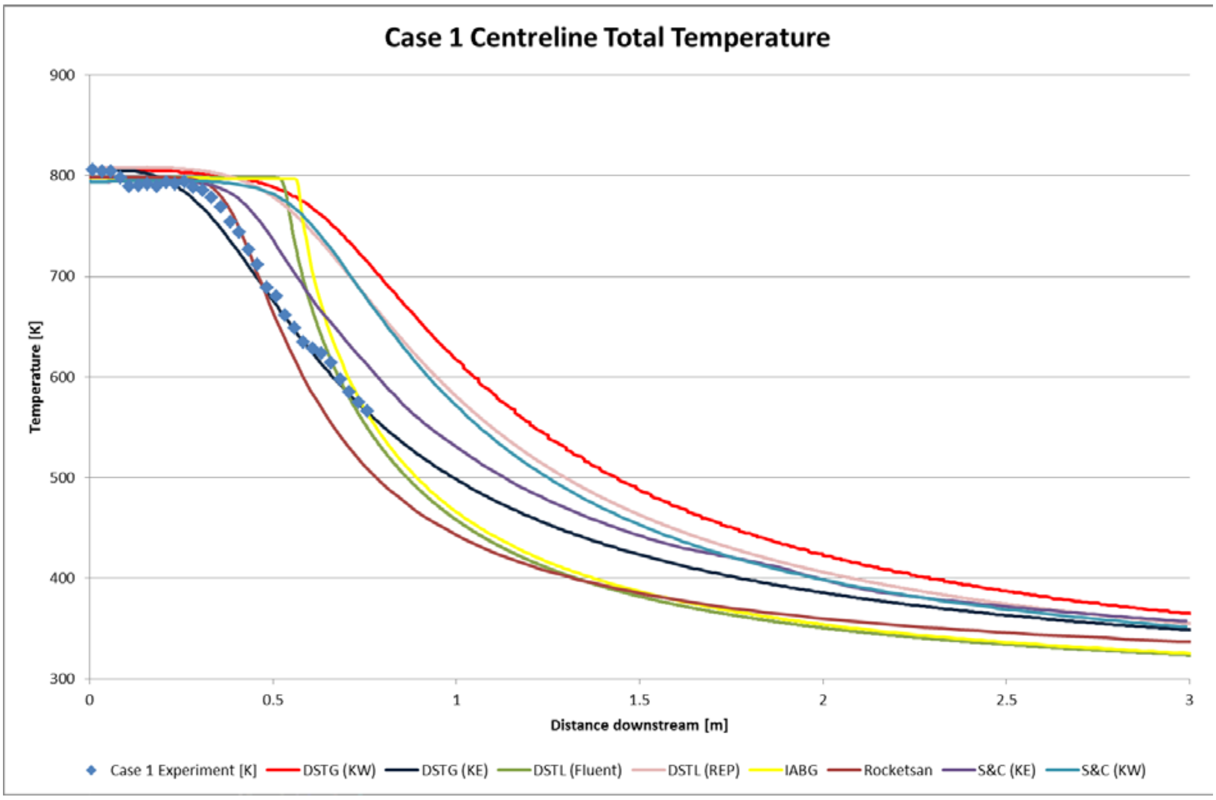

**Figure 2: Centreline total temperature plots for axisymmetric case with no free stream – predictions and experimental results**

The k-ε model is more suited to air breathing exhausts, but the extended mixing length upstream of the convergent nozzle requires the turbulence model parameters to be adjusted to take this into account. S&C investigated further modifications to the k-ε model, including the RODI [3] axisymmetric correction. The RODI correction decreases the mixing rate which matches the flowfield in this case, and the prediction was further improved by modifying the turbulence model boundary conditions.

### 2.1.2 Mach 0.3 free stream case

The presence of a free stream around an exhaust flow in principle should reduce the shear between the exhaust and its surroundings. This in turn should result in the core of the exhaust being longer for the free stream case than for the static one. However, the way in which the two streams come together can influence the change in the shear layer and modelling may not always take this into account.

The general trends observed from Case 1 were also seen in the case with a free stream Mach number of 0.3 (Figure 2-3) – although each nation successfully predicted a longer core and less rapid mixing when a free stream was present, relative to the static case.

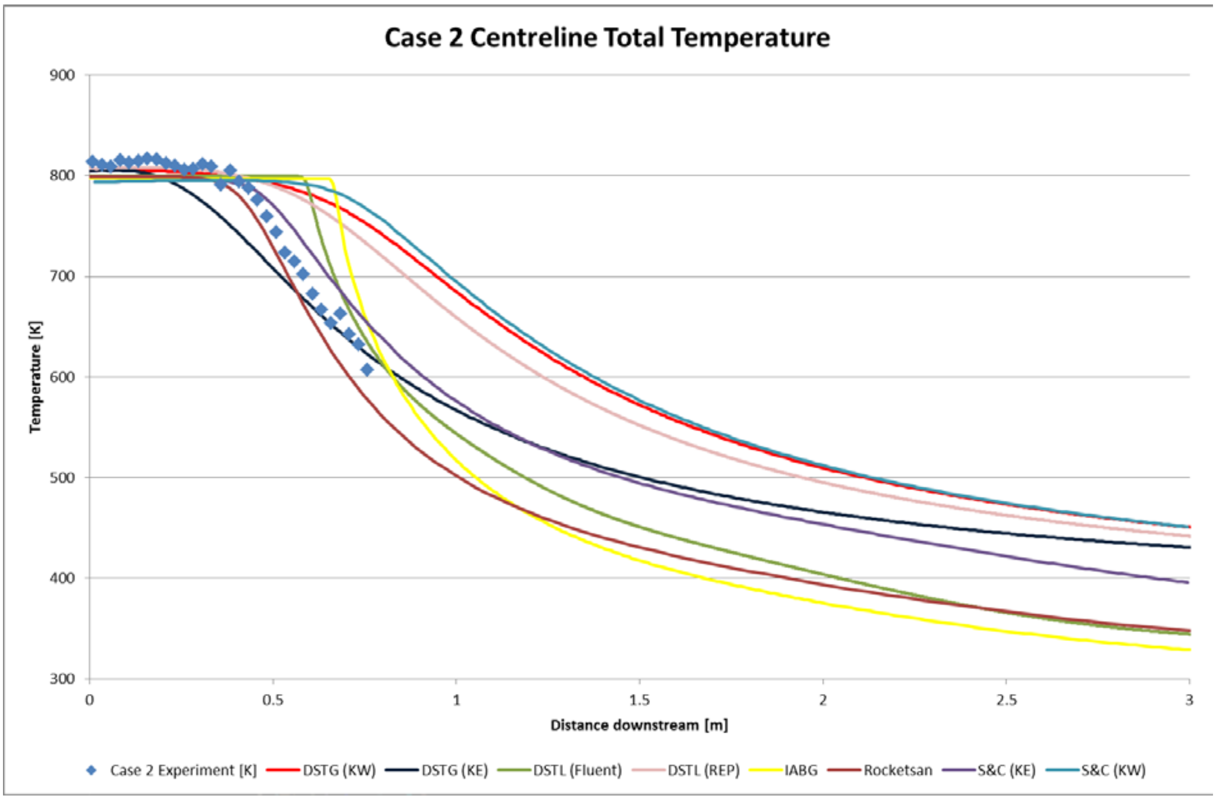

**Figure 3: Centreline total temperature plots for axisymmetric case with a free stream Mach 0.3 – predictions and experimental results**

## 2.2 Rectangular gas turbine exhaust

A rectangular nozzle geometry with an aspect ratio of 1.41 was selected from [2].

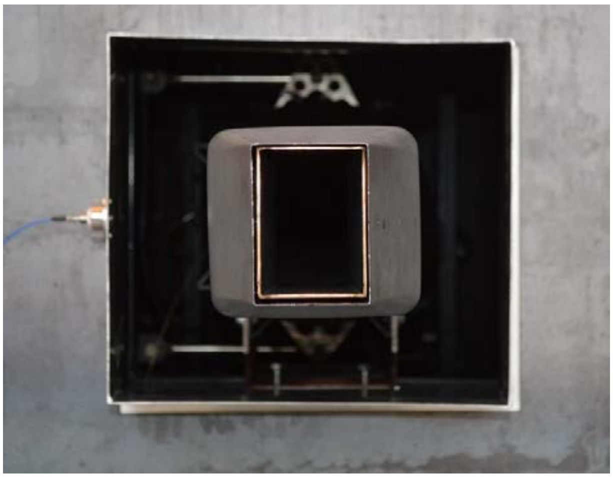

**Figure 4: Rectangular nozzle and aft body in the square free stream**

The rectangular nozzle presents two further challenges beyond those of the axisymmetric case.

1. The solution must now be 3D (quadrant symmetry or fully 3D) rather than 2D.
2. The nozzle is overexpanded (nozzle exit pressure below ambient) and therefore displays a shock structure which will influence both the plume length, and hotspots in the flow.

For the rectangular nozzle, results were gathered with the nozzle in two orientations so that the different plume thickness could be characterised in the infrared.

### 2.2.1 Sea level static calculation

Figure 2-5 shows the total temperature centreline data for the rectangular static case.

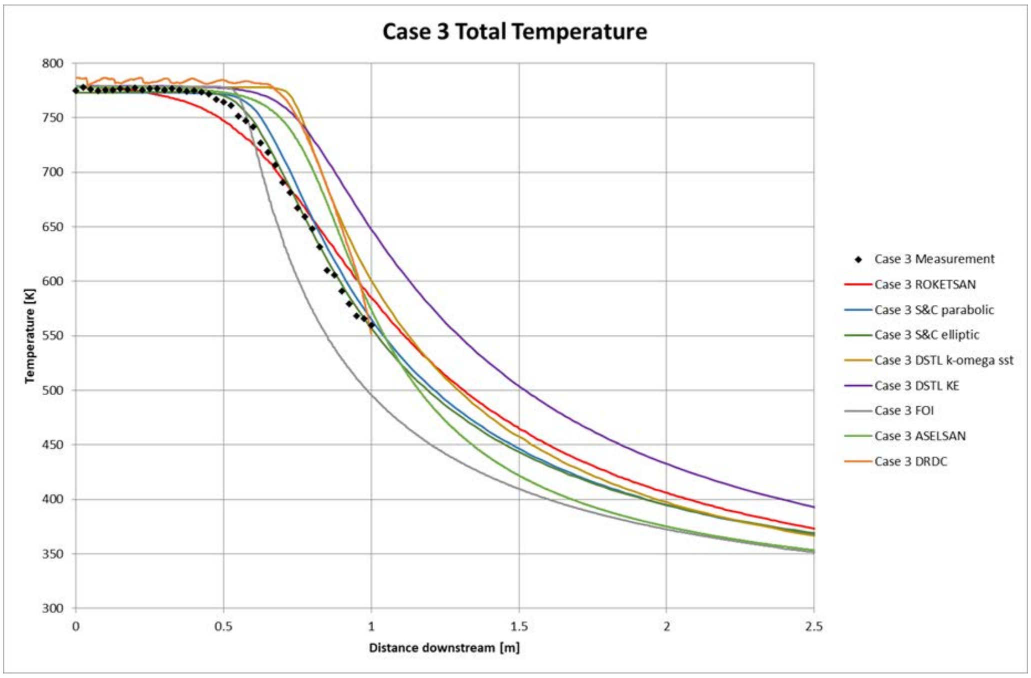

**Figure 5: Centreline plot of total temperature for rectangular case without free stream – predicted and experimental data**

Again the accuracy of capturing core length and mixing rate varied between modelling approaches. Here S&C's calculations using both a parabolic and elliptic solver were the closest match although the predictions do not capture the shock structure. S&C's PLUMES software has an automatically generated hexahedral mesh which allows rapid CFD calculations of exhaust plumes without the need to generate an explicit mesh. The plume is modelled from the exit plane, which in this case matches the boundary condition specifications.

### 2.2.2 Mach 0.3 calculation

When including the freestream, measurements and CFD predictions both show an increase in core length. There is also a reduction in downstream mixing when the freestream is present. Capturing this downstream mixing accurately was found to be challenging and was a greater contributor to predicting the overall plume signature than correctly predicting the location and intensity of the shocks.

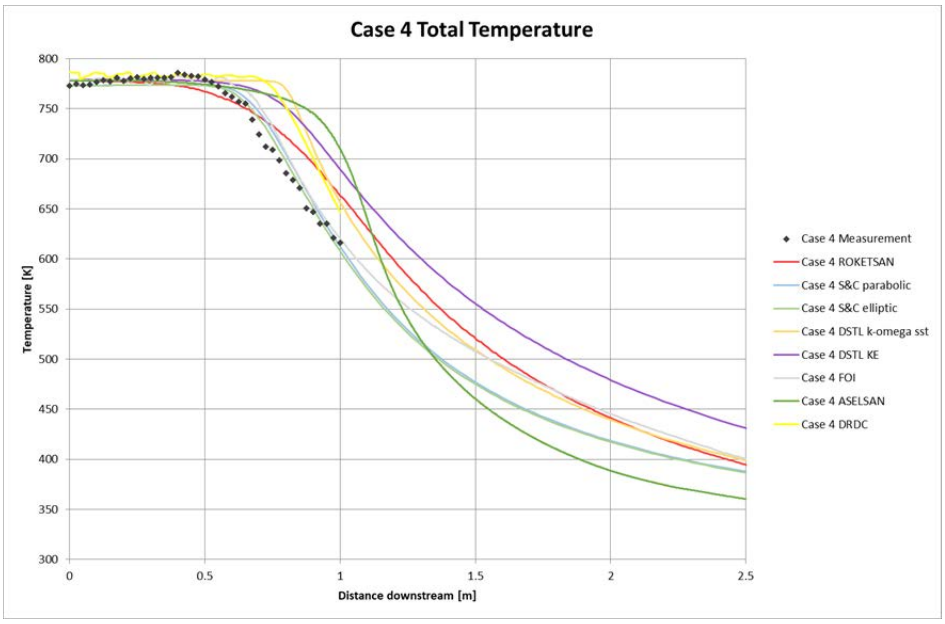

**Figure 6: Centreline plot of total temperature for rectangular case with Mach 0.3 free stream – predicted and experimental data**

## 2.3 Discussion

All of the above were calculated using Reynolds-Averaged Navier Stokes (RANS) codes with various two equation turbulence models, differing finite volume mesh element types and refinement, and first or second order discretisation. The use of large eddy simulation for these cases was considered but it was found that there was insufficient time within the work to allow for such a simulation. This in itself gives an indication as to the current maturity and applicability of the CFD technologies available to provide data for signature calculations.

Generally speaking, it appears that in order to achieve good resolution of shock structures within an exhaust plume there is a need to utilise a turbulence model which is not prone to producing numerical diffusion and to combine this with a higher order discretisation scheme suited to the very high gradients found in the shock system. However, such an approach does not produce very good results in terms of predicting the core length of the exhaust. The mixing is suppressed too much by the turbulence model and a long core length is predicted.

The k-W model which predominates in the REP code is one in which the constants were tuned to produce good results for rocket exhaust plumes in which the Mach numbers are around 2.0. Trying to apply this to a transonic aircraft exhaust where Mach numbers may be closer to unity results in a predicted core which is longer than in practice.

Likewise the Fluent k-ω SST model which may have compressibility correction applied when using the ideal gas equation, will produce good (multiple) shock prediction but an over predicted core length for an air breathing plume.

Using a simple k-ε model can produce a better prediction of the core length. This is particularly true if the upstream conditions are known either through additional modelling or by measurements. However, the increase in mixing which allows the core length to be better predicted does result in more smearing of the

shocks.
Much of what has been observed here agrees with the original findings of SEINER et al.[4].

Many signature prediction based exhaust plume prediction codes (such as REP and NPLUME) start the calculation at the exit plane of the nozzle. They also assume that the flight stream is brought to this plane with purely axial velocity and no boundary layer influence. In practice, this assumption, whilst made because no other information may be available, is likely to be inaccurate. When predictions are carried out with two uniform and coaxial streams the mixing rate is reduced in proportion to relative velocity of the two streams. However, in reality it is likely that two stream interact much more energetically than this, with non-axial velocity in the flight stream and substantial shear layers where the two streams meet. The extension in core length is therefore often observed to be significantly less than that predicted for the idealised case. However, predicting both plume and base region flow development is a relatively expensive task in terms of signature evaluation.

AVT-232 also tried to investigate the significance of discrepancies in the CFD results on the infrared signature. In general, the infrared codes (line-by-line and band models) used by each nation matched in signature prediction across the medium wave (MW) and long wave (LW) bands when they were applied to a gas field of known composition – as shown in Figure 7. Therefore the group concluded that it is the CFD flowfield calculation which contains the most unknowns.

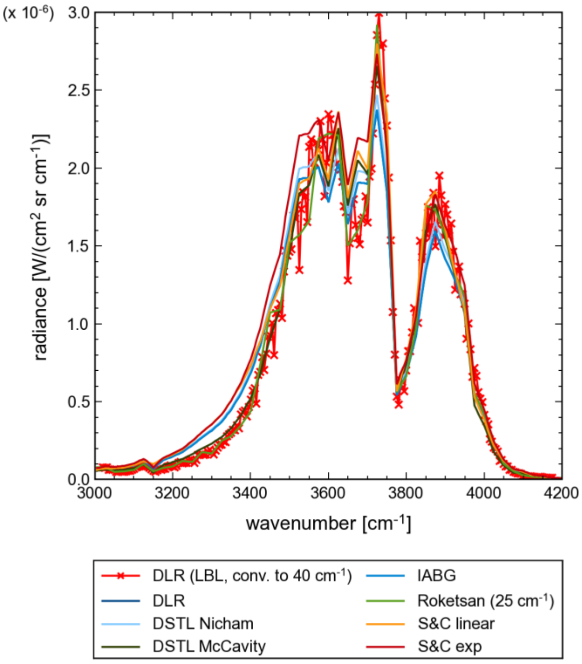

**Figure 7: Radiance plots for a theoretical gas slab containing 1% $H_2O$**

Much of the radiated energy which determines the contrast of the plume with the background is present in the core flow region. As the plume mixes downstream both the temperature, and the species primarily involved in the infrared emission ($CO_2$, $H_2O$ and CO, and carbon particles), mix out to lower levels. As the radiation is effectively a function of the temperature to the power of four, the downstream regions contribute very little to the signature compared to the core. Not surprisingly then, incorrectly predicting the core length has an impact on the ability to predict the contrast irradiance. Below is an example of the how the contrast radiant intensities compare for the axisymmetric plume case in static conditions. The experimental measurement in Figure 8 is compared with a REP case (k-W) in Figure 8. It can be seen that the intensity is over predicted by approximately 30%.

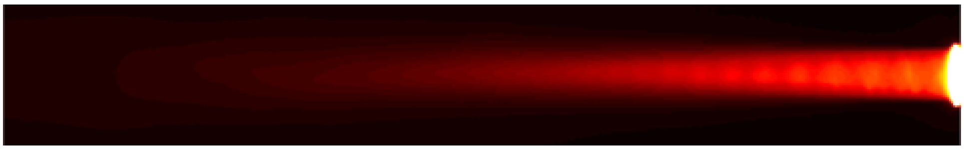

**Figure 8: Case 1 Measurement – Plume Apparent Contrast Radiant Intensity = 0.51Wsr$^{-1}$**

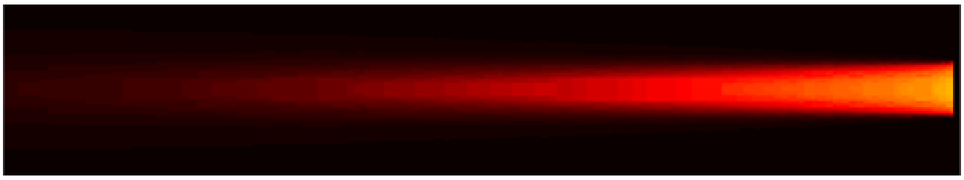

**Figure 9: Case 1 Dstl – REP - Nicham – Plume Apparent Contrast Radiant Intensity = 0.65Wsr$^{-1}$**

## 3.0 MULDICON UAV

The combination of an aircraft exhaust plume with the surrounding airframe structure can modify both the exhaust structure and the signature of the vehicle. The AVT-281 team investigated these gas-body interactions by modelling a theoretical UAV – MULDICON. This vehicle is a generic UCAV geometry created primarily by DLR to demonstrate a multidisciplinary design approach. It has been used in various NATO groups including AVT-251. The external geometry is shown in Figure 10.

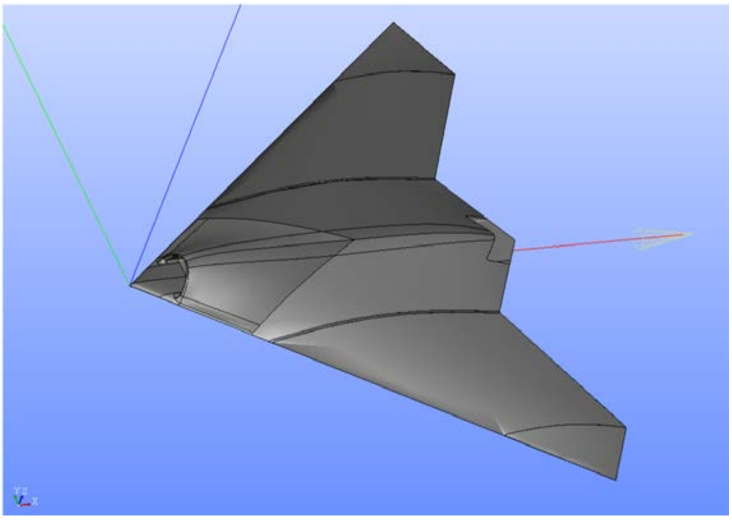

**Figure 10: The MULDICON geometry (courtesy of DLR)**

The air path geometry through the engine consists of a slightly S-shaped intake duct and a convoluted exhaust consisting of a round-to-rectangular transition on both the core and bypass flows, a slight S-bend, and a twin exhaust from the core flow. The resulting plume mixes on an aft body surface before exhausting into the free air. Figure 11 shows the exhaust geometry in detail.

**Figure 11: The MULDICON exhaust duct showing the bypass (pink) and core (red) (courtesy of DLR)**

Boundary conditions were provided, although these were for a relatively benign case.

## 3.1 Full body modelling

S & C Thermofluids focused on modelling the exhaust region of the vehicle rather than explicitly defining the boundary conditions relating to forward flight and the resulting temperatures on the upstream geometry. This paper will discuss the results from two different modelling approaches:

- Generating a high fidelity CFD mesh of the full exhaust duct and aft body and calculating the flowfield using ANSYS Fluent

- Performing a CFD calculation in the PLUMES software whereby the plume is allowed to influence the temperatures of any geometry where the plume impinges on the surface

These two approaches will now be discussed in more detail.

### 3.1.1 MULDICON in ANSYS Fluent

The airframe and plume were modelled explicitly, with a conformal hexahedral mesh generated in Pointwise (see Figure 12) and the solution run in ANSYS Fluent.

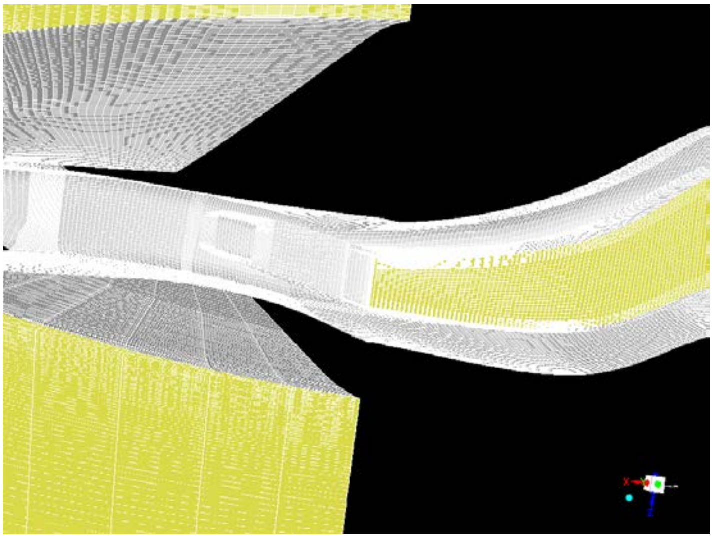

Figure 12: The MULDICON mesh in Pointwise

Achieving convergence of this model was difficult due to the multiple boundary conditions on the bypass, the two exhaust exits and the flow over the aft body.

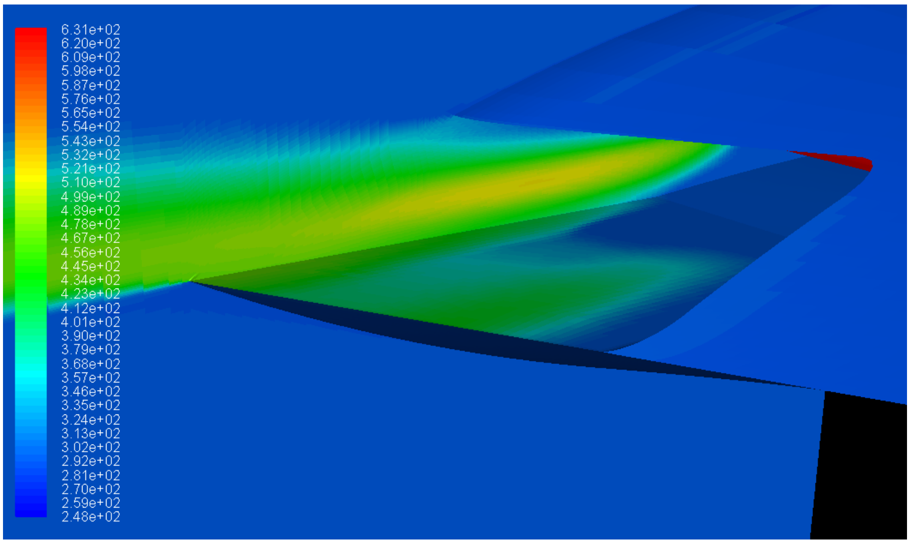

Figure 13: Fluent prediction of MULDICON exhaust flowfield

### 3.1.2 MULDICON in PLUMES

- Airframe “mapped” into fully orthogonal Cartesian plume mesh
- Boundary conditions set for 5 plume segments
    - Central core and 4 surrounding bypass
- 6th plume used to define the free stream flow

- Interaction of plume and airframe modelled

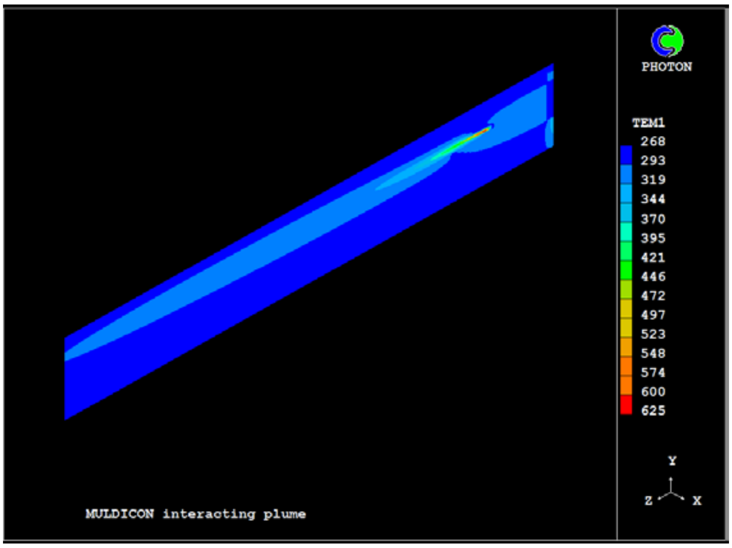

Figure 14: PLUMES prediction of MULDICON exhaust flowfield

The result can then be combined with a full body geometry with assigned temperatures as shown in Figure 15 below. A signature prediction can then be made with this assembled geometry – see Figure 16.

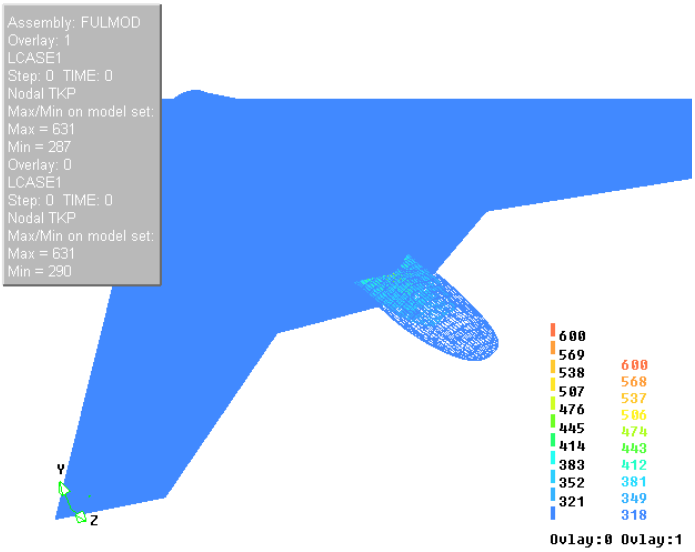

Figure 15: Combining PLUMES flowfield with MULDICON surface model

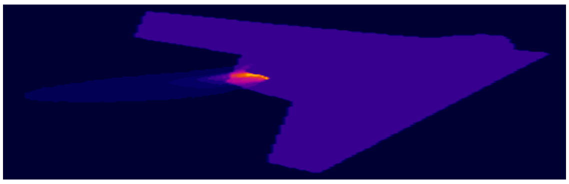

Figure 16: IR prediction of MULDICON surface and exhaust plume

### 3.2 Discussion

Producing a full CFD model of an aircraft to include a complex intake and exhaust system for the purpose of signature prediction is still a difficult and time consuming task. It may be difficult to justify this expenditure at certain stages within the design process. Indeed finding the resource for it may not be possible. Ideally a rapid method for providing at least a first order indication of the signature level is desirable.

Unfortunately the case chosen here has no truth data and so it is difficult to make a judgement on the acceptability of the lower cost and more rapid approach. This aspect needs further investigation in future aircraft signature prediction method studies.

## 4.0 DESIGN TECHNIQUES

As discussed in the introduction, the survivability of a platform is increased if its infrared signature is reduced. For combat aircraft, the main threats are from air-to-air missiles and MANPADS. The signature of the vehicle from below, nose-on and tail-on therefore all need to be considered.

### 4.1 Rapid exhaust mixing

A key method of reducing the exhaust signature is to encourage rapid mixing of the exhaust with the surrounding free air. Historically several techniques have been used to achieve this:

- a rectangular exhaust with a high aspect ratio (as seen on the F-117)

- A curved aft deck to use the Coanda effect, where the jet attaches to the surface and entrains ambient air as it turns before it detaches to leave the vehicle. The exhaust flow is subject to additional shear as it turns over the surface thus increasing the mixing rate. The use of such surfaces also provides obscuration of the hotter parts of the propulsion system.

Optimising the design of these complex shapes for a particular engine, to maximise the IR benefits while minimising the thrust loss, can be performed using CFD. However the flow conditions present challenges. The rectangular exhaust is discussed in the GETL case above. For the Coanda surfaces, it is ability to predict the attachment (or lack of it in extreme conditions) and the additional shear that results from it.

### 4.2 Alternative fuels

Climate change directives, such as achieving zero net greenhouse gas emissions by 2050, are likely to drive the availability of alternative fuels for future combat air systems more than the potential signature benefits.

But the use of hydrogen as a fuel, the availability of cryogenics on board an aircraft and the increased use of electrical propulsion systems are likely to lead to major changes and reductions in air vehicle signature.

S & C Thermofluids has carried out initial infrared measurements of the impact of hydrogen fuel on plume signature. Figure 17 below shows the hydrogen combustor used to produce a plume from a circular nozzle.

The removal of carbon dioxide (as well as carbon monoxide and carbon particles) from the exhaust flow, leaving only water as the main emitting species, drastically reduces the radiation from the plume. Figure 18 shows a MW image of the hydrogen plume. It can be seen that plume is hardly discernible from the background which is at ambient temperature. This compares to Figure 8 where the presence of carbon dioxide in the plume produces an IR source with significant contrast to the background.

The current tools available for signature prediction are able to cope with these changes as presently foreseen.

Figure 17: S & C Thermofluids hydrogen combustor rig

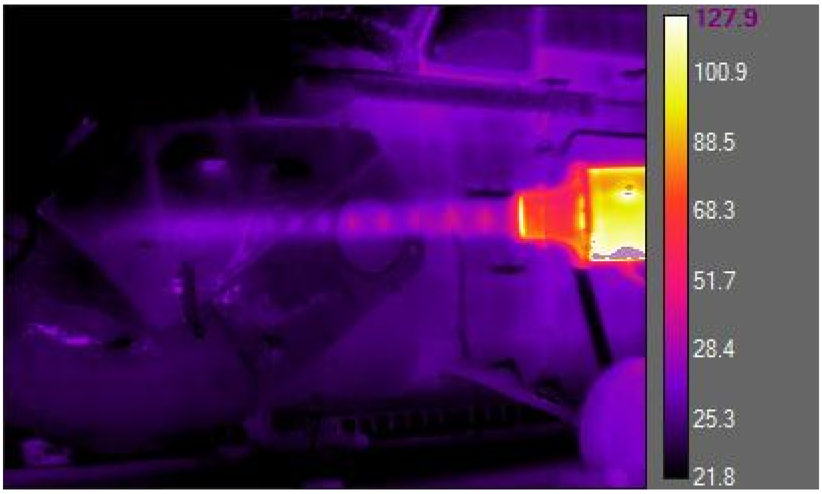

Figure 18: MW infrared image of a hydrogen plume (scale is apparent temperature °C)

# 5.0 CONCLUSIONS

The work undertaken by AVT-232 and the following group AVT-281 has been considered within the context of air vehicle rear aspect signature prediction. The infrared signature of air vehicles is significantly influenced by the exhaust plumes from the propulsion system as well as hot surfaces. All nations are able to draw on readily available CFD resources to predict the flowfield of such exhaust plumes in order to provide thermal and chemical composition data to feed into an infrared (IR) signature prediction.

Whilst all of the different (IR) prediction tools which use the CFD data seem to show similar capability, there was found to be significant differences in the plume flowfield prediction results obtained by the different participants in AVT-232 and AVT-281. Most of the CFD modelling produced results in which the core length predicted was longer than that produced in an air breathing exhaust flow experiment.

Many of the CFD codes use turbulence models which, in conjunction with 2nd order discretisation schemes, are able to replicate multiple shock systems which may be present in the exhaust plume. But such turbulence models tend to under predict the mixing rate of the flow and produce a longer exhaust plume core than is observed in practice. Turbulence models which are able to replicate the mixing rate needed to predict the core length correctly tend to smear out the shock system. Furthermore a knowledge of the nozzle and free stream flow development is needed (either through CFD or measurement) in order to predict the structure of the plume more correctly.

Predicting the core length of the exhaust plume correctly is important in obtaining accurate infrared signature evaluation.

Within AVT-281 a study has been carried out of a theoretical UAV – MULDICON. This further exercised the use of CFD to provide signature data for a full and complex air vehicle.

The methodologies for reducing rear aspect signature have tended to make use of rectangular nozzles and sometimes aftdecks which may be curved to increase obscuration. All of these features challenge the CFD modelling still further.

Given the questions raised about the fidelity of the CFD results in terms of exhaust plumes, and the potential expense of producing a high fidelity (geometrically) CFD model for a complex air vehicle, there needs to be a further evaluation of the application of CFD in this context. This evaluation needs to consider what is the optimum approach in terms of speed of solution and accuracy of prediction when using CFD to study complex air vehicle infrared signature.

In the future, it is possible that climate change solutions will result in technologies becoming available which may reduce rear aspect signature. For example using hydrogen as a fuel would significantly reduce the plume contribution to the air vehicle signature. The current prediction tools are able to handle such foreseen changes in propulsion technologies.

# REFERENCES

[1] Applied Vehicle Technology (AVT) Task Group 232 (2018) “Joint Exercise on IR Signature Prediction’ (Exercice interarmées sur la prediction de la signature infrarouge)”. STO TECHNICAL REPORT TR-AVT-232, Reference AC/323(AVT-232)TP/768, published March 2018.

[2] AGARD Fluid Dynamics Panel (Working Group #17) (1995) “Aerodynamics of 3-D Aircraft Afterbodies”, Advisory Group for Aerospace Research & Development, Advisory Report AR-318, September 1995.

[3] Rodi, W. (1972) “The prediction of free turbulent boundary layers by use of two-equation model of turbulence,” Ph. D. Dissertation in Mechanical Engineering, Imperial College, London.

[4] Seiner, J.M., Dash, S.M., and Wolf, D.E. (1985) “Analysis of Turbulent Underexpanded Jets, Part II: Shock Noise Features Using SCIPVIS”, AIAA Journal, vol. 23, no. 5, pp 669-677.