Determination of Flame Plume Characteristics utilising CFD and Experimental Approaches

NEIHAD HUSSEN AL-KHALIDY
CFD, Wind and Energy Technical Discipline,
SLR Consulting,
202 Submarine School, Sub Base Platypus, North Sydney,
AUSTRALIA

Abstract: - The potential for plumes to affect the safety of aircraft operations is often predicted using MITRE EPA Models. For many projects, key input parameters to MITER EPA are not available and conservative assumptions or models such as OHIO model are used to characterize the combustion and approximate the key input parameters to EPA plume rise model. These assumptions and conservative models lead to inaccurate results of simulations. The current study provides a novel approach to use a combination of Computational Fluid Dynamics (CFD) tool and EPA Models to reliably predict the risk of turbulence and upset being encountered by a range of aircraft types that operate through a rising plume.

The main objective of the current study is to develop a Computational Fluid Dynamics (CFD) combustion model and a procedure to determine an improved set of flare inputs for the Air Quality (AQ) and MITER EPA models. A CFD model has been developed to determine flame plume characteristics (Effective Height, Effective Diameter, Temperature and Velocity) from two flare stacks which are part of a trailer-mounted Mobile Purge Burner (MPB) system.

A subsequent experimental test of a similar trailer-mounted MPB system has validated the CFD results. Plume temperatures within the combustion zone of the flares were very much in line with the temperatures predicted by the CFD Simulation study. Plume temperatures above the MPB System appear to drop very quickly, such that the plume temperature fell from just under 500°C at 5 m above ground level to around 15-16°C (and close to the ambient temperature) at 22 m above ground. Again, this is consistent with the CFD Study results.

The CFD simulations in the current study accounted for the turbulent flow with chemical species mixing and reaction and utilised an advanced radiation model to solve participating radiation in the combusted zones.

This study assesses all the parameters that have impact on the accuracy of the numerical model including computational domain, mesh distribution, numerical scheme and flame plume characteristics including ambient conditions (wind speed and temperature) and combustion under various air to fuel ratio scenarios.

Key-Words: - CFD, Combustion, Flame Plume Characteristics, Complex Radiation, Modelling Inputs to MITER EPA

Received: December 26, 2022. Revised: April 14, 2023. Accepted: May 15, 2023. Published: June 6, 2023.

1 Introduction

Exhaust plumes that originate from power stations, cooling towers, industrial facilities such as smelters, and flaring associated with the de-pressurisation of gas systems, can result in a potential hazard to aircraft operations because of the velocity, turbulence, and/or location of the associated gas plume. Plumes can affect the handling characteristics of aircraft, create the potential for aircraft stalling or rolling and, in extreme circumstances, cause airframe damage.

Observations of plume trajectories from a 500 MW power station have been used to test the ability of a simple parameterization of entrainment to describe the rising plume in [1].

As the government body that regulates Civil Australian Aviation Safety (CASA) and the operation of Australian aircraft overseas, CASA is required by legislation to assess the potential for plumes to affect the safety of aircraft operations pursuant to CASA 139.370, [2]. The Advisory Circular sets out the procedure for conducting the assessment of a proposal that will create a plume rise. This methodology relies on a plume rise model originally developed by Commonwealth Scientific

Industrial Research Organisation (CSIRO) to model air pollution called TAPM (The Air Pollution Model). TAPM is a PC-based, nestable, prognostic meteorological and air pollution model (with photochemistry) driven by a Graphical User Interface and is a viable tool for year-long simulations, [3], [4].

In 2012, the United States Federal Aviation Administration (FAA) commissioned the MITRE Corporation, [5], to develop an exhaust plume analyses model that would predict the risk of turbulence and upset being encountered by a range of aircraft types that operate through a rising plume. The outcome of this work was the MITRE Exhaust Plume Analyzer (EPA) model. In September 2015, the US FAA formally adopted the MITRE EPA model as the assessment tool for assessing the impact of plume rise sources on aircraft safety.

The MITRE EPA model, [5], uses a combination of models to simulate the plume and aircraft behaviour; including a convective flow model describing the mean flow of the plume, a turbulence model computing the probability of experiencing a gust capable of causing severe turbulence or aircraft upset and aircraft response models judging the required vertical gust to achieve severe turbulence or aircraft upset. The key input parameters for flare plume modelling are:

- Effective Height of the Plume at the Equivalent Exhaust Emission Point H_{eff} (m)
- Effective Diameter of the Plume at the Equivalent Exhaust Emission Point Dia_{eff} (m)
- Velocity of the Plume at the Equivalent Exhaust Emission Point Vp (m/s)
- Temperature of the Plume at the Equivalent Exhaust Emission Point Tp (^oC)

For many projects, measurement data for the above key input parameters are not available and conservative assumptions or models (e.g., OHIO EPA model) are used to characterize the combustion source in terms of an "equivalent" stack and approximate the above parameters.

The current study presents Computational Fluid Dynamics (CFD) as a tool to determine an improved set of flare inputs for the EPA models in lieu of using conservative and unrealistic assumptions and models. The main elements of this study are:

- Assess the fuel combustion. The gas composition used in this study consists of Methane 88.8%, CO_2 1.9%, N_2 1.5%, Ethane 7.8% and Propane 0.2%.
- Determine the fuel rate in the system.
- Model the combustion process to estimate the chemical reacting species and associated energy due to the chemical reaction, mixture speed in three directions, pressure profile, temperature, and turbulence parameters.
- Predict the participating radiation during the combustion process.
- Predict flare characteristics for various atmospheric conditions (eg. near calm wind, light breeze, etc.).
- Assess the impact of the Air to Fuel ratio on the flame plume.
- Determine the flame plume characteristics (Effective Height, Diameter, Temperature and Velocity).

2 Problem Description

The use of Mobile Purge Burner (MPB) Trailer to burn off residual gas to depressurise redundant main and support commissioning operations is necessary to avoid release of natural gas to atmosphere (environmental regulations) and prevent a natural gas plume gathering in airspace.

The assessed MPB Trailer in the current study consists of two stacks (Refer Figure 1). The trailer system is designed for 1050 kPag with a step-down regulator set between 480 kPag to 1030 kPag.

A trailer inlet pressure of 1030 kPa is adopted for the flow rate calculation as this will result in a higher flow rate.

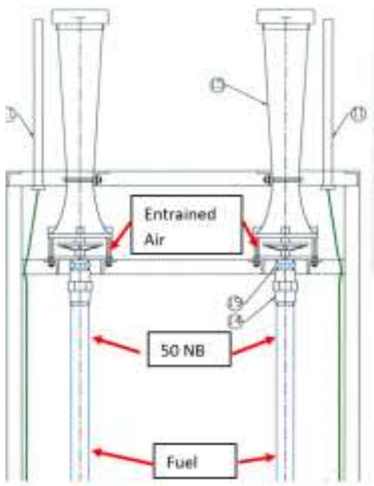


Fig. 1: Mobile Purge Unit

3 Problem Formulation

The CFD model solves the continuity and momentum, energy and chemical species equations. The equations for a steady state case can be written as follows, [6]:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}) + \rho g_i + F_i$$

$$\frac{\partial}{\partial x_i}(\rho h) + \frac{\partial}{\partial x_i}(u_i \rho h) = \frac{\partial}{\partial x_i}(k_{effective} \frac{\partial T}{\partial x_i}) + S$$

$$\nabla \cdot (\rho \vec{v} Y_i) = -\nabla \cdot \vec{J}_i + R_i + S_i$$

Where ρ is the density of air, u is the air velocity, p is the static pressure, ρg and F are the gravitational body and external body forces, τij is the stress tensor, h is the enthalpy, $k_{effective}$ is the effective thermal conductivity and S is the volumetric heat source. J_i is diffusion flux of specie i, R_i is the net rate of production of species i by chemical reaction and S_i is the rate of creation reaction by addition from the dispersed phase plus any user-defined sources.

The local mass fraction of each species Yi is predicted through the solution of a convection-diffusion equation for the i^{th} species, [6].

4 CFD Simulation Methodology

4.1 Geometry for CFD Modelling

A detailed 3D model of the MPB trailer was created based on the provided engineering drawings. The overall model is shown in Figure 2.

- The CFD model includes all the flare features as per the given engineering drawings.
    - The main pipe is 50 NB x 603 Schedule 45 Pipe.
    - The flare tip diameter is 165 mm.
    - The flare height is 3.1 m above local ground level.
- Air is introduced through an annular inlet. The air disc intake is modelled as per the provided layout drawings.
- The CFD model also accounts for the impact of the approaching winds on the flare characteristics.

4.2 Boundary Conditions and Properties

4.2.1 Fuel Composition and Operating Conditions

The gas composition used in the MPB consists of Methane 88.8%, CO_2 1.9%, N_2 1.5%, Ethane 7.8% and Propane 0.2%.

For the current study it is conservatively assumed that the fuel consists of 100% methane. Methane is more flammable than ethane.

The fuel rate in the system has been predicted in a previous study using HYSYS software. The study found the following:

- The total flare flow rate (from both stacks) will be limited by the pressure regulator located on the trailer.
- The regulator is a DN25 Fisher 627 with 1/8"orifice and downstream pressure setpoint of 210 kPag. The minimum gas temperature of 10ºC was adopted as this gives the worst case (i.e., largest flowrate through the Fisher 627 regulator).
- The trailer system is designed for 1050 kPag with a step-down regulator set between 480 kPag to 1030 kPag. A trailer inlet pressure of 1030 kPag was adopted for the flow rate calculation as this will result in a higher flow rate.
- The fuel mass flow rate in the system is 0.0159 kg/s. The flow is equally split through the two stacks, i.e., 0.00799 kg/s through each flare stack.
- The fuel temperature is 5ºC pre-combustion.

4.2.2 Air Intake

The air disc (Refer to Figure 2) can be adjusted. In the current study, two scenarios were analysed:

- Scenario AFI-1: Limited air flow intake to complete the chemical reaction.
- Scenario AFI-2: Increased air flow intake to accelerate the combustion process.

4.2.3 Wind and Ambient Temperature Conditions

The CFD results are presented for the following worst-case scenarios:

- Scenario 1: Near Calm Wind Condition (Wind Speed = 0.1 m/s)

- Scenario 2: Very light Breeze with an Average Wind Speed of 0.28 m/s (1 km/hr)

The impact of the atmospheric air temperature has also been assessed in the current study. The computational domain for CFD modelling is shown in Figure 3.

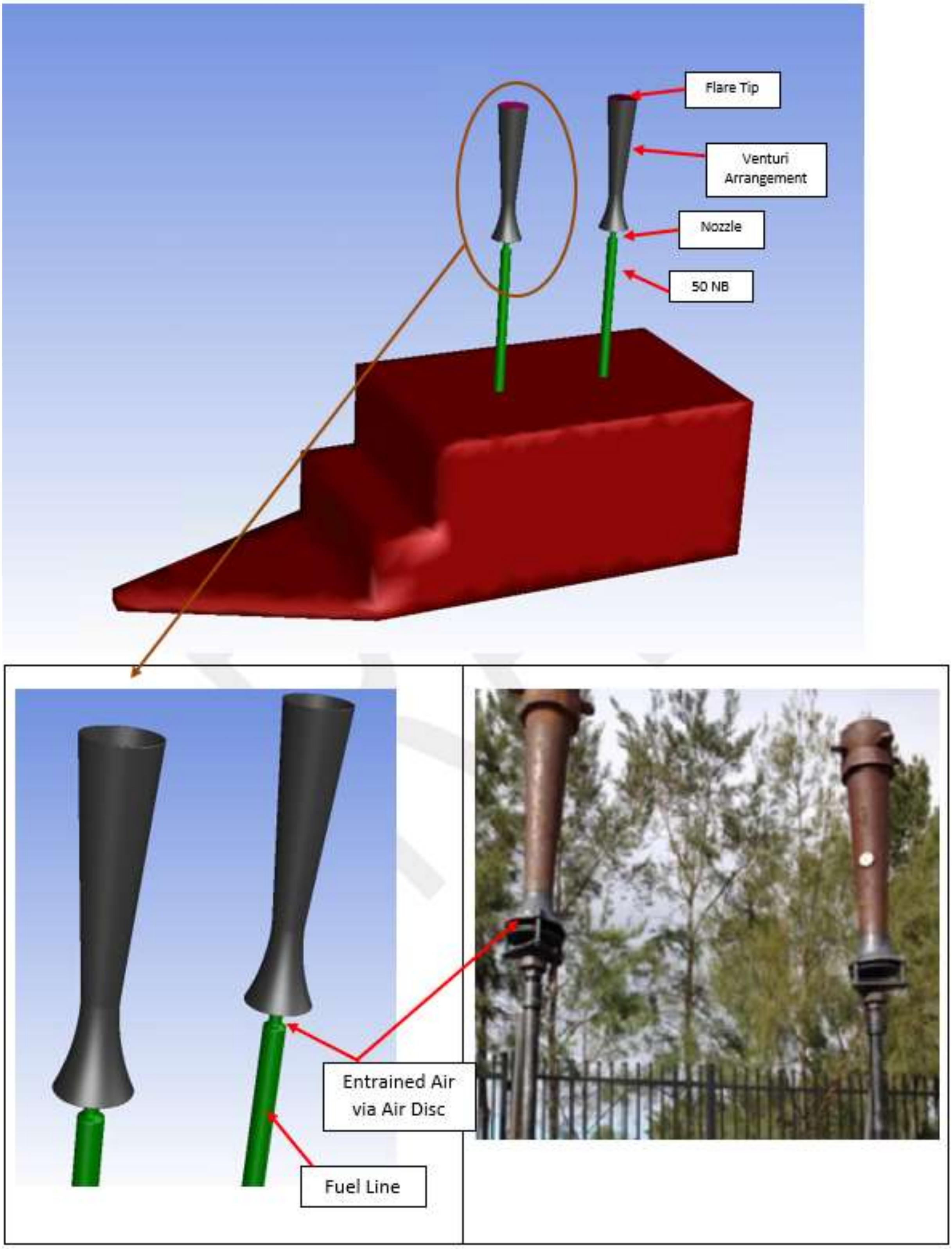


Fig. 2: 3D Geometry for CFD Modelling

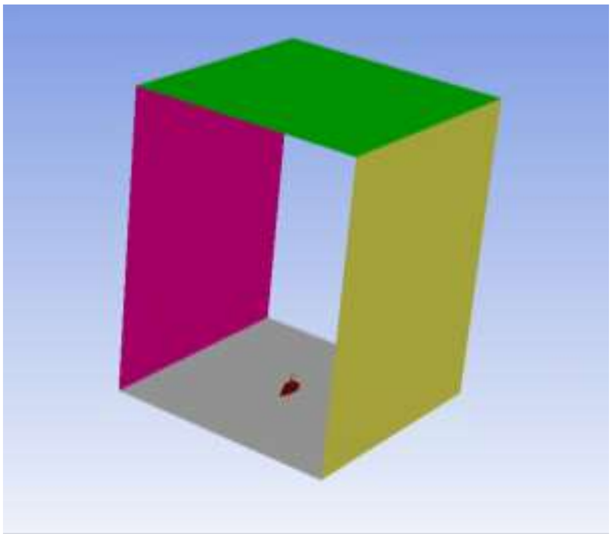


Fig. 3: Computational Domain for CFD Modelling

4.3 Combustion Modelling

Natural gas combustion is a complex thermodynamic process and involves a large number of interacting processes, including turbulent flow, gas phase chemical reactions, and heat transfer.

A generalized Eddy-Dissipation Model (EDM), [6], was used to analyse the methane-air combustion system. The EDM model is suitable for chemical reactions where fuels burn quickly, and the overall rate of reaction is controlled by turbulent mixing.

The combustion is modelled using a global chemical reaction mechanism, assuming complete conversion of the CH_4 to H_2O and CO_2 plus heat.

$$CH_4+2O_2 \rightarrow CO_2+2H_2O$$

This reaction is defined in terms of stoichiometric coefficients, formation enthalpies, and parameters that control the reaction rate. The rate at which the reaction proceeds is directly proportional to the concentrations of the two reactant species.

The EDM models have been implemented in major commercial CFD software such as ANSYS-Fluent, [6], and the model has been used in the CFD simulation of a variety of different combustion systems, [7], [8], [9], [10], [11], and, importantly, validated against experimental data.

The air-fuel mixing and transport of chemical species are modelled by solving conservation equations describing convection, diffusion, and reaction sources for each component species with reactions occurring in the fluid phase (volumetric reactions).

The effects of radiation are also examined in the current study using the Discrete Ordinates (DO) Radiation Model, [12]. The DO model provides the ability to solve problems ranging from surface-to-surface radiation to participating radiation in combustion problems.

4.4 Discretization

The software package used in the current CFD analysis is the commercially available code Fluent.

The CFD model solves continuity, momentum, energy and transport species equations to predict the combustion and fluid flow characteristics in the computational domain.

- The quality of the mesh is a critical aspect of the overall numerical simulation, and it has a significant impact on the accuracy of the results and solver run time. A mesh sensitivity assessment has been carried out for the current study. A procedure was developed to adopt a very fine mesh at the areas of interest, i.e., in the immediate area around the two flares.
- Initial runs were conducted using approximately 10 million mixed cells. The mesh was then optimized using polyhedral elements.
    - For the current analysis, polyhedral elements with a total 10,915,033 nodes have been used to cover the computational domain.
    - Polyhedral cells are especially beneficial for handling complex flows and used to provide even more accurate results than with a hexahedral mesh. For a hexahedral cell, there are three optimal flow directions which lead to the maximum accuracy while for a polyhedron with twelve faces there are six optimal directions which, together with the larger number of neighbours, lead to a more accurate solution with a lower cell count, [13], [14].
    - In general, very fine meshes are used for the` entire computational domain and the maximum Y+ in the computational domain is 52. An example of the final mesh scheme is shown in Figure 4.
- The Eddy-Dissipation Model (EDM) was used to model the combustion process and analyze the methane-air combustion system.
- A Realizable k-ε turbulence model was adopted in the current study.
- The Discrete Ordinates (DO) Radiation Model was used to predict the participating radiation during the combustion process.
- An iterative procedure was used to estimate

the chemical reacting species and associated energy due to the chemical reaction, mixture speed in three directions, pressure profile, temperature and turbulence parameters.

- For the pressure velocity coupling, a global solver based on the COUPLE algorithm was employed.

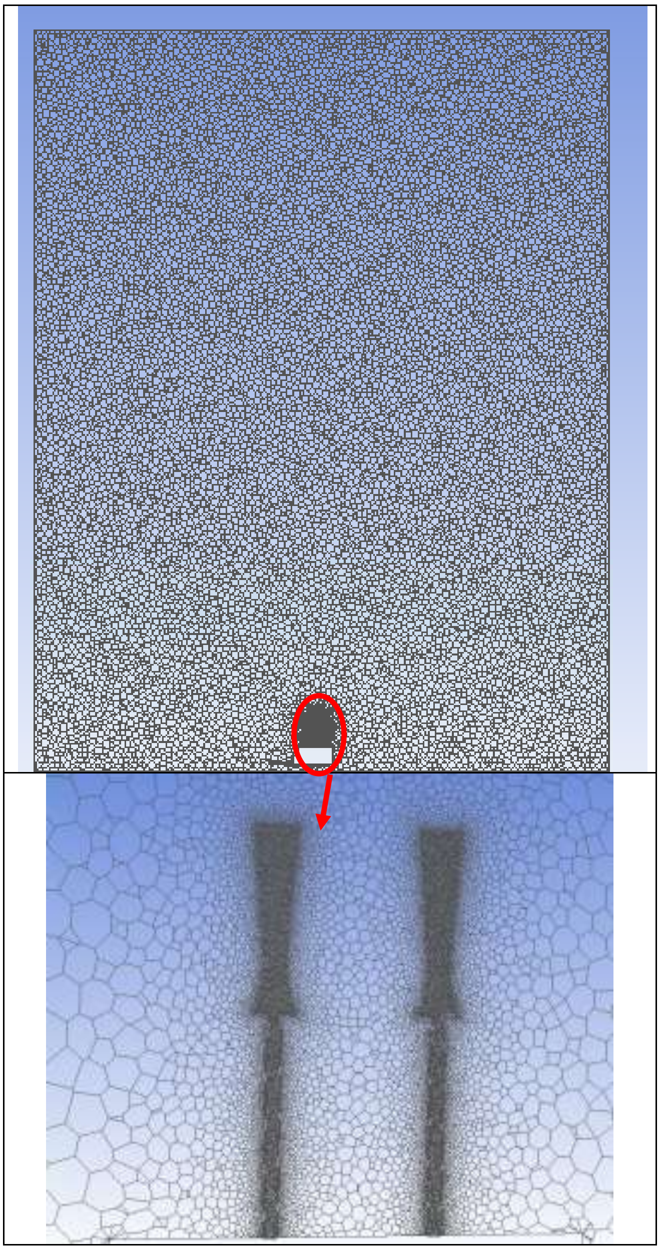


Fig. 4: Mesh Density for the Area of Interest

- A second order numerical scheme was used for discretization of pressure and momentum to obtain more accurate results.

All of the above represent state-of-the-art CFD modelling techniques for combustion simulations.

The normalised residuals of continuity x-, y-, and z-velocity, reacting species, energy, DO intensity, k and epsilon was reduced between five and seven orders of magnitude demonstrating a valid numerical solution. Figure 5 shows that the normalised residuals for all variables.

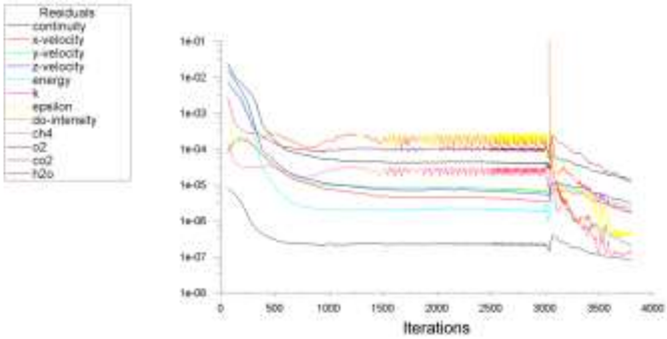


Fig. 5: Scaled Residual History

5 CFD Results and Discussion

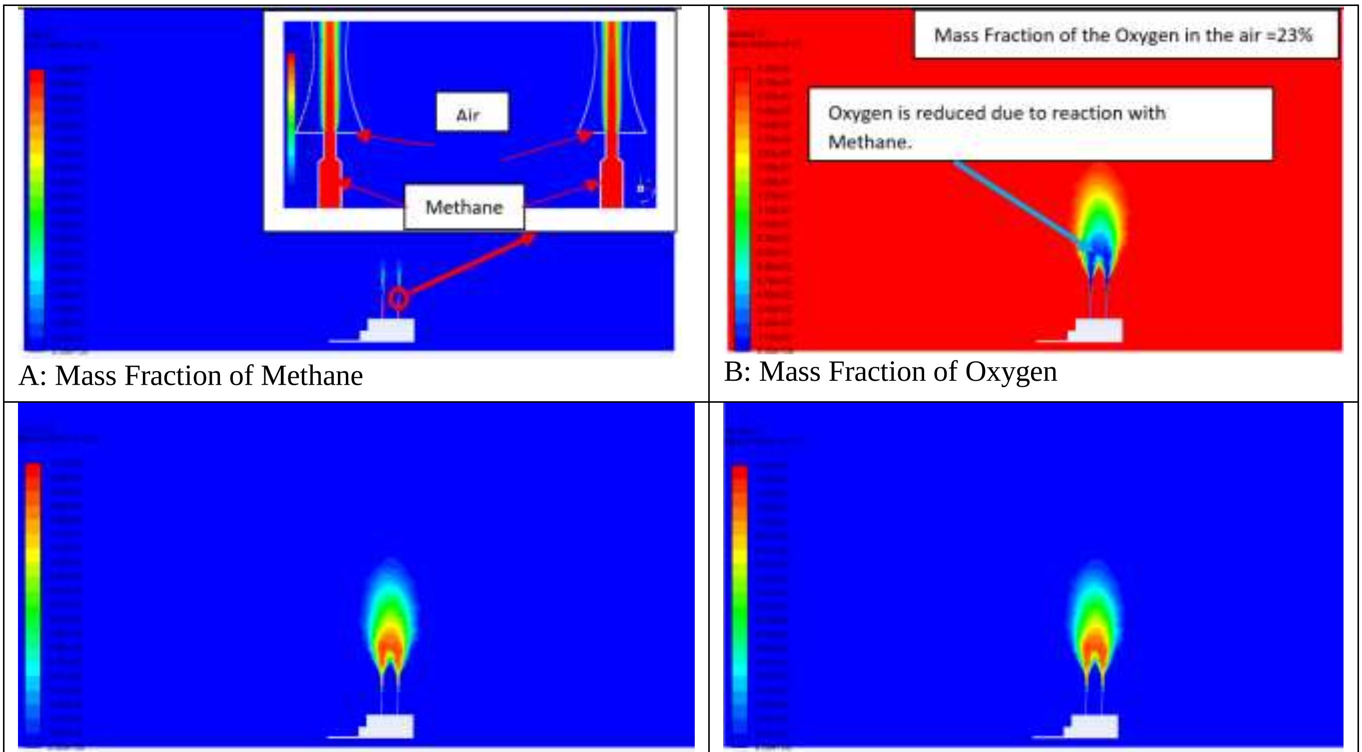
5.1 Near Calm Wind Condition (Approaching Wind = 0.1 m/s)

5.1.1 Scenario AFI-1 – Air to Fuel (Methane) Ratio = 1.5

The objective of this scenario is to predict the characteristic of plum during poor combustion process. Limited air flow intake (0.015 kg/s) (Scenario "AFI-1") with an ambient temperature of 15ºC was initially used to complete the combustion process.

The mass fraction of reactants is shown in Figure 6. The following conclusions can be reached from Figure 6:

- Methane and air are introduced at the base of the combustion chamber. Refer Figure 6A
- The mass fraction of the methane is reduced gradually due to the methane oxidization reaction process.
- Initial mass fraction of the air is 23% (Figure 6B) as per the given boundary condition.
- As the reaction initiates, the oxygen in the air is consumed inside the chamber and above the tip of the flare. Refer to Figure 6B



A: Mass Fraction of Methane

B: Mass Fraction of Oxygen

C: Mass Fraction of CO_2

D: Mass Fraction of H_2O

Fig. 6: Mass Reaction of Reacting Species

- Figure 6C shows the mass fraction variation of CO_2 products. The formation of CO_2 continues to increase as the methane consumption increases.
- Figure 6D shows the mass fraction variation of H_2O products. As the reaction proceeds, the H_2O formation increases and the final mass fraction of H_2O in the mixture is 12.8%.

The predicted mass fractions of H_2O and CO_2 are in a very good agreement with the data in the open literature (e.g., [15]).

Figure 7 shows the mean velocity results at a 2D Section through the purge burner. Dark blue represents still conditions at 0 m/s, red represents 26 m/s.

- The methane velocity is increased at the inlet nozzle to approximately 25 m/s at the base of the combustion chamber.
- The average velocity at the tip of the flare is reduced to 4.8 m/s due to the combustion process and venturi shape of the flare.
- The maximum velocity at 7.1 m above ground (area of interest for the Air Quality assessment) is 0.43 m/s.

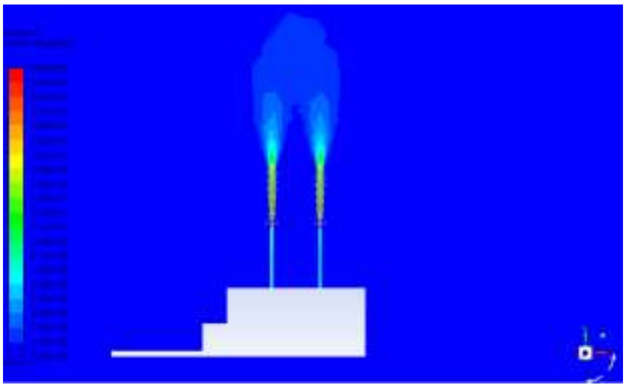


Fig. 7: Velocity Magnitude (m/s) - Near Calm Wind Condition, Approaching Wind =0.1 m/s

Figure 8 with contours of static temperature indicates the following:

- The predicted maximum flame temperature is 1700°C, occurring at approximately 1 m above the flare tip position.
- The mass-weighted average temperature at the tip of the flare is 910°C. The mass-weighted average is computed by dividing the summation of the value of the temperature multiplied by the absolute value of the dot product of the facet area and momentum vectors by the summation of the absolute value of the dot product of the facet area and momentum vectors:

$$\overline{T} = \frac{\int T\rho \overline{v} \cdot d\overline{A}}{\int \rho \overline{v} \cdot d\overline{A}}$$

This parameter is introduced to assess the impact of various operating conditions on the temperature profile at the tip of the flare.

- The predicted peak temperature at 7.1 m above ground is 480°C. Refer to Figure 8B
- The temperatures above the MPB System appear to drop very quickly

A plume iso-surface results (Temperature and Velocity) indicating the approximate three-dimensional spread of plume with an upstream ambient wind speed of 0.1 m/s is shown in Figure 9.

5.1.1.1 Impact of Atmospheric (Ambient) Air Temperature

The impact of the atmospheric air temperature is shown in Figure 10 and Figure 11. The following conclusions can be reached from the above figures:

- When the outside air temperature was reduced to 5°C (with no changes to other boundary conditions), the mass weighted average temperature at the tip of the flare increased slightly to 928°C. The predicted plume temperature at 7.1 m above ground also increased minimally to 481°C (Refer Figure 11B).
- When the outside air temperature was increased to 35°C (with no changes to other boundary conditions), the mass weighted average temperature at the tip of the flare was 880°C. The predicted plume temperature at 7.1 m above ground was slightly lower at 470°C (Refer Figure 12B).

From the above, a lower atmospheric ambient temperature leads to higher static temperature at the tip of the flare and slightly higher plume temperature at the area of interest (7.1 m above ground).

5.1.1.2 Scenario AFI-2 – Air to Fuel (Methane) Ratio = 6

For the AFI-2 scenario, the amount of entrained air in the system was increased by 400% without increasing the fuel mass flow rate. The results of the simulations are presented in Figure 12 and Figure 13. The following major conclusions can be reached from the above figures:

- The predicted peak velocity at 7.1 m above ground increases to 1.4 m/s. Refer Figure 13A
- The predicted peak static temperature at 7.1 m above ground increases to 620°C. Refer to Figure 13B
- The predicted plume equivalent diameter for the high temperature zone is approximately 1 m.

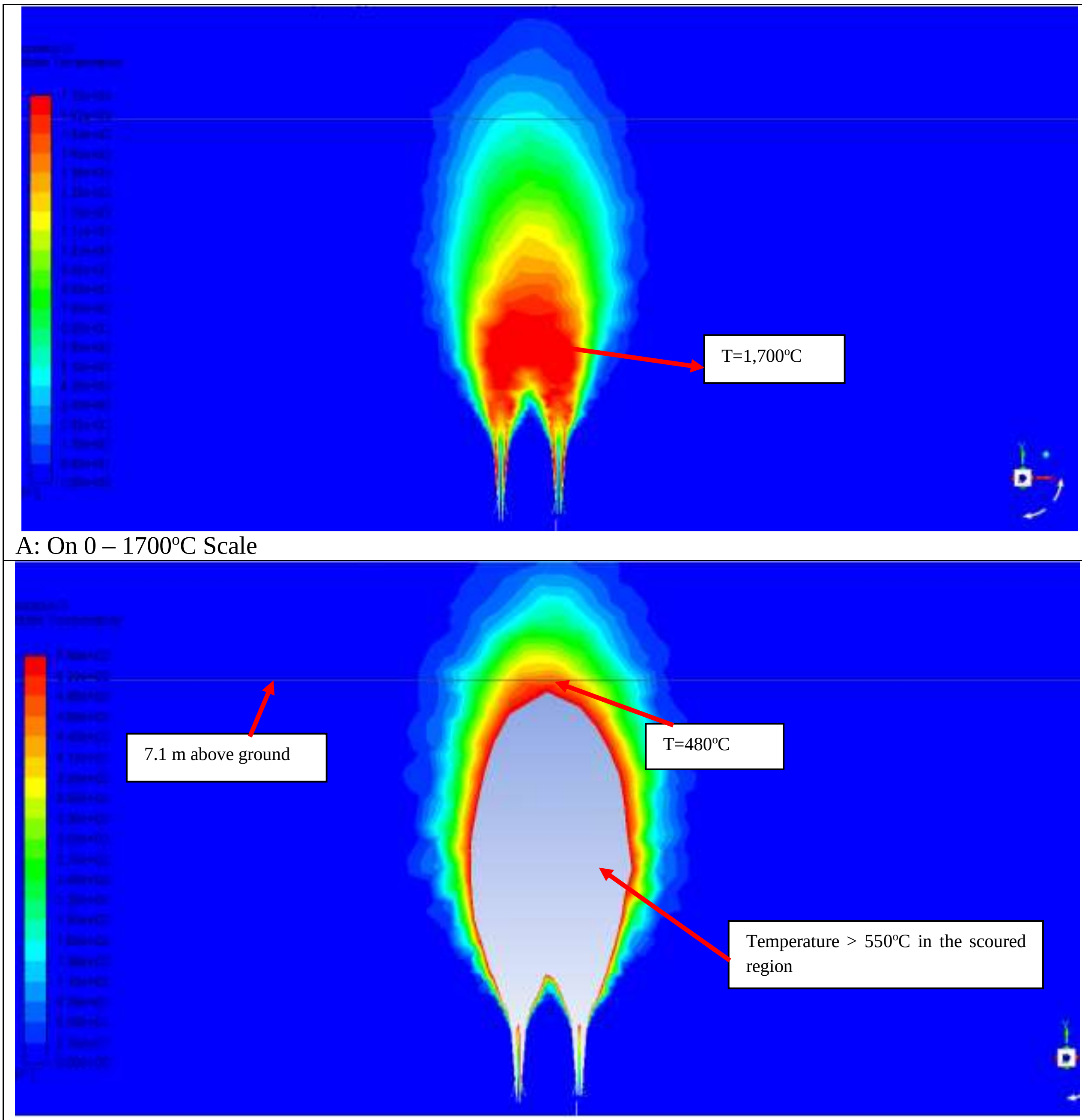
The following set of input parameters is obtained from the above CFD simulation and used for MITRE EPA simulations:

- H_{eff} = 7.1 m
- Dia_{eff} = 1.0 m
- Vp = 1.4 m/s
- Tp = 620°C

The MITRE EPA simulations are not subject to the current study. The following high-level comments are made based on the simulation:

- Based on the CFD results, the MITRE EPA results showed that the proposed flaring would comply with the MITRE EPA aviation safety thresholds (for critical vertical velocity and critical turbulence respectively) for all category aircraft, including Light GA and Light Sport (which covers helicopter operations).
- Based on the OHIA model. The MITRE EPA results showed that the proposed flaring would not comply with the MITRE EPA aviation safety thresholds. The OHIO EPA Model is standard industry practice in the US Gas Industry for the characterisation of a combustion source in terms of an “equivalent” stack; it is also commonly used globally.

The static temperature at the area of interest is reduced when the approaching wind is increased to 0.28 m/s (1 km/hr) due to wind interaction with the flame. Refer Figure 14.

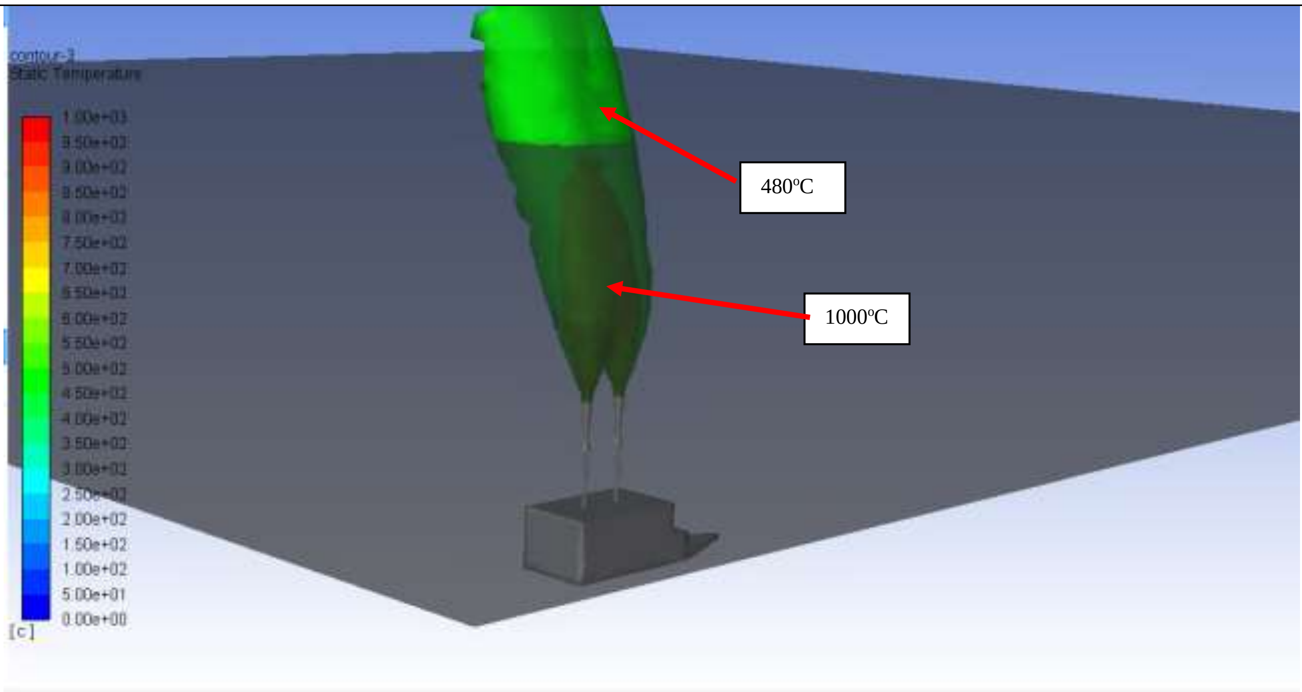


A: On 0 – 1700ºC Scale

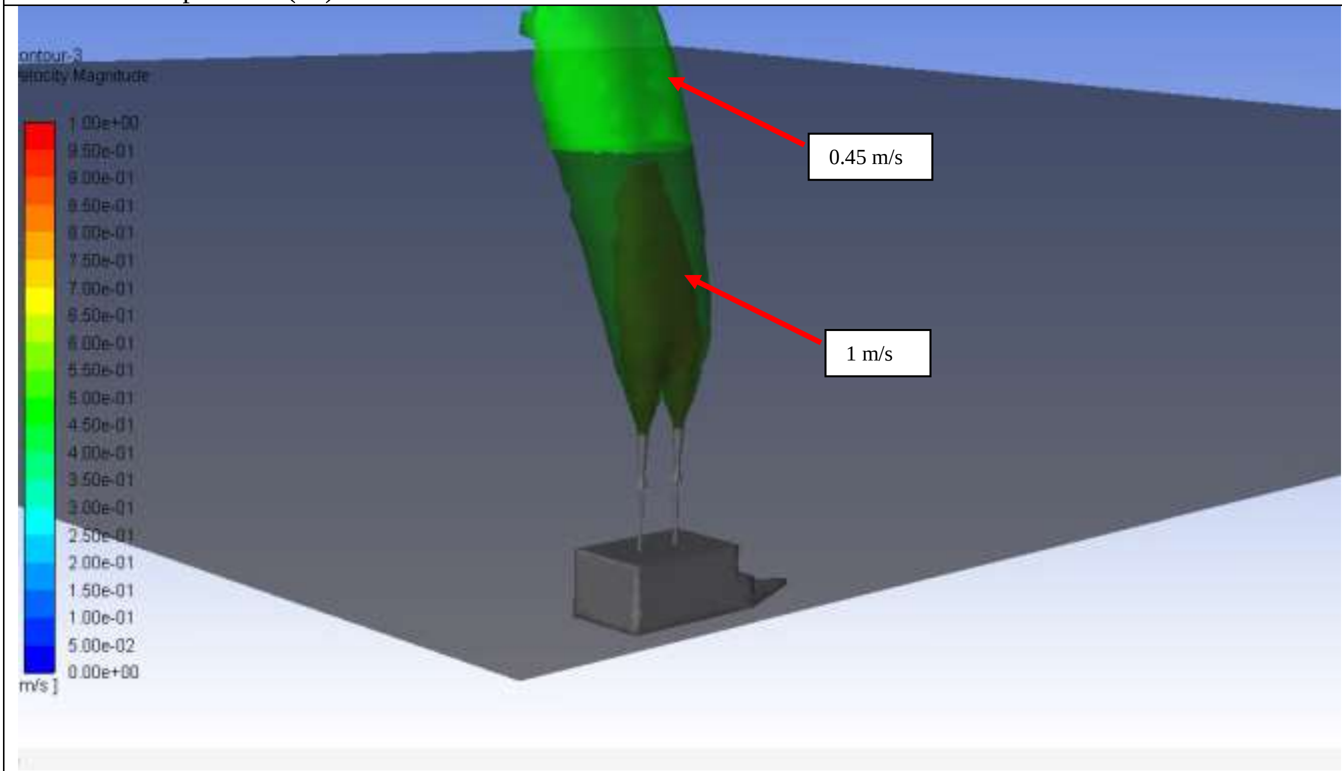
B: On 0 – 650 ºC Scale

Fig. 8: Static Temperature (ºC) - Atmospheric Temperature = 15ºC (Near Calm Wind Condition, Approaching Wind =0.1 m/s)

DOI: 10.37394/232015.2023.19.52

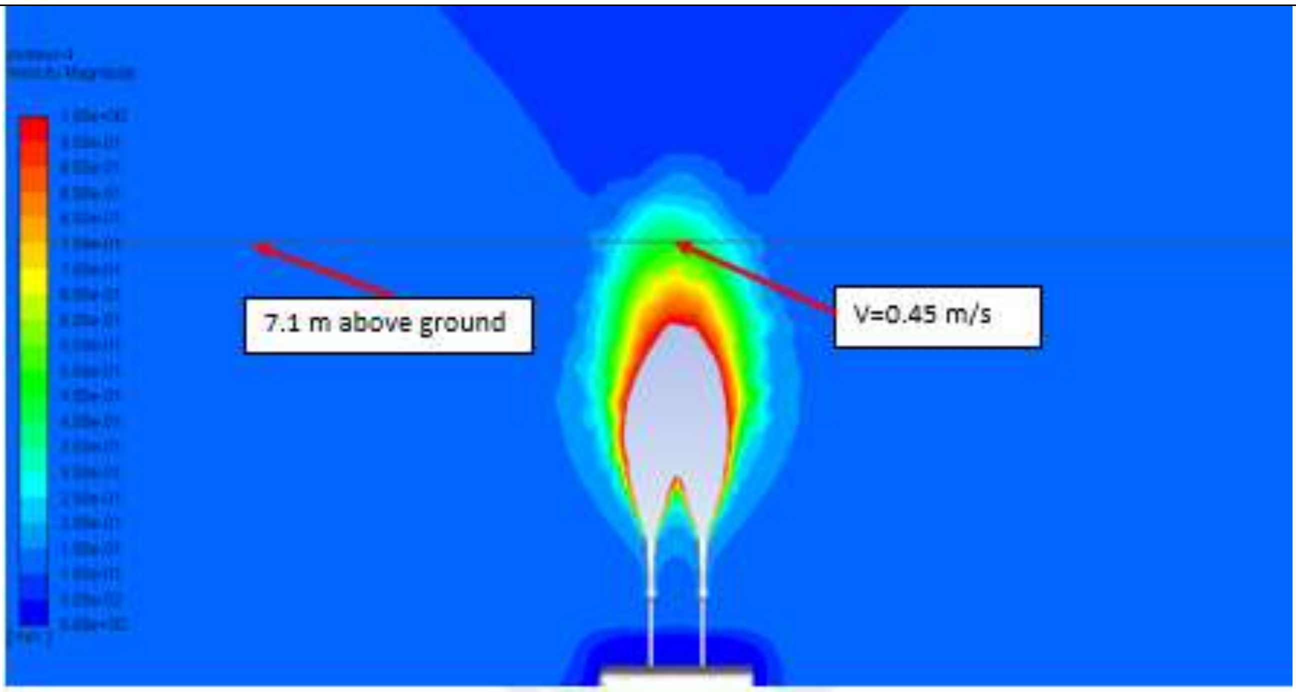


A: Static Temperature (ºC)

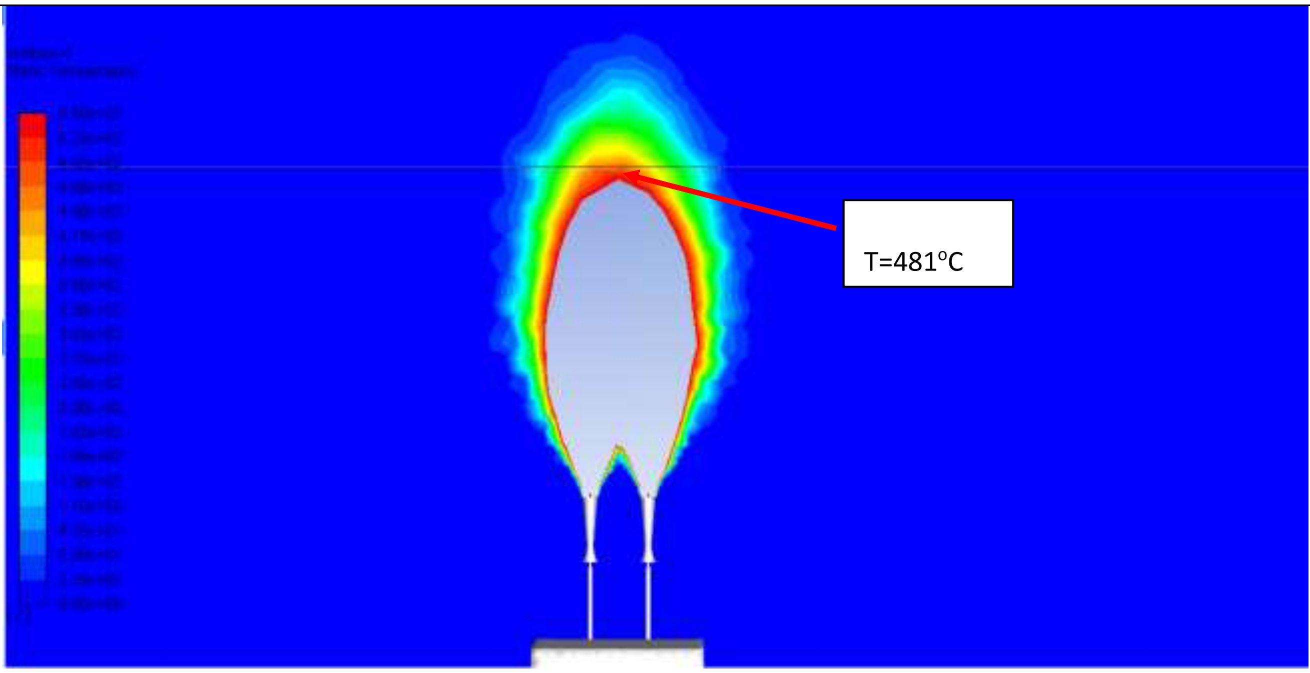


B: Velocity Magnitudes (m/s)

Fig. 9: Iso Surface of Velocity Magnitude (m/s) and Static Temperature (ºC) - Near Calm Wind Condition, Approaching Wind =0.1 m/s

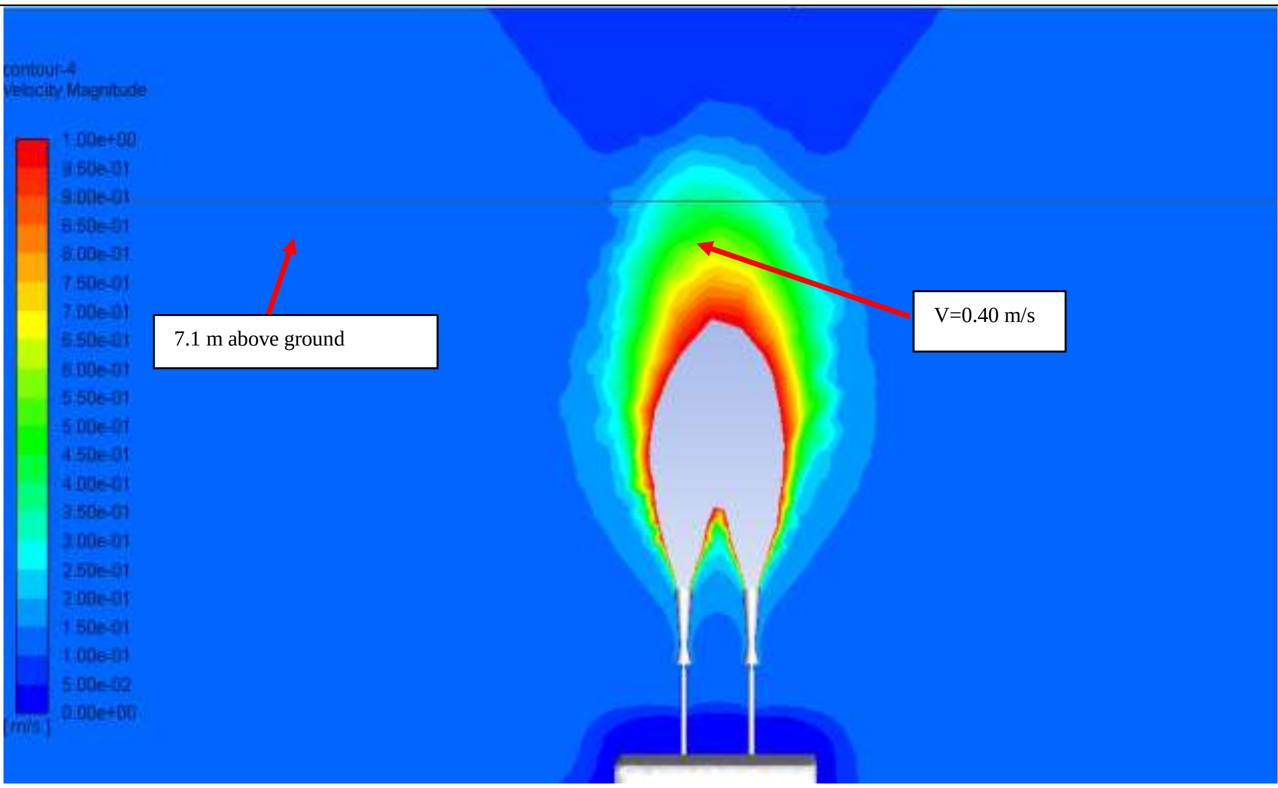


A: On 0 – 1 m/s Scale

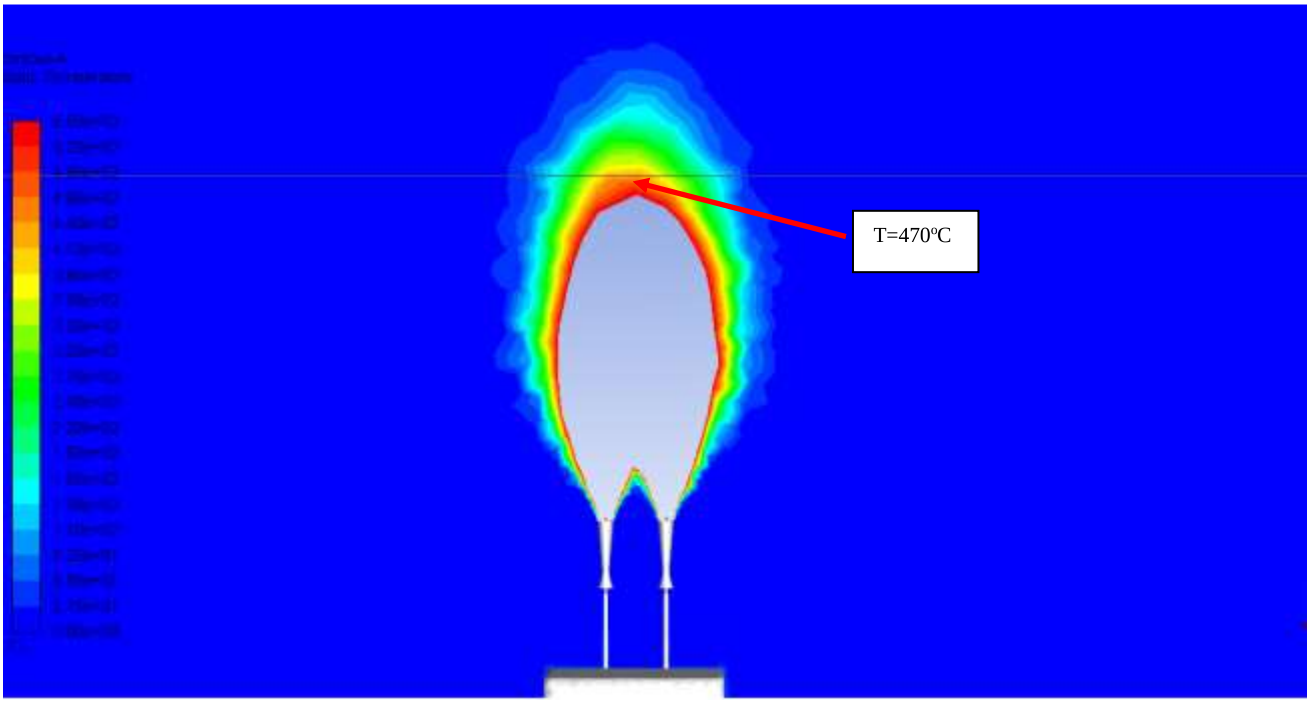


B: Static Temperature on 0 – 550°C Scale

Fig. 10: Static Temperature (°C) - Atmospheric Temperature = 15°C (Near Calm Wind Condition, Approaching Wind =0.1 m/s)

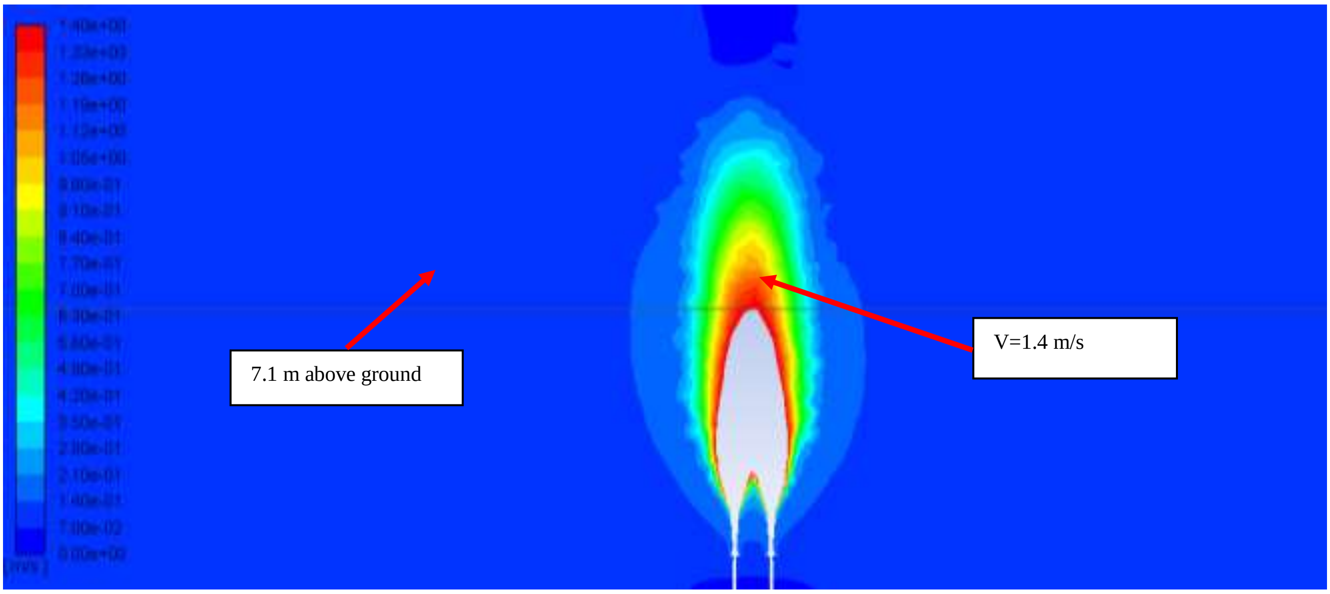


A: Velocity Magnitudes on 0 – 1 m/s Scale

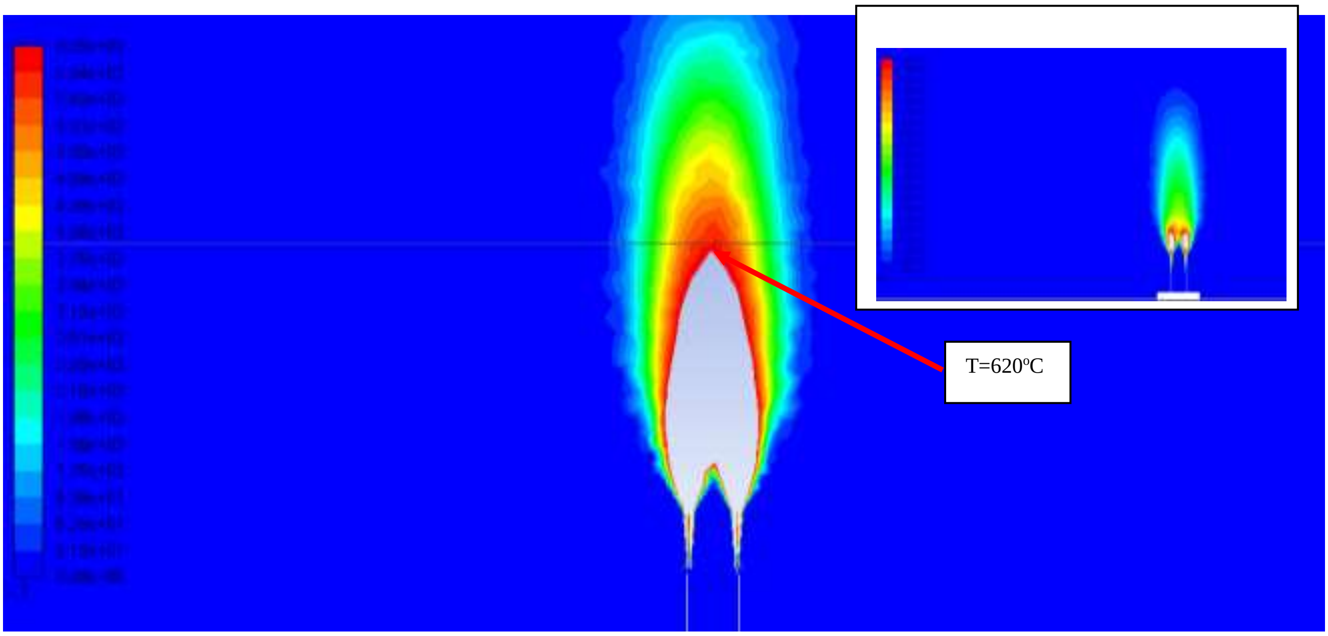


B: Static Temperature on 0 – 550ºC Scale

Fig. 11: Static Temperature (ºC) - Atmospheric Temperature = 35ºC (Near Calm Wind Condition, Approaching Wind =0.1 m/s)

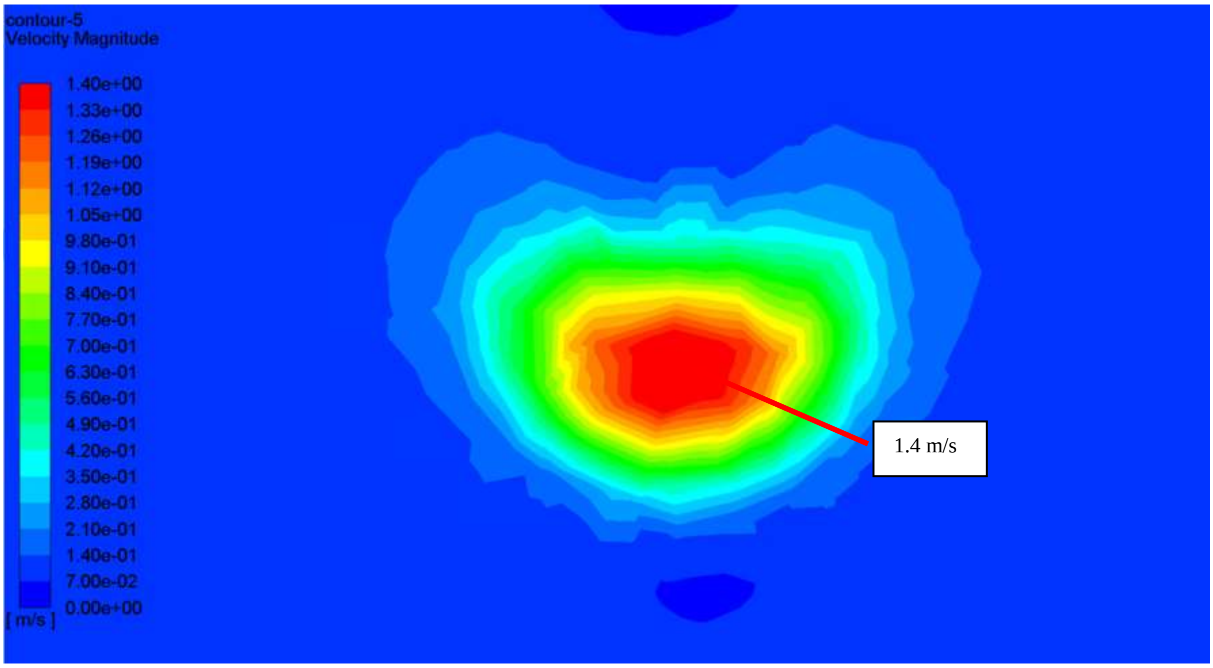


A: Velocity Magnitudes on 0 – 1.4 m/s Scale

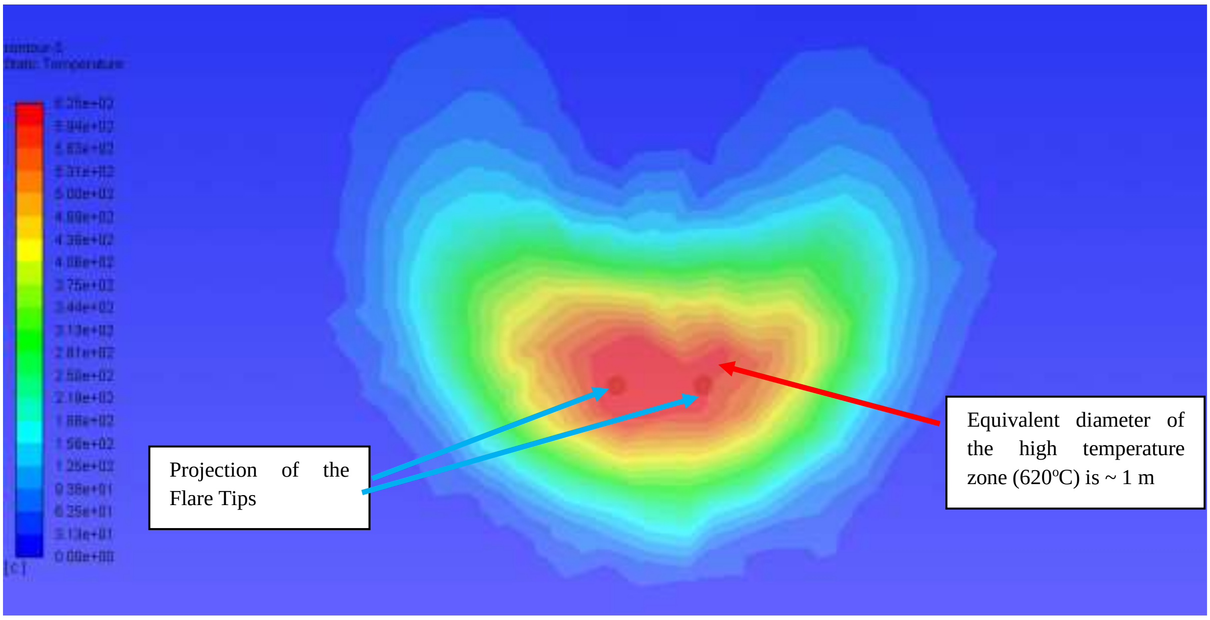


B: Static Temperature on 0 – 625ºC Scale

Fig. 12: Contours of Plume Velocity (m/s) and Temperature (°C) – Atmospheric Temperature = 5°C (Near Calm Wind Condition, Approaching Wind =0.1 m/s)

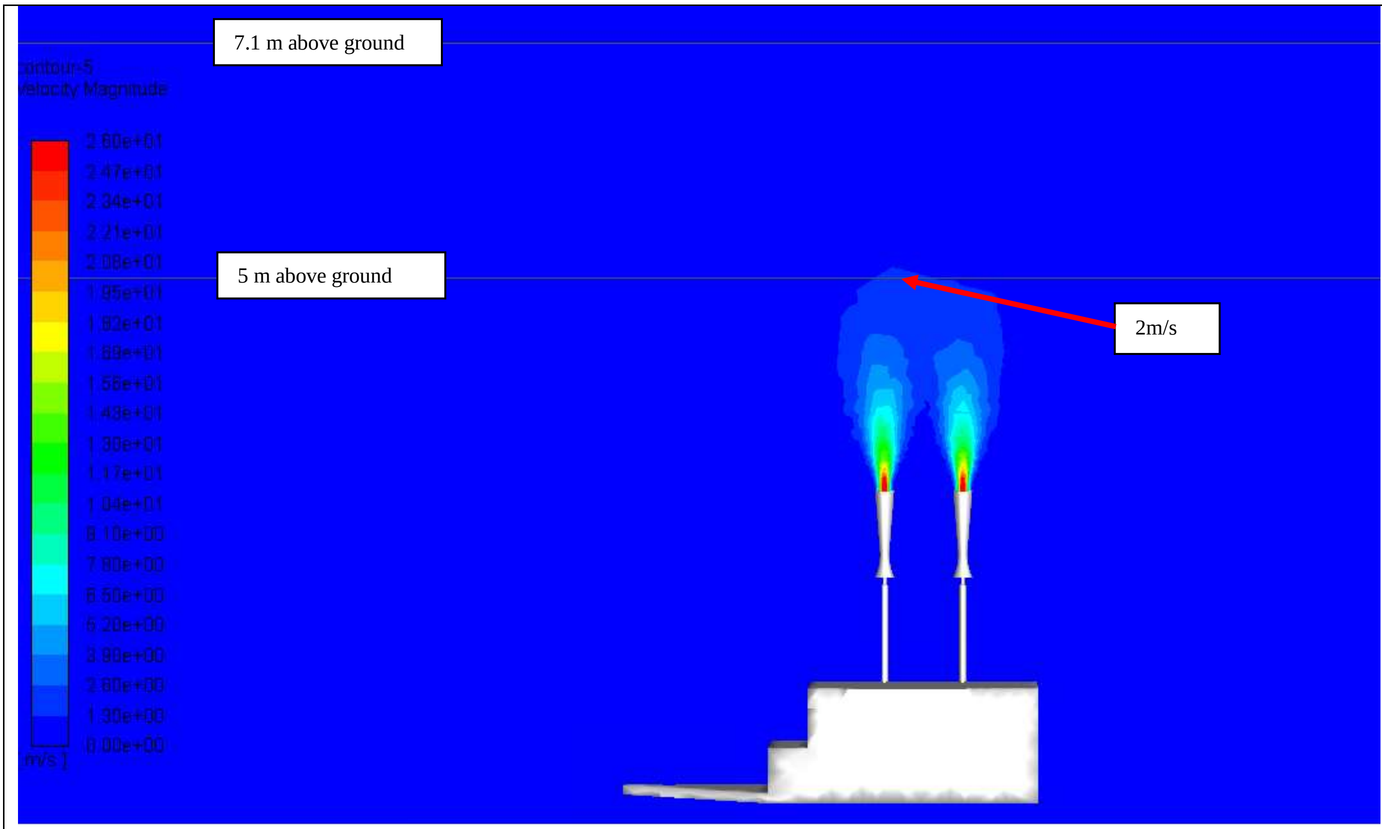


A: Velocity Magnitudes on 0 – 1.4 m/s Scale

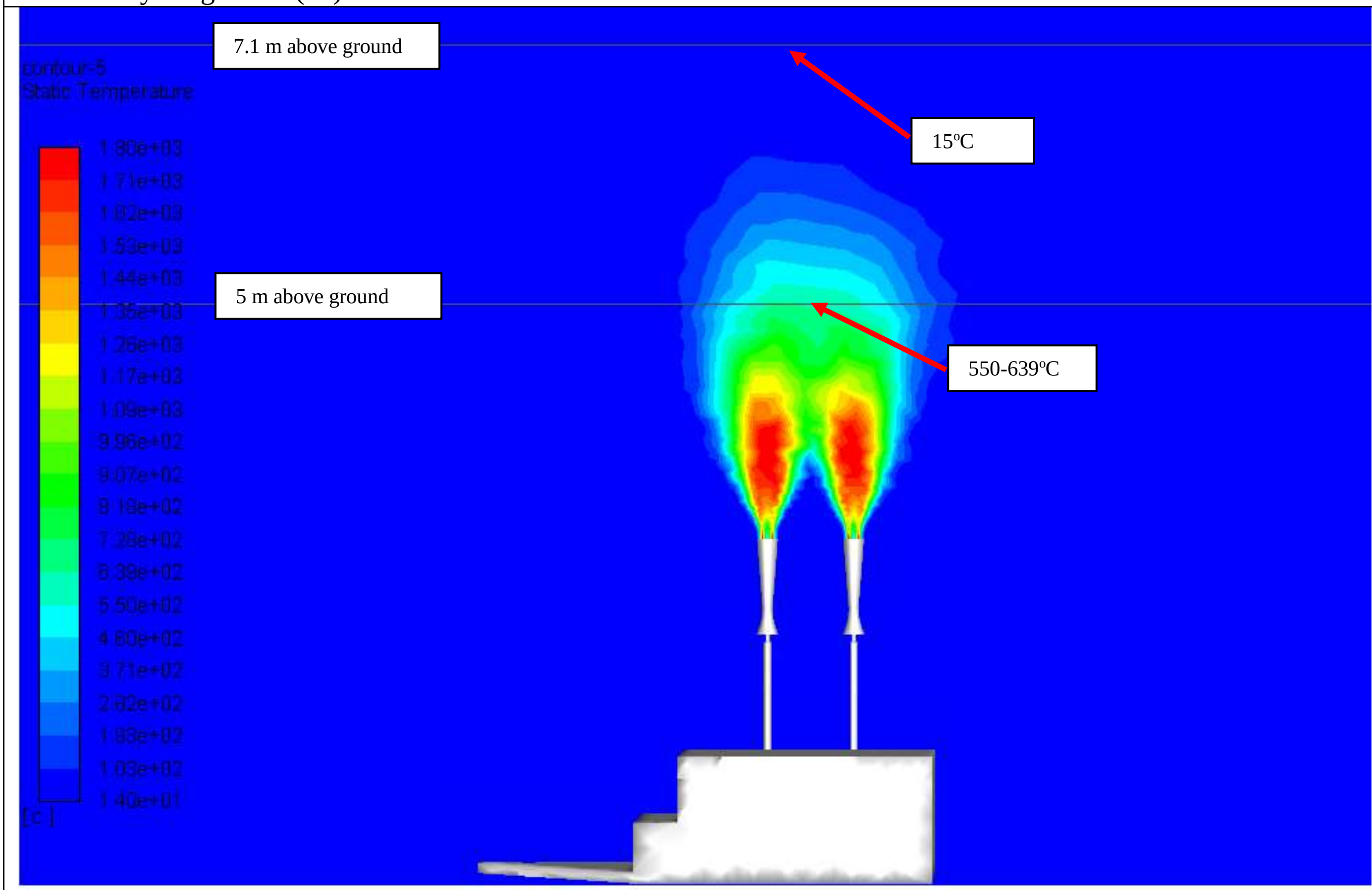


B: Static Temperature on 0 – 625ºC Scale

Fig. 13: Contours of Velocity Magnitude (m/s) and Plume Temperature (ºC) at 7.1 m above Ground (Near Calm Wind Condition, Approaching Wind =0.1 m/s)



A: Velocity Magnitude (ºC)



B: Velocity Magnitudes (m/s)

Fig. 14: Contours of Plume Velocity (m/s) and Temperature (°C) – Atmospheric Temperature = 15°C (Very Light Breeze Wind Condition, Approaching Wind =0.28 m/s)

6 In-Situ Measurements and Validation of CFD Results

Subsequently , a test was carried out using a similar Mobile Purge Burner (MPB) unit. The distance between the flare centrelines in the tested unit was 700 mm higher than that in the CFD model.

The test environment cannot be controlled precisely due to change to the mean and gust wind speeds.

6.1 Test Conditions

The test was undertaken under the following test conditions:

- Test Date: Wednesday, 16 February 2022
- Test Time & Duration: 9:30 am / ~15 minutes.
- Ambient Temperature: 21.6°C (recorded at the nearby Bureau of Met Weather Station)
- Ambient Mean Velocity: 4 km/h (recorded at the nearby Bureau of Met Weather Station). Lower wind speeds are anticipated at the test location due to shielding from suburban development.
- Rain: NIL

6.2 Measurement Instrument

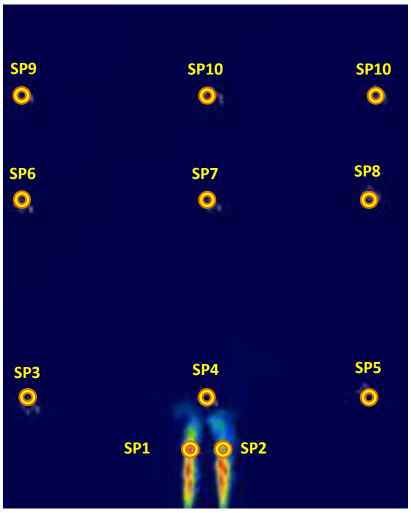
Infrared Vido Camera was used to measure plume details at the areas of interest. The Video Camera was placed at a distance of 50 m from the MPB system. Spot Readings were taken at a number of locations above the MPB flared tubes, ranging from 5 m above ground to 22 m above ground.

6.3 Key Output

A key output test from the above trial has been reproduced in Figure 15.

A - Snapshot from Video Recording 3

Measurements	
Sp1	423.3 °C
Sp2	298.9 °C
Sp3	18.1 °C
Sp4	36.2 °C
Sp5	18.7 °C
Sp6	13.9 °C
Sp7	16.6 °C
Sp8	16.9 °C
Sp9	16.2 °C
Sp10	14.8 °C
Sp11	15.7 °C



B – Time Recording of Peak Flame Temperatures

The chart below shows the mean peak readings (of two flares) recorded with the captured footage clip (~1min).
Mean readings typically range between 440°C and 500°C.
Extremes range from below 400°C up to just below 600°C

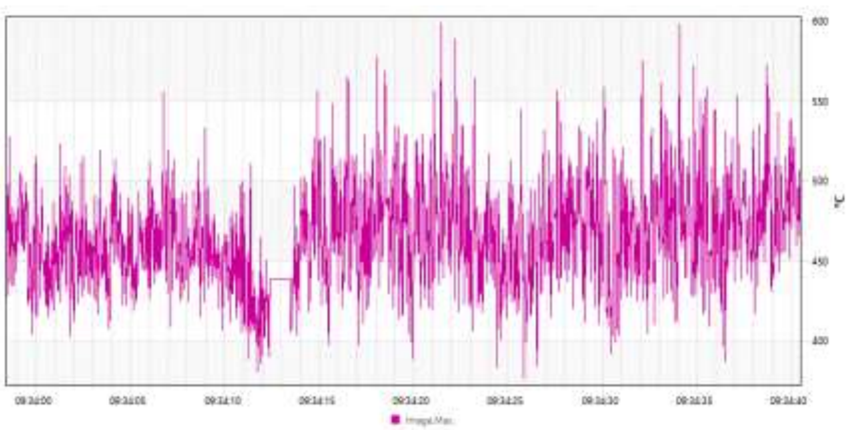


Fig. 15: Video Output 3 (camera viewed sideways)

Key results include the following:

- Height = 5 m above ground:
  - Temperatures at SP1 & SP2 = 491.0°C & 353.9°C
- Height = 22 m above ground:
  - Temperatures at SP9, SP10 & SP11 = 16.2°C, 14.8°C & 15.7°C

Based on the video output from the Field Trial, the following conclusions were reached:

- Plume temperatures within the combustion zone of the flares were very much in line with the temperatures predicted by the CFD Simulation study.
- The CFD Plume Rise Model, which adopts higher plume temperatures and velocities due to unchanged boundary conditions (e.g., mean wind speed) than indicated in the test study, would therefore appear to be moderately conservative.
- Plume temperatures above the MPB System appear to drop very quickly, such that the plume temperature fell from just under 500°C at 5 m above ground level to around 15-16°C (and close to the ambient temperature) at 22 m above ground. Again, this is consistent with the CFD Study results.

4 Conclusion

The risk of turbulence and upset being encountered by a range of aircraft types that operate through a rising plume is typically assessed using MITRE EPA simulations. Key input parameters to MITER EPA include Effective Height of the Plume, Equivalent Exhaust Emission Point, Velocity of the Plume at the Equivalent Exhaust Emission and the Equivalent Exhaust Emission Point Tp (ºC).

It is a common practice to use series of "best estimate" Scenarios (e.g., OHIO Model) to predict key input parameters to the MITRE EPA simulations. The best estimate scenarios may generate a very conservative input and are not suitable for all flare systems due to changes to geometry, number of stacks, combustion parameters, etc.

The current study presents a CFD as a reliable tool to determine the flame plume characteristics (Effective Height, Effective Diameter, Temperature and Velocity) on the example of trailer-mounted Mobile Purge Burner (MPB) system for a project site in Sydney Australia. The assessed MPB consists of two-flare stacks.

The CFD simulations accounted for the turbulent flow with chemical species mixing and reaction and utilised an advanced radiation model to solve participating radiation in the combusted zones.

A subsequent experimental test of a similar trailer mounted MPB system has validated the CFD results. Plume temperatures within the combustion zone of the flares were very much in line with the temperatures predicted by the CFD Simulation study. Plume temperatures above the MPB System appear to drop very quickly, such that the plume temperature fell from just under 500°C at 5 m above ground level to around 15-16°C (and close to the ambient temperature) at 22 m above ground. Again, this is consistent with the CFD Study results.

This study assesses all the parameters that have impact on the accuracy of the numerical model including, computational domain, mesh distribution, numerical scheme and flame plume characteristics including ambient conditions (wind speed and temperature) and combustion under various air to fuel ratio.

The accuracy of the results has been improved by optimizing the mesh size in the computational domain via an initial mesh sensitivity analysis, the use of second-order numerical schemes for the discretization of pressure and momentum equations, and the use of a staged approach and powerful hardware to enable modelling chemical reaction, plume dispersion, turbulence, and radiation in the computational domain.

References:

[1] K. Spillane, Observations of Plume Trajectories in the Initial Momentum Influenced Phase and Parameterization of Entrainment, *Atmospheric Environment*, Vol.11, No7, pp 1207-1214, 1983.

[2] Australian Government, Civil Aviation Safety Authority, Plume Rise Assessment", Advisory Circular, AC 139.E-0.2 V1.0, D19/177237, May 2022.

[3] TAPM (The Air Pollution Model), SCIRO, http://www.csiro.au/Outcomes/Environment/Population-Sustainability/TAPM.aspx

[4] P. Hurley, P. Manins, et al, Year-long, High-Resolution Urban Airshed Modelling: Verification of TAPM Predictions of Smog and Particles in Melbourne, Australia, *Atmospheric Environment*, Vol.37, no.14, 2003, pp 1899-1910.

[5] https://www.mitre.org

[6] *Ansys Fluent Theory Manual*, ANSYS, USA, 2022.

[7] D. Gouldy, J. Hopper, Determining the Risk

of Experiencing Severe Turbulence when Flying through and Exhaust Plume, *Environmental Science*, Corpus ID 204831699, 2014.

[8] B. Magnussen and B. Hjertager, On Mathematical Models of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion, *In 16th Symp. (Int'l.) on Combusti*on, The Combustion Institute, 1976.

[9] H. Gabler, R. Yetter, and I. Glassman, Symmetric Whirl Combustion: A New Approach for Non-premixed Low NOx Gas Turbine Combustor Design. *Proceedings of the 34th AIAA/ASME/SAE/ASEE Joint Propulsion Conference*, Cleveland, USA, 1998.

[10] S. Kim, K. Ito, D. Yoshihara and T. Wakisaka, Application of a Genetic Algorithm to the Optimization of Rate Constants in Chemical Kinetic Models for Combustion Simulation of HCCI engines, *JSME Int. J. Ser. B*, Vol.48, No.717, 2005.

[11] P. Wang, The Model Constant A of the Eddy Dissipation Model, *Progress in Computational Fluid Dynamics International Journal*, *Vol.*16, No.2, 2016.

[12] E. hui and G. Raithby. Computation of Radiant Heat Transfer on a Non-Orthogonal Mesh Using the Finite-Volume Method, *Numerical Heat Transfer*, Part B. Vol.23, 1993, . pp. 269–288.

[13] N. H. Al-Khalidy, Integrated Approach to Predict Wind Resource Energy from an Urban Wind Turbine in a complex Built Environment, *Fluid Mechanic*, Vol.7, No.2, 2021, pp. 17-28.

[14] N. H. Al-Khalidy, Building Generated Wind Shear and Turbulence Prediction Utilising Computational Fluid Dynamics, *WSEAS Transactions On Fluid Mechanics***,** Vol.13, 2018, pp.126-135.

[15] N. Bhondwe, C.D. Koshti "Analysis of Premixed Methane-Air Combustion" using Laser Ignition" International Engineering *Research Journal Special Edition PGCON-MECV H-2017*, 2017.

Contribution of Individual Authors to the Creation of a Scientific Article (Ghostwriting Policy)

The author contributed in the present research, at all stages from the formulation of the problem to the final findings and solution.

Sources of Funding for Research Presented in a Scientific Article or Scientific Article Itself

No funding was received for conducting this study.

Conflict of Interest

The authors have no conflict of interest to declare.