Journal of Solid and Fluid Mechanics (JSFM), 15(4): 9-11, 2025

Shahrood University of Technology

Journal of Solid and Fluid Mechanics (JSFM)

DOI: 10.22044/jsfm.2025.16058.3959

Manufacturing Engineering of Iran

Investigating the Effectiveness of Turbulence Models in Numerical Simulation of Rocket Exhaust Plume

Hossein Ansarian[1*], Mostafa Hadidoolabi[2]

[1]PhD, Malek Ashtar University of Technology
[2]Assoc. Prof., Malek Ashtar University of Technology
*Corresponding author: hossein.ansarian@gmail.com

Received: 04/11/2024 Revised: 06/04/2025 Accepted: 08/31/2025

Abstract

This study evaluates the effectiveness of various turbulence models for the numerical simulation of a solid rocket motor exhaust plume. The complex nature of plume flow includes compressibility effects, shock wave interactions, secondary combustion, and turbulence, all of which require accurate computational modeling. In this research, Reynolds-Averaged Navier-Stokes (RANS) methods are employed using ANSYS Fluent to analyze the plume flow field. Several turbulence models, including k-ε, k-ω, the modified SST k-ω, and adjusted versions of k-ε, are examined to predict shockwave locations and turbulence dissipation effects. The numerical results are validated against experimental data obtained from video analysis of static rocket motor firings. The findings indicate that the standard k-ε model provides better agreement with observed shock positions, whereas modifying the turbulent Mach number parameter in the k-ω model significantly improves plume structure prediction. The study highlights that modifying turbulence models can enhance plume simulation accuracy, improving predictions of thermal radiation and infrared signatures for aerospace and defense applications. These findings are crucial for refining exhaust plume models used in missile signature analysis and launch vehicle design.

Keywords: Rocket Exhaust Plume, Computational Fluid Dynamics, Turbulence Models, Numerical Simulation, ANSYS Fluent.

1. Introduction

Rocket plumes form as a result of thrust generation and involve high-temperature, compressible, turbulent, and multi-species flows where chemical reactions occur. Modeling rocket plumes using computational fluid dynamics offers a valuable way for predicting missile detectability and analyzing system vulnerability.

A useful set of test data has been collected by the QinetiQ Missile Propulsion Group [1] from over thirty different types of rocket motors. These rocket motor firing tests are typically static tests, where thrust, radiation, temperature, and pressure data are recorded. One well-known numerical code for predicting the baseline flow field of plumes is REP3 [2], designed as a low-cost computational model for axisymmetric 2D plumes using a parabolic solver. A number of recent studies have focused on turbulence modeling for these flows. Watts [1] compared experimental results with REP3 and the k-ε and k-ω models, reporting that Reynolds-averaged models predict longer plume lengths. Childs and Matsuno [3] developed a new turbulence model based on the SST k-ω framework to simulate plumes, accounting for compressibility, streamline curvature, and swirling effects.

A comprehensive evaluation comparing different turbulence models in numerical simulations of rocket plume flows has yet to be conducted. Additionally, newer versions of numerical software offer improved turbulence modeling options to address existing shortcomings, necessitating further investigation into their impact on the studied flows. The present study focuses on the exhaust plume of a small tactical solid-fuel rocket motor without aluminum or solid particles in the flow field. ANSYS Fluent is used as the numerical software, and for comparison with prior results, data from the REP3 code, as presented by Watts [1], is utilized. All models assume that the rocket motor has been fired statically at sea level with negligible wind speeds.

2. Methodology

The Design Modeler package was applied to prepare the geometry, and the ANSYS meshing software generated the computational grid. The 2D axisymmetric structured mesh consists of approximately 40,000 cells. A grid independence study, based on static temperature variations along the plume centerline, was conducted for different turbulence models using four grids with 30,000, 35,000, 40,000, and 45,000 cells. Convergence was achieved with the third grid.

The CEA2 combustion code [4] provided the required boundary conditions for the rocket nozzle exit which include velocity, static pressure and temperature, density, specific heat capacity, pressure, and mass fractions of the product species. Reactions were modeled as finite-rate volumetric reactions, with 25 reactions configured to calculate secondary combustion in the plume flow field. Species in Fluent were modeled as a gas mixture, with density set as an ideal gas. For each chemical species, C_p was defined as a piecewise polynomial function of temperature. Second-order spatial discretization was used for both flow and turbulence equations. The Reynolds-averaged models employed in this study include Spalart-Allmaras, standard k-ε, and k-ω variants.

3. Results and Discussion

The first set of simulations used standard settings for the Spalart-Allmaras, k-ε, and k-ω models. Figure 1 shows temperature contours of the plume flow field computed using the Spalart-Allmaras and standard k-ε turbulence models, alongside an image of the static rocket motor firing. The Spalart-Allmaras model fails to capture the real plume flow characteristics well, while the k-ε model shows better agreement.

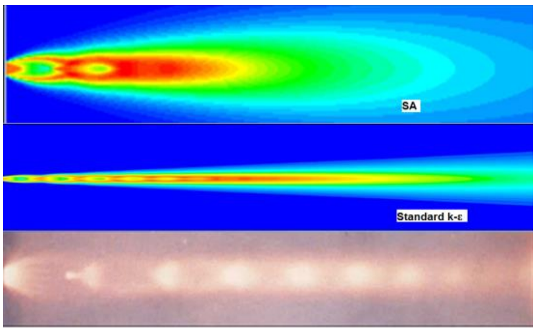

Figure 1. SA and standard k-ε model compared to the real image

Figure 2 plots the temperature variations along the centerline, comparing the SA, standard k-ε, standard k-ω, and REP3 results. The position of the first shock wave remains largely unaffected by turbulence model changes and is determined by the boundary conditions set at the nozzle exit plane pressure. Further downstream, shock structures vary in number and position depending on the turbulence model. The REP3 code predicts a shorter and wider plume compared to Fluent, higher temperatures for the first shock wave, and fewer shock waves than the other turbulence models. The SA model is unsuitable for this flow field. The standard k-ε model produces a shock count similar to experimental observations in the plume core, while the standard k-ω model generates nearly twice as many shocks. Several modified k-ε turbulence models exist, with adjusted constants for specific flow types. Results show that modifying these constants significantly impacts shock structures in the region where plume transition occurs and a viscous mixing layer is present.

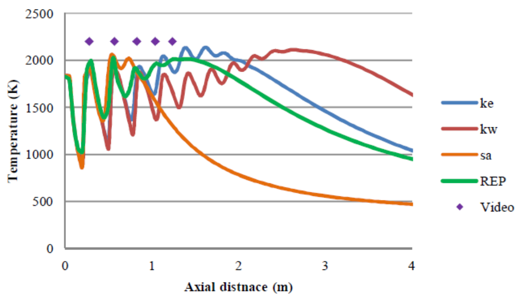

Figure 2. Results of the standard RANS models and the experimental data for the centerline temperature

Figure 3 compares axial static temperature predictions along the centerline for standard k- ω, SST k- ω, and k-ε models against experimental data for shock wave positions. Since the SST k- ω model blends the standard k- ω (near walls) and k-ε (free stream) models, it yields results similar to the standard k-ε model.

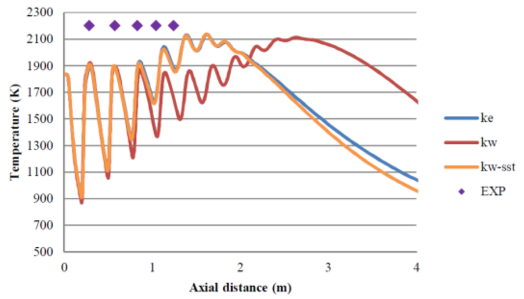

Figure 3. Results of the k-ε, standard k-ω, and SST k-ω models and the experimental data for the centerline temperature

The standard k-ω model offers three modifications: low-Reynolds-number correction, compressibility correction, and shear flow correction. Figure 4 displays the temperature contours which highlight the effects of deactivating these options separately. The low-Reynolds-number correction has no effect on the plume flow field, as it pertains to near-wall regions. Disabling the shear flow correction reduces the turbulent mixing zone downstream but does not alter shock structures in the turbulent core. When the compressibility correction is disabled, it drastically shortens the plume. The default turbulent Mach number (M_{t0}) for compressibility correction is 0.25; adjusting this to 0.5 yields the best results.

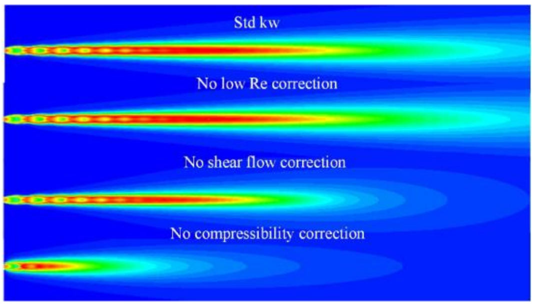

Figure 4. Temperature contours for the k-ω model with different settings

4. Conclusions

The plume flow field simulations demonstrate that Fluent models generally align better with measured shock wave positions and counts compared to REP3. The standard k-ε model matches experimental shock wave counts and axial positions more closely than the standard k- ω turbulence model. However, improvements to both k-ε and k- ω models can enhance plume flow field accuracy. These models can be used for predicting the plume detectability and evaluating the results compared to the measured radiative intensity levels.

5. References

[1] Watts D (2016) Assessing computational fluid dynamics turbulence models for rocket exhaust plume simulation. *52nd AIAA/SAE/ASEE Joint Propulsion Conference*, USA, 4591.

[2] Cousins JM (1982) Calculations of conditions in an axisymmetric rocket exhaust plume: The REP3 computer program. *RARDE*, Westcott, UK.

[3] Childs RE, Matsuno KV (2024) Novel turbulence model tailored for complex rocket motor exhaust jets. *AIAA AVIATION Forum and Exposition*.

[4] Sanford G, McBride B (1996) Computer program for calculation of complex chemical equilibrium compositions and applications. *NASA Reference Publication*.