Three-Dimensional Numerical Simulation of Liquid Oxygen/Kerosene Engine Exhaust Plume Impingement on Flame Trench Postprint

Authors: Cai Honghua, Nie Wansheng, Su Lingyu, Hou Zhiyong

Date: 2017-11-07T00:00:00+00:00

Abstract

To investigate the flow field characteristics of a liquid oxygen/kerosene engine plume impinging on deflector trenches, three-dimensional numerical simulations were conducted using CFD software for the impinging flow field on three trench configurations: without a deflection device, with a wedge-shaped deflection device, and with a conical deflection device, and comparative analysis was performed on the plume gas flow. The results indicate that: when the plume gas vertically impinges on a trench without a deflection device, gas flowing along the trench side walls and upward-reflected gas create a high-temperature effect on the environment directly above the trench bottom; the wedge-shaped deflection device eliminates the high-temperature effect from both side-wall flow and upward-reflected gas; the conical deflection device eliminates the high-temperature effect from upward-reflected gas but has room for improvement in preventing side-wall gas flow; compared to the no-device case, the maximum surface pressures on the wedge-shaped and conical devices increased by 10.31% and 33.81%, respectively, while their maximum temperatures under plume impingement decreased by 4.04% and 8.95%, respectively.

Full Text

Three-dimensional Numerical Simulation of LOX/Kerosene Engine Exhaust Plume Impinging on the Diversion Trough

CAI Hong-Hua, NIE Wan-Sheng, SU Ling-Yu, HOU Zhi-Yong
(Department of Aerospace Equipment, The Academy of Equipment, Beijing 101416, China)

Abstract

To investigate the flow field characteristics of a LOX/kerosene engine exhaust plume impinging on a diversion trough, three-dimensional numerical simulations were performed using CFD software for three different configurations: a trough without deflector, a trough with a wedge deflector, and a trough with a conical deflector. The flow fields of plumes impinging on these different troughs were compared and analyzed. The results indicate that when the LOX/kerosene engine exhaust plume vertically impacts a trough without deflector, the gas flowing along the side surfaces of the trough and the upward-reflected gas create high-temperature effects on the environment directly above the trough base. The wedge deflector effectively eliminates this high-temperature effect by preventing gas flow along the side surfaces and upward reflection. The conical deflector can only prevent the high-temperature effect from upward-reflected gas, but still allows some gas to flow along the side surfaces, leaving room for improvement. Compared with the no-deflector case, the maximum pressure on the wedge and conical deflectors increased by 10.31% and 33.81%, respectively, while the maximum temperature decreased by 4.04% and 8.95%, respectively.

Keywords: LOX/kerosene engine; exhaust plume; impingement; deflector; numerical simulation

0 Introduction

LOX/kerosene engines have become the development direction for liquid rocket engines due to their high thrust, non-toxicity, and environmental cleanliness. During rocket launch, high-speed, high-temperature exhaust gases are ejected. In the initial launch phase and during test stand operations, the engine is relatively close to the ground platform. The thermal and dynamic impact effects of the exhaust plume can not only cause vibration responses in the launch structure and initial flight disturbances to the vehicle, but also threaten other equipment on the ground platform. Therefore, it is necessary to study the impact effects of exhaust gases on launch platforms.

In spacecraft design, computational studies are typically employed to investigate the thermal effects of rocket engine exhaust plumes on downstream structures near the nozzle exit. Launch sites and engine test stands utilize diversion troughs built beneath the platform to guide plume flow and mitigate impact effects on the platform and facilities. To reduce the ablative impact on diversion troughs and improve their safety and service life, researchers continuously investigate deflectors of various shapes.

In recent years, numerical studies on rocket engine plume impingement on deflectors have advanced considerably. Ma et al. conducted numerical research on solid rocket motor plume impact on launch platforms. Mayer et al. developed a program to investigate heat transfer characteristics associated with liquid rocket engine plume impingement. Seiji Tsutsumi et al. performed simulations of the impingement flow field during the initial stage of solid rocket launch, studying the acoustic characteristics of plumes impinging on deflectors of different shapes. Michael et al. developed a simulation program for solid rocket motor impingement flow fields and validated it experimentally using a horizontal test stand. Jeffrey et al. used CFD software to simulate the flow field characteristics of rocket plumes impinging on diversion troughs with conical deflectors. Daniel C. Allgood et al. performed CFD simulations of hydrogen-oxygen engine plume impingement on diversion troughs for the Ares rocket series. Bruce T. Vu et al. used CFD to study the impingement flow field of Space Shuttle main engine and solid rocket booster plumes on troughs with wedge deflectors.

This paper presents a computational method for LOX/kerosene engine plume impingement on diversion troughs. The approach first computes the engine internal flow field, then uses the calculated nozzle throat section parameters as inlet boundary conditions for the plume impingement flow field calculation. This method fully accounts for the effects of combustion in the chamber and afterburning reactions on plume impingement. Based on this method, three-dimensional numerical calculations of LOX/kerosene engine plume impingement effects were performed using CFD software to investigate the flow field characteristics and patterns for different deflector geometries.

1.1 Physical Model

This study focuses on a LOX/kerosene engine with a total propellant flow rate of 390 kg/s and an oxygen/kerosene mixture ratio of 2.4, operating in a fuel-rich condition. The diversion trough and deflector structures are shown in [Figure 1: see original paper]. Figure 1(a) illustrates a simple trough without deflector, consisting of a bottom surface, side surfaces, and a ramp. The engine nozzle exit diameter is D, and the trough dimensions are: side height a = 7.64D (distance between nozzle exit and trough bottom), bottom width b = 6.37D, ramp length c = 12.74D, and bottom longitudinal width d = 6.37D. Figure 1(b) shows a wedge deflector installed on the trough bottom based on the configuration in Figure 1(a), while Figure 1(c) shows a conical deflector installed on the trough bottom. The curved surface designs of the wedge and conical deflectors follow reference [13].

1.2 Chemical Reaction Mechanism

Reference [14] compared computational results for liquid kerosene/gaseous hydrogen/gaseous oxygen and gaseous kerosene/gaseous hydrogen/gaseous oxygen engine internal flow fields, demonstrating that both methods yield correct flow field predictions. To account for afterburning effects on plume impingement, this study employs a simplified single-step reaction that models actual kerosene combustion as the oxidation of a kerosene surrogate fuel C12H23 into complete reaction products H2O and CO2. The numerical calculation involves four gas species: kerosene vapor, oxygen, carbon dioxide, and water vapor. While the single-step global reaction yields temperatures higher than actual values, it provides high computational efficiency. Since this study aims to compare and analyze the cooling performance of different troughs, the single-step global reaction approach yields correct comparative results. The reaction is represented as: 1223222CH+17.75O12CO+11.5HO

1.3 Governing Equations and Solution Models

The conservation-type three-dimensional Navier-Stokes equations for multi-component reacting flows serve as the governing equations for fluid motion, species and energy transport, and combustion. The generic form is:

$$\frac{\partial \mathbf{U}}{\partial t}+\frac{\partial \mathbf{F}}{\partial x}+\frac{\partial \mathbf{G}}{\partial y}+\frac{\partial \mathbf{H}}{\partial z}=\frac{\partial \mathbf{F}_v}{\partial x}+\frac{\partial \mathbf{G}_v}{\partial y}+\frac{\partial \mathbf{H}_v}{\partial z}+\mathbf{J}$$

where $\mathbf{U}$ is the vector of conserved variables, t is time, $\mathbf{F}$, $\mathbf{G}$, $\mathbf{H}$ are convective flux vectors, $\mathbf{F}_v$, $\mathbf{G}_v$, $\mathbf{H}_v$ are viscous flux vectors, and $\mathbf{J}$ is the source term. These equations represent the continuity, x-, y-, z-momentum, energy, and species conservation equations.

The N-S equations describing rocket engine combustion flow are strongly coupled nonlinear partial differential equations. The Pressure Implicit with Splitting of Operators (PISO) algorithm has been successfully applied to compute rocket engine combustion flows. The finite volume method discretizes the plume flow field governing equations, which are solved using the realizable k-ε two-equation model and the finite-rate/eddy-dissipation model. The Arrhenius formula calculates the chemical source terms.

1.4 Computational Mesh and Boundary Conditions

A mature mesh processing software was used for grid generation. Due to the 1/3 symmetry characteristic of the injector panel nozzle distribution, structured grids were employed for the three-dimensional engine internal flow field computational domain to improve efficiency, as shown in [Figure 2: see original paper]. Because the plume impingement flow field on the diversion trough and deflector exhibits 1/4 symmetry, a 1/4 symmetric flow field was selected as the computational domain for plume impingement simulations, as shown in [Figure 3: see original paper]. Following grid independence verification, final mesh schemes of 970,000 and 1,110,000 cells were selected for the engine internal flow field and plume impingement flow field calculations, respectively.

The engine internal flow field boundary conditions are defined as: mass flow inlet boundary of 1.178 kg/s with an oxygen/kerosene mixture ratio of 2.4; wall boundary with no-slip condition; pressure outlet boundary at 300 K and 101,325 Pa. The plume impingement flow field boundary conditions are: mass flow inlet boundary determined from the engine internal flow field calculation;

pressure far-field boundary at 300 K and 101,325 Pa; wall boundary with no-slip condition.

2.1 Model Validation

The present model was used to compute the plume impingement flow field under conditions specified in reference [16], and the results were compared with experimental data. Comparisons between computed and measured flow fields at different pressure ratios (NPR = nozzle total pressure/ambient pressure at nozzle exit) are shown in [Figure 4: see original paper], with computed velocity contours on the left and schlieren measurements from the literature on the right. The nozzle exit diameter is d = 2.54 cm, the contraction ratio of the nozzle convergent section upstream of the throat is approximately 5, and the distance between the ground plate and nozzle exit is h = 2d. Figure 4(a) shows the flow field comparison at NPR = 2.5, where oblique shock waves are present. Figures 4(b) and 4(c) show flow fields at NPR = 3.7 and NPR = 5.0, respectively, where oblique shocks are replaced by Mach disk phenomena that grow larger with increasing interaction intensity. Figure 5 compares experimental and simulated pressure distributions on the impingement plate surface at NPR = 3.7 for various h/d ratios. The computed flow field structures and plate surface pressure distributions agree well with experimental measurements, validating the accuracy of the plume impingement flow field model.

2.2 Engine Internal Flow Field Results and Analysis

The LOX/kerosene engine internal combustion flow field was first computed to obtain temperature and velocity distributions, as shown in [Figure 6: see original paper]. The fuel-rich oxygen and kerosene mixture undergoes complete chemical reaction in the engine to produce carbon dioxide and water, with flow field parameters in the nozzle throat and expansion section exhibiting axisymmetric characteristics.

2.3 Plume Impingement Flow Field Results and Analysis

Using the nozzle throat section parameters from the engine internal flow field calculation as inlet boundary conditions and employing the single-step global reaction to describe the afterburning process, three-dimensional flow fields were computed for LOX/kerosene engine plumes impinging on three configurations: no deflector, wedge deflector, and conical deflector. Figures 7(a)-7(c) show the pressure, temperature, and velocity fields for plume impingement on a trough without deflector. After impinging on the trough bottom, the plume gas primarily flows in three directions: along the trough ramp, along the side surfaces, and upward due to vertical impact reflection. Both the side-flowing and upward-reflected gas create high-temperature effects on the environment directly above the trough base.

Figures 8(a)-8(c) present the pressure, temperature, and velocity fields for plume impingement on a trough with a wedge deflector. After directly impacting the wedge apex, the plume gas flows primarily along the deflector-guided ramp, avoiding high-temperature effects on the environment above the trough base. Figures 9(a)-9(c) show the flow fields for plume impingement on a trough with a conical deflector. After impacting the conical deflector apex, the plume flows along the guided direction, preventing upward reflection caused by vertical impact on the trough bottom. However, because the conical deflector guides some gas along the trough side surfaces, it only partially mitigates the high-temperature effects, leaving room for improvement.

Pressure distributions on the trough bottom, wedge deflector surface, and conical deflector surface are shown in [Figure 10: see original paper]. Compared with the no-deflector trough bottom, the maximum pressure on the wedge and conical deflectors increased by 10.31% and 33.81%, respectively. This occurs because: (1) the deflector reduces the distance between the nozzle exit and impact surface, and (2) the deflector geometry reduces the area of maximum force application, with the conical deflector having a smaller maximum force area than the wedge deflector.

Parameter distributions along the centerline from nozzle throat to impact plane are shown in [Figure 11: see original paper]. Figures 11(a)-11(c) display pressure, velocity, and temperature distributions. When the plume impacts the trough or deflector surface, velocity drops abruptly to zero, causing sudden pressure and temperature increases. Because the deflectors effectively guide plume flow, the maximum temperature on the wedge and conical deflectors decreased by 4.04% and 8.95%, respectively, compared with the no-deflector case.

Conclusions

This study yields the following conclusions:

(1) When a LOX/kerosene engine exhaust plume vertically impacts a trough without deflector, the gas flows simultaneously along the trough ramp and side surfaces. Both the side-flowing and upward-reflected gas create high-temperature effects on the environment directly above the trough base.

(2) When impinging on a trough with a wedge deflector, the plume gas flows along the deflector and trough ramp, eliminating high-temperature effects on the environment above the trough base from side-flowing and upward-reflected gas.

(3) When impinging on a trough with a conical deflector, the plume gas flows along the deflector, ramp, and side surfaces, only partially mitigating high-temperature effects on the environment above the trough base. Improvement is still needed to prevent gas flow along the side surfaces.

(4) Compared with the no-deflector case, the wedge and conical deflectors exhibit maximum pressure increases of 10.31% and 33.81%, respectively,

due to reduced nozzle-to-surface distance and smaller maximum force areas. However, because the deflectors effectively guide plume flow, the maximum temperatures on the wedge and conical deflectors decreased by 4.04% and 8.95%, respectively.

References

[1] NIE Wansheng, YANG Junhui, HE Haobo, et al. The IR Radiation Characteristic of Exhaust Plume of the Liquid Rocket Engine[J]. Journal of National University of Defense Technology, 2005, 27(5): 91-94. (in Chinese)

[2] ZHOU Fan, JIANG Yi, HAO Jiguang. Exploring on Cooling Effect of Water Injection on Rocket Motor Exhaust[J]. Journal of Propulsion Technology, 2012, 33(2): 249-252. (in Chinese)

[3] MA Yanli, JIANG Yi, HAO Jiguang, et al. Study on the Impact Effect of Solid Rocket Motor Exhaust Plume to the Launching Platform[J]. Journal of Solid Rocket Technology, 2010, 33(4): 373-376.

[4] E. Mayer, R. Prickett. Rocket Plume Impingement Heat Transfer on Plane Surfaces[J]. AIAA Journal, 1987, 24(4).

[5] David R Gonzalez, Paul Wallman, Matthew Sanford, et al. Characterization of Rocket Plume Fluid Dynamic Environment Using Numerical & Experimental Approaches[J]. AIAA Journal, 2013, 50(3): 527-539.

[6] E. Mayer, R. Prickett. Rocket Plume Impingement Heat Transfer on Plane Surfaces[R]. AIAA 86-1321, 1986.

[7] Seiji Tsutsumi, Susumu Kato, Kato Fukuda, et al. Effect of Deflector Shape on Acoustic Field of Launch Vehicle at Lift-off[C]// Orlando: 47th AIAA Aerospace Sciences Meeting Including The New Horizons Forum and Aerospace Exposition, 2009, AIAA 2009-328.

[8] Michael C Dawson, Freddie Douglas, Peter A. Orlin. ASRM Plume Deflector Analysis Program[C]// Nashville: 17th Aerospace Ground Testing Conference, 1992, AIAA 1992-3868.

[9] Freddie Douglas, Michael C Dawson, Peter A Orlin. ASRM Subscale Plume Deflector Testing[C]// Nashville: 17th Aerospace Ground Testing Conference, 1992, AIAA 1992-3869.

[10] Jeffery A. Housman, Michael F. Barad, Cetin C. Kiris. Space-Time Accuracy Assessment of CFD Simulations for the Launch Environment[C]// Honolulu: 29th AIAA Applied Aerodynamics Conference, 2011, AIAA 2011-3297.

[11] DANIE C A, VINEET A. Computational Plume Modeling of Conceptual ARES Vehicle Stage Tests[C]// Cincinnati: AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, 2007, AIAA 2007-5708.

[12] BRUCE T V, NILI B, OSHIN P, et al. Multiphase Modeling of Water Injection on Flame Deflector[C]// San Diego: 21st AIAA Computational Fluid Dynamics Conference, 2013, AIAA 2012-2592.

[13] J. K. Prasad, R. C. Mehta, A. K. Sreekanth. Impingement of Supersonic Jets on An Axisymmetric Deflector[J]. AIAA JOURNAL, 1994, 32(7): 1535-1538.

[14] T S Wang. Thermo-kinetics characteristics of kerosene combustion[C]// Denver: 34th AIAA Thermophysics conference, 2000, AIAA 2000-2511, 1-11.

[15] NIE Wansheng, FENG Songjiang. Liquid Rocket Engine Combustion Kinetics Model and Numerical Calculation[M]. Beijing: National Defense Industry Press, 2005. (in Chinese)

[16] Alvi F S, Iyer K G. Mean and Unsteady Flowfield Properties of Supersonic Impinging Jets with Lift Plates[C]// Bellevue: 5th AIAA/CEAS Aeroacoustics Conference, 1999, AIAA 1999-1836.

Author: CAI Hong-Hua
Address: P.O. Box 98, No. 1 Bayi Road, Huairou District, Beijing
Postal Code: 101416
Mobile: 15300249295
E-mail: honghuacai@aliyun.com

Note: Figure translations are in progress. See original paper for figures.

Source: ChinaXiv —Machine translation. Verify with original.