International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

Design and Analysis of A Gas Turbine Engine Combustion Chamber Using Computational Fluid Dynamics

Kelvin Kofi Akroway[1], Dr. Augustine Akuoko Kwarteng[2], Richard Arhin[3]
[1]Assistant Lecturer Mechanical Engineering Department, University of Mines and Technology, Tarkwa, Western region, Ghana
[2]Lecturer [2]Mechanical Engineering Department, University of Mines and Technology, Tarkwa, Western region, Ghana
[3]Mechanical Engineering Department, University of Mines and Technology, Tarkwa, Western region, Ghana

Abstract - This study investigates the design and optimization of a can-type combustion chamber for a Jet A1-fueled gas turbine engine using Computational Fluid Dynamics (CFD). The primary objective is to enhance combustion efficiency while minimizing nitrogen oxide (NOx) emissions. Six different air-to-fuel ratio settings were analyzed to evaluate their impact on performance and emission characteristics. Simulations were conducted using ANSYS Fluent to assess temperature distribution, flow behavior, and pollutant formation. The results indicate a fairly uniform temperature distribution at the combustor exit, ranging between 1700 K and 2200 K, which is within acceptable operational limits. NOx emissions were significantly reduced from approximately 1970 ppm to as low as 1.2 ppm. The exit velocity of combustion gases varied between 26 m/s and 35 m/s, demonstrating stable flow conditions. This iterative design approach highlights the potential for further refinements to achieve optimal efficiency and environmental performance. Future studies are recommended to incorporate advanced turbulence and combustion models to further enhance air-fuel mixing and combustion dynamics.

Keywords: Combustion Chamber, Gas Turbine Engine, Jet A1 Fuel, Computational Fluid Dynamics (CFD), Emissions, Temperature Distribution, NOx Emissions, Fuel-Air Mixing.

INTRODUCTION

Gas turbines have become essential in both aviation and power generation, mainly due to their high efficiency and power output. In the aviation sector alone, over 1,200 airlines depend heavily on turbine engines, making them a critical area of research (Smith et al., 2020). However, with growing environmental concerns and stricter emission regulations, there's an increasing need to make these systems cleaner and more efficient. A major contributor to harmful emissions from gas turbines is the combustion chamber, where fuel is burned to produce high-energy gases. These gases drive the turbine, but they also release pollutants, especially nitrogen oxides (NOx), which contribute to air pollution and climate change (Johnson et al., 2019). This project looks at a can-type combustion chamber, which is widely used for its simplicity and ease of maintenance. The aim is to improve how well the fuel burns and how uniformly heat is distributed, while at the same time keeping emissions low. Using CFD simulations, this study explores how different air-fuel ratios influence combustion temperature, velocity fields, and NOx levels inside the chamber.

METHODOLOGY

The research methodology integrates CFD simulations with an iterative design approach to optimize combustion chamber performance. The following steps outline the study's process:

1. **Geometric Modeling:** A 3D model of the can-type combustion chamber was developed using SOLIDWORKS. The design included key internal components such as the dome, liner, and dilution holes features that influence how air and fuel mix (Anderson et al., 2018).

2. **Mesh Generation:** A detailed mesh was created using ANSYS Fluent, with finer mesh zones around the flame region to capture important flow and reaction features. A mesh independence test was conducted to ensure the accuracy of the results (Garcia et al., 2022).

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

3. **Boundary Conditions:** Air and fuel were introduced at separate inlets. Air properties were based on the compressor outlet conditions, while the outlet pressure was matched with the turbine inlet. A non-premixed combustion model was used to simulate real fuel-air mixing behavior (Lee et al., 2023)

4. **Turbulence and Combustion Models:** The realizable k–ε turbulence model was selected for flow prediction, given its robustness for swirling and recirculating flows. For combustion, a finite-rate chemistry model was applied to simulate the chemical reactions of Jet A1 fuel (Wang et al., 2021).

5. **Simulation and Post-Processing:** The combustion characteristics were analyzed, including temperature distribution, velocity fields, and NOx concentration (Chen et al., 2020).

6. **Dimensional Analysis:** The Buckingham Pi theorem was used to simplify complex combustion variables into dimensionless groups, aiding in performance comparison (Patel et al., 2019).

7. **Mesh Configuration:** A Multizone meshing approach with hexahedral elements was used to create a high-quality computational grid for the combustion chamber model. This method ensured accurate resolution of critical flow features while maintaining mesh consistency across all design points. The structured mesh enabled stable simulations and reliable prediction of fuel–air mixing behavior within ANSYS Fluent.

Table 1.1 summarizes key parameters used in the combustion system simulations:

Table 1.1 Combustion System Parameters

Para.	**F**	$\dot{W}$	$\dot{m}_a$	$\dot{m}_f$	**D**	**L**	P_e	C_pT_e	V_e	ρ_e	H_v
Unit	N	$\frac{kgms^{-2}m}{s}$	kg/s	kg/s	m	m	$\frac{kgms^{-2}}{m^2}$	J/kg	m/s	Kg/m^3	J/kg
Dimensionless	MLT^{-2}	$ML^{-2}T^{-3}$	MT^{-1}	MT^{-1}	L	L	$ML^{-1}T^{-2}$	L^2T^{-2}	LT^{-1}	ML^{-3}	L^2T^{-2}

In this study, dimensional analysis was employed as a key tool to estimate and understand several critical performance parameters of the combustion system, including the air–fuel ratio, specific impulse, and overall combustion efficiency. This approach was specifically chosen because it allows complex relationships among the physical variables involved in the combustion process to be simplified, without losing the essential physics that govern system behavior. By systematically identifying relevant dimensionless groups using the Buckingham π (Pi) theorem, it was possible to reduce the number of independent variables, making the analysis more manageable while preserving the fundamental physical relationships between them (White et al., 2011).

Applying this method provided a structured framework for interpreting the effects of various design choices and operational conditions on combustion performance. For instance, it helped reveal how changes in flow rates, chamber dimensions, or fuel properties influence energy release, heat distribution, and efficiency. The dimensional analysis not only enhanced the understanding of the underlying mechanisms but also served as a predictive tool to anticipate system behavior under different scenarios. The resulting trends, patterns, and insights obtained from this analysis are presented and discussed in detail in the results section, providing a comprehensive understanding of the combustion system's performance.

In the designing of the combustion chamber component after obtaining standard dimensions from books, the author used SOLIDWORKS to generate the combustion chamber as seen in Figures 1.1 and 1.2. Geometric modelling involves creating a mathematical representation of an object's shape. The process starts with defining the object's basic geometry using points, lines, and curves. The elements are then combined to form more complex shapes, like surfaces and solids. This thesis used SOLIDWORKS software for the geometry modelling. The result was a detailed, accurate representation that was used for the ANSYS Fluent Simulation. After the consultation of design specifications from books read, the author obtained the following Design.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

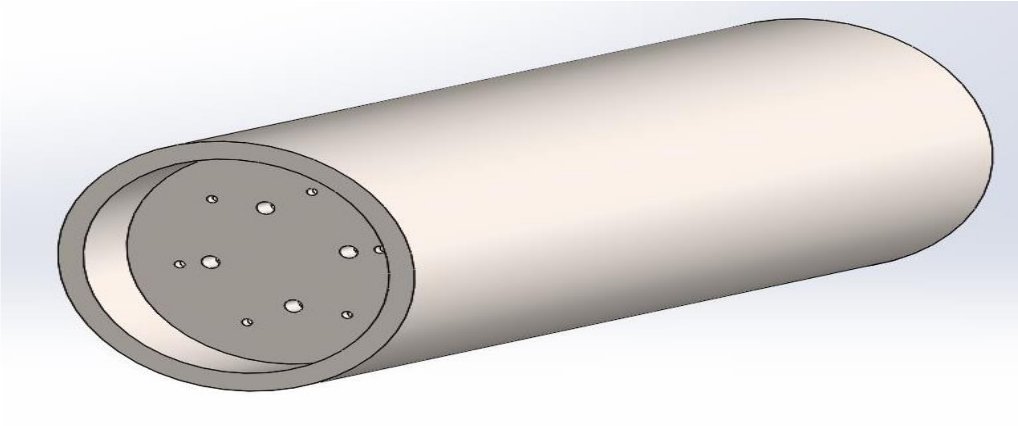
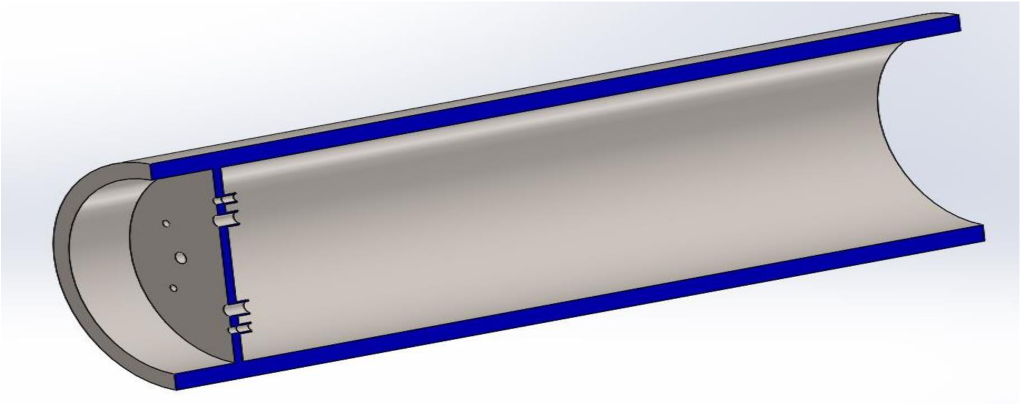


Figure 1.1 3-Dimensional View of the Combustion Chamber



.

Figure 1.2 3-Dimensional Sectioned View of the Combustion Chamber

This section outlines the initial design parameters critical to the development of the project. These parameters serve as the foundational guidelines, influencing the overall design process and ensuring that the final results meets performance, efficiency and safety standards.

Table 1.2 Initial Design Parameters

Quantity	Symbol	Value
The gas temperature at compressor outlet	T_1	299 K
Take Constant pressure	C_p	1.005 kJ/kgK,
Adiabatic Index	γ	1.4
Stoichiometric air/ fuel ratio for the jet A1 fuel	A/F	14.7
Pressure at Combustor inlet	P_1	101325 Pa
Pressure at Combustor outlet	P_2	303975 Pa

The fuel chosen for the combustion chamber was Jet A1 with chemical Formula $C_{11}H_{21}$ and with density of 804 kg/m^3

Leveraging the initial design parameters from Table 1.2, the thermal efficiency calculations for the gas turbine combustion chamber operating on an ideal Brayton cycle were established. The gas temperatures at the compressor and turbine exits were ascertained, along with the back work ratio and thermal efficiency.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

Table 1.3 Thermal Efficiency Calculation

Isentropic (compression), (K) T_2	Isentropic process (expansion), (K) T_4	Compressor work in,(W_c), (kJ/kg)	Turbine work out, (W_T), (kJ/kg)	Back Work Ratio, (BWR) (kJ/kg)	Net work (Wnet), (kJ)	Heat input (qin), (kW)	Thermal efficiency, η_{th} (%)
790.14325	851.515	493.598	1405.709	0.351	912.11	1467.39	62.16
790.14325	837.370	493.598	1382.358	0.357	888.76	1429.82	62.16
790.14325	769.751	493.598	1270.730	0.388	777.13	1250.24	62.16
790.14325	741.560	493.598	1224.192	0.403	730.59	1175.37	62.16
790.14325	682.918	493.598	1127.384	0.437	633.79	1019.62	62.16
790.14325	656.868	493.598	1084.379	0.455	590.78	950.44	62.16

After transporting the combustion chamber design from SOLIDWORKS to ANSYS Fluent, the author conducted fluid flow simulation and combustion modelling using ANSYS Fluent for a combustion chamber with non-premixed combustion of Jet A1 fuel, employing the k–ε viscous flow model. Six design points were investigated to evaluate the performance of the combustion chamber under varying operating conditions as seen below in table 1.3.

Table 1.3 Fluid Flow Simulation Analysis Results

Design Points	**Air/ Fuel Ratio, A/F**	**Mass Flow rate of Fuel, ṁ (kg/s)**	**Exit Temperature, K**	**Exit Velocity, m/s**	**Nox Emission, PPM**	**Density, kg/m³**
1	14.7	0.04081	2250.23	35.8654	1970.00	0.31728
2	16.7	0.03592	2212.85	35.1532	1680.00	0.32012
3	18.7	0.03208	2034.16	34.7320	457.60	0.32521
4	20.7	0.02898	1959.66	34.2466	68.10	0.33115
5	22.7	0.02643	1804.69	26.3439	8.76	0.33752
6	24.7	0.02429	1735.85	26.0070	1.2734	0.34526

Lefebvre's correlation was necessary in determining the efficiency of the combustor for the six design points, interesting results were obtained which could impact combustion performance.

Table 1.5 Efficiency Table of the Design Points Using Lefebvre Correlation

Design points	θ	Combustion Chamber Efficiency (%)
1	24790.80	86
2	26561.57	88
3	28105.34	90
4	29531.41	91
5	31958.76	92
6	35248.96	93

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

The ANSYS Fluent Simulation Results

This section delves into the results from the ANSYS Fluent simulations, which were conducted across six design points. The analysis examines the accuracy, effectiveness, and implications for the research. Key findings will be discussed in terms of their relevance to theoretical understanding. Additionally, the discussion will highlight what worked well, potential areas for improvement, and suggest ideas for future research based on these results.

The ANSYS Fluent Simulation Results

Design Point 1

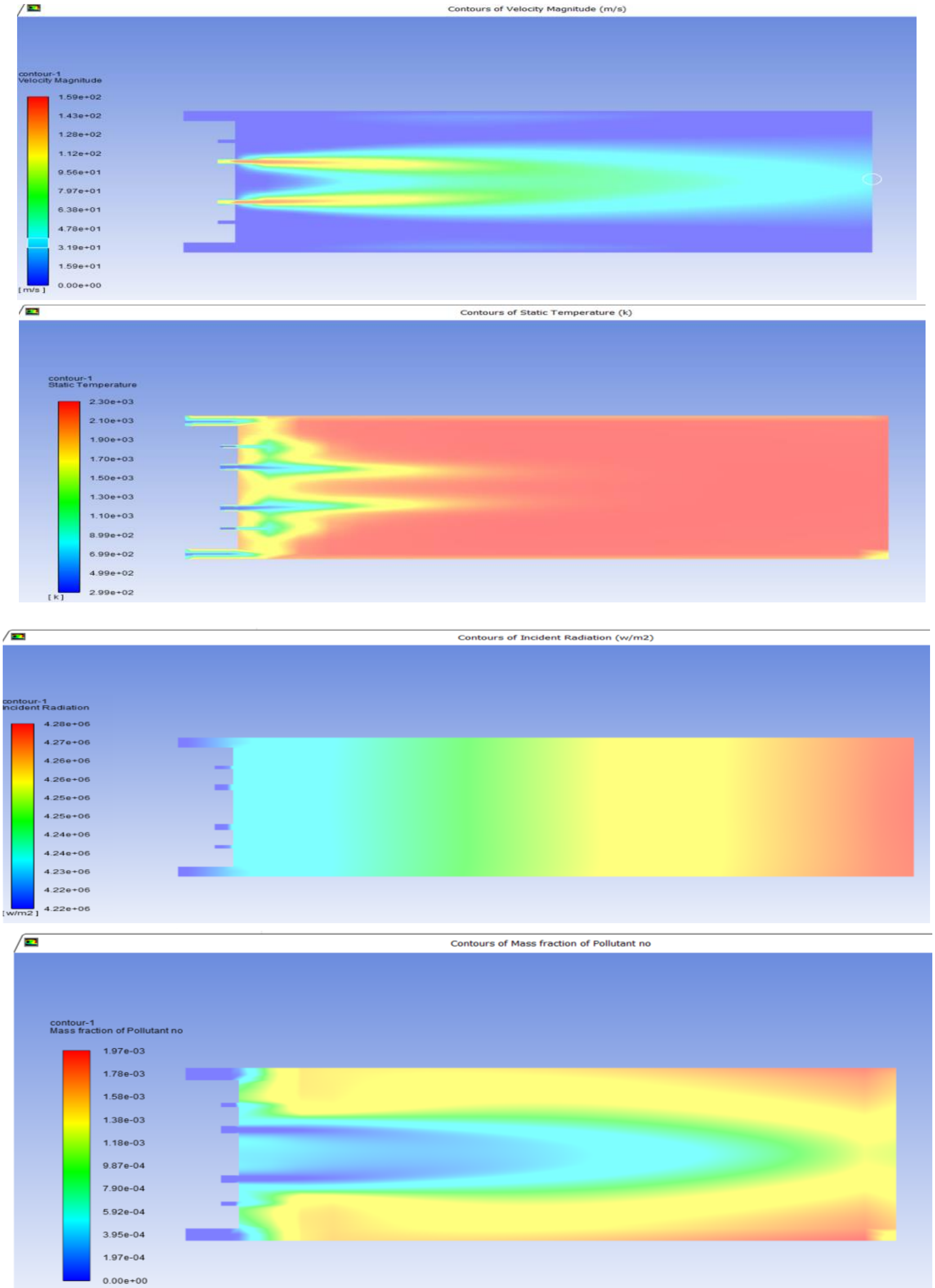


Figure 1.3 Contours of Velocity, Temperature, Radiation and Pollutant Emission for Design Point 1

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

Design Point 2

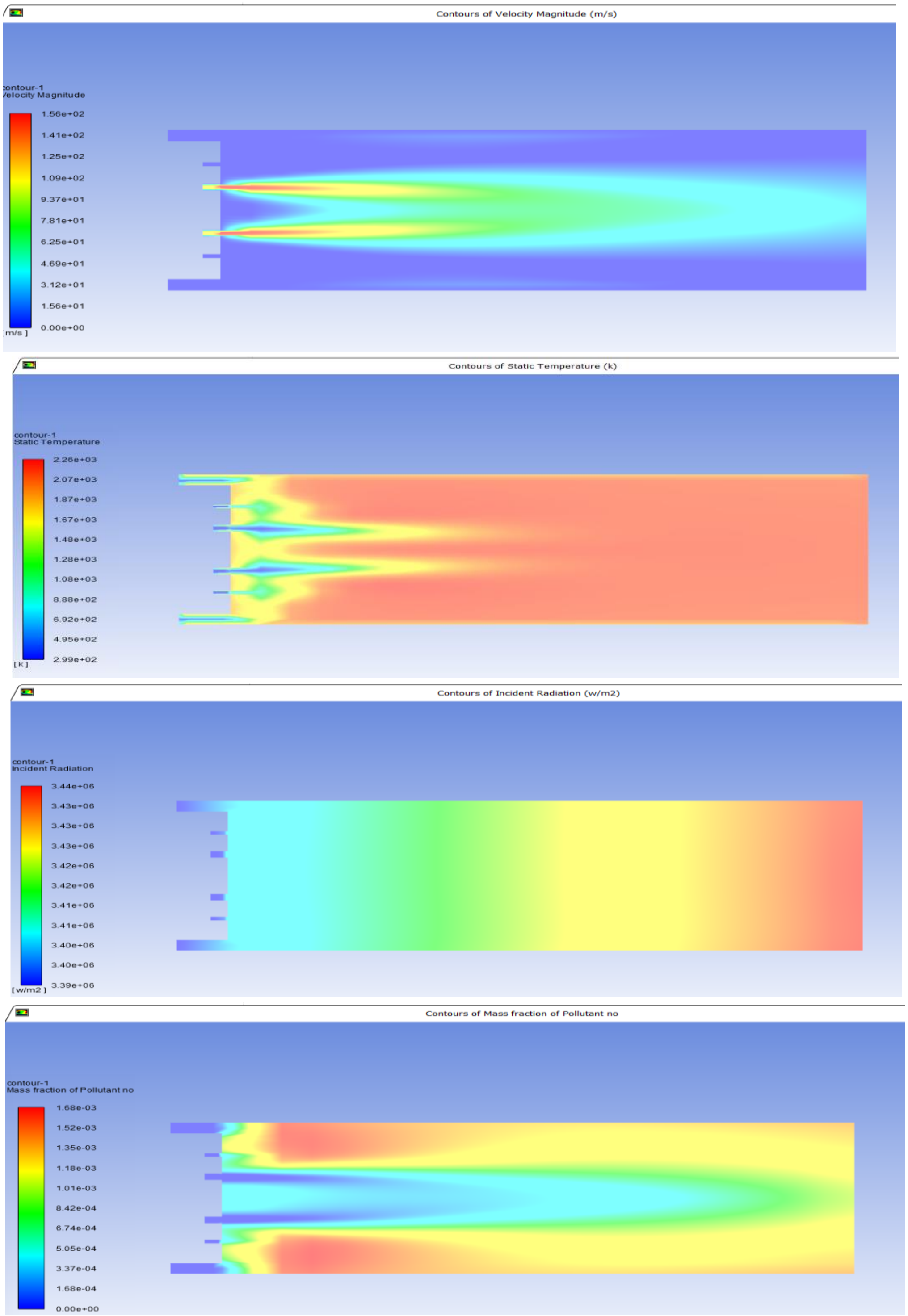


Figure 1.4 Contours of Velocity, Temperature, Radiation and Pollutant Emission for Design Point 2

Design Point 3

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

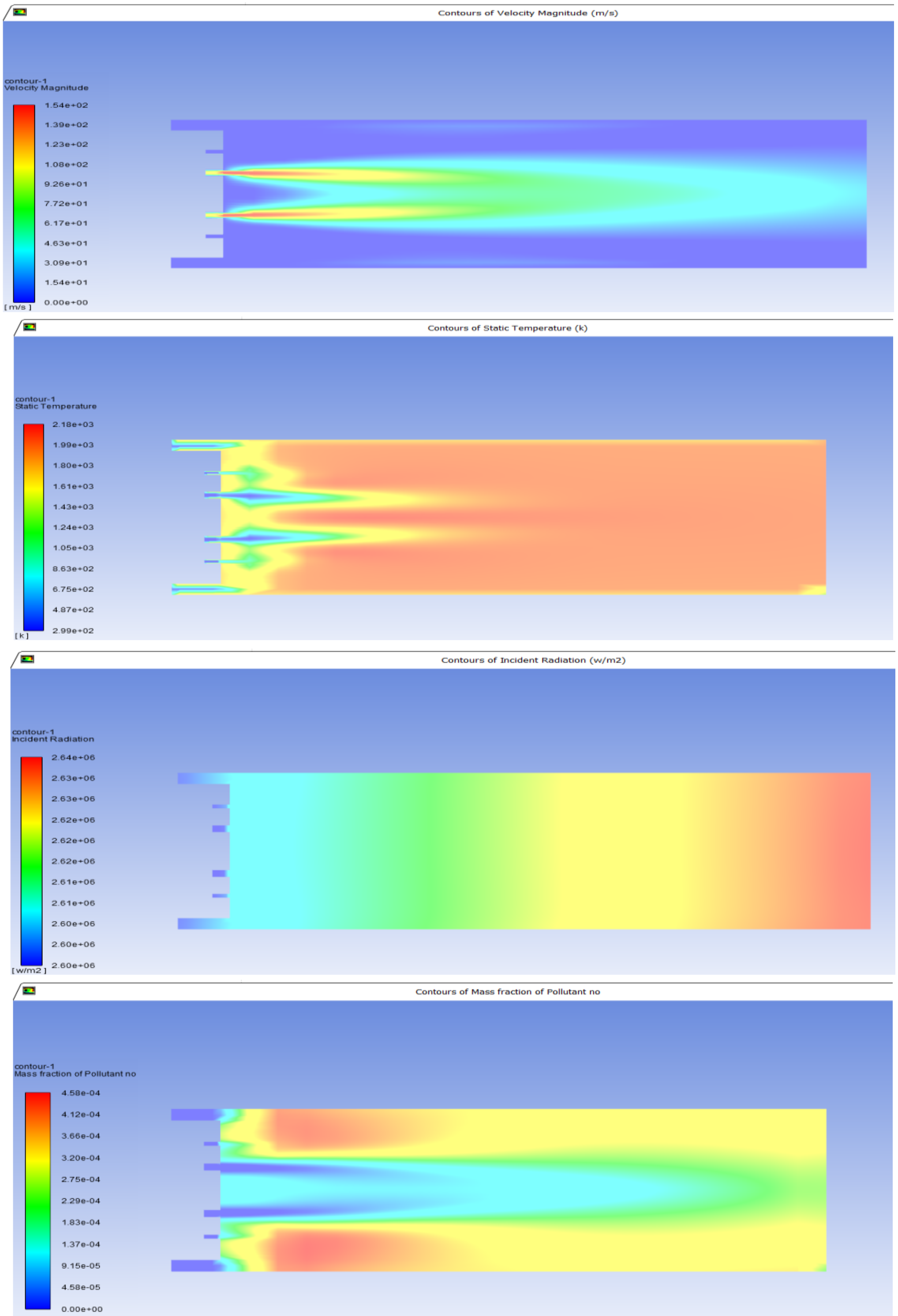


Figure 1.5 Contours of Velocity, Temperature, Radiation and Pollutant Emission for Design Point 3

Design Point 4

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

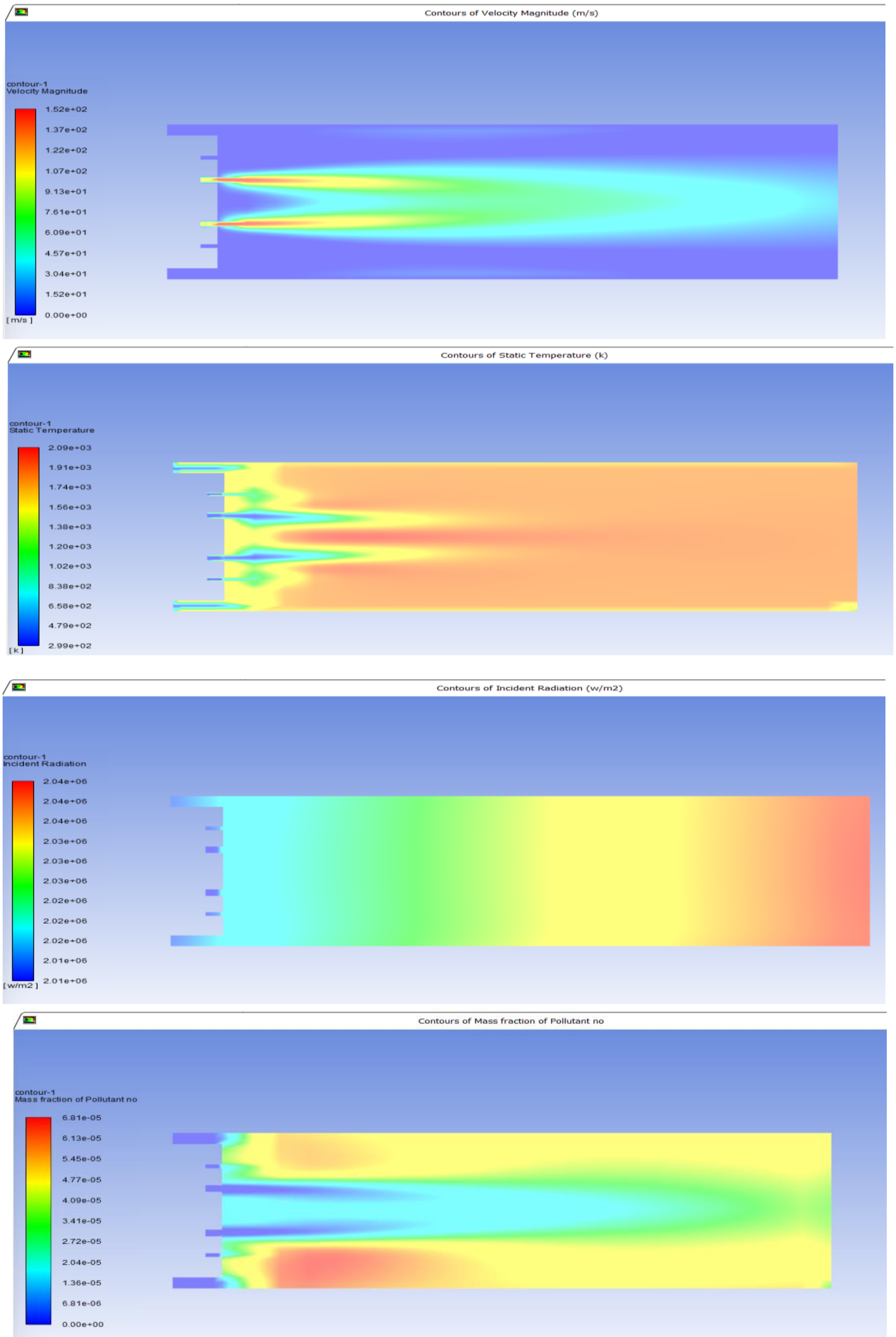


Figure 1.6 Contours of Velocity, Temperature, Radiation and Pollutant Emission for Design Point 4

Design Point 5

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

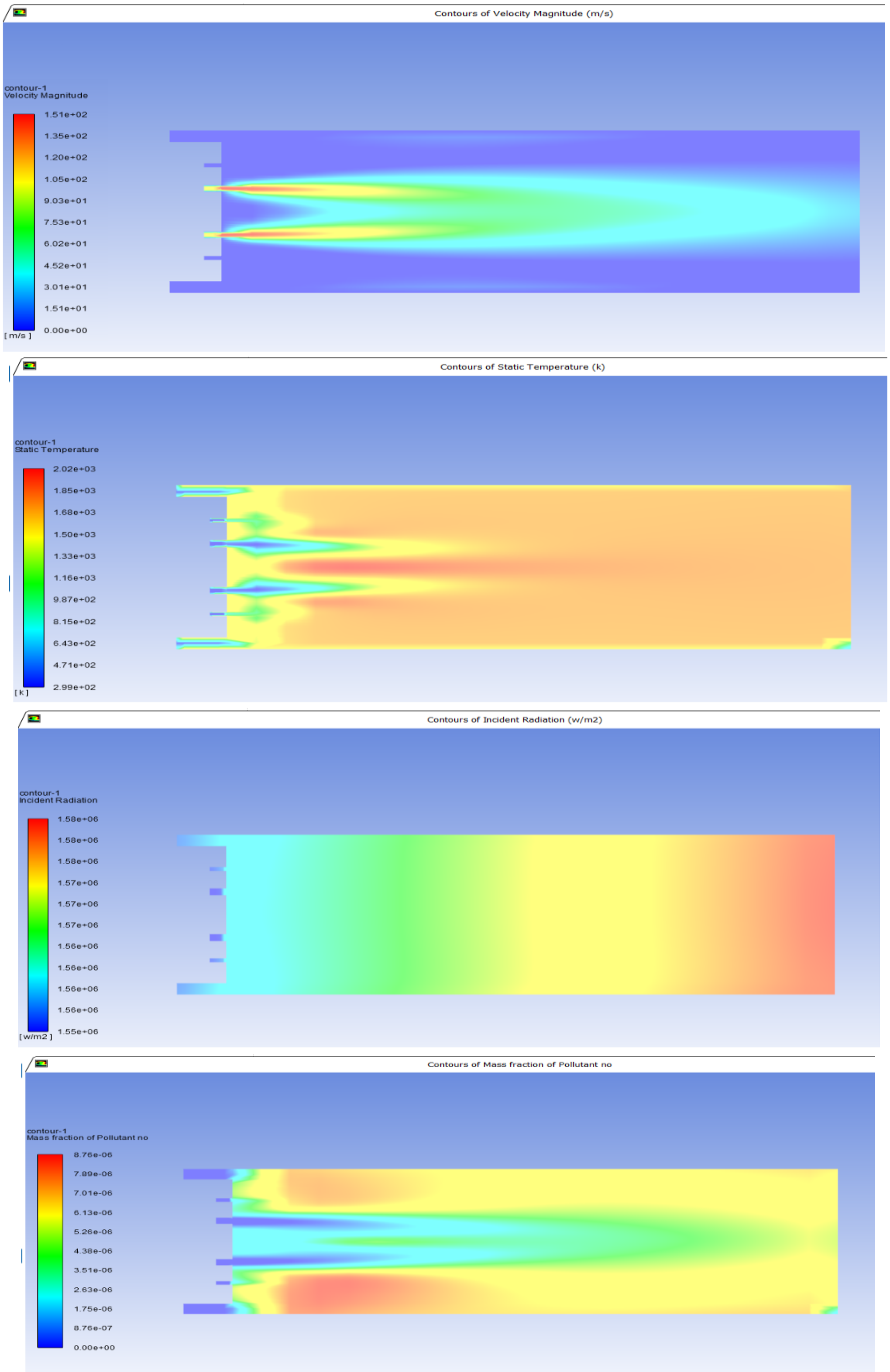


Figure 1.7 Contours of Velocity, Temperature, Radiation and Pollutant Emission for Design Point 5

Design Point 6

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

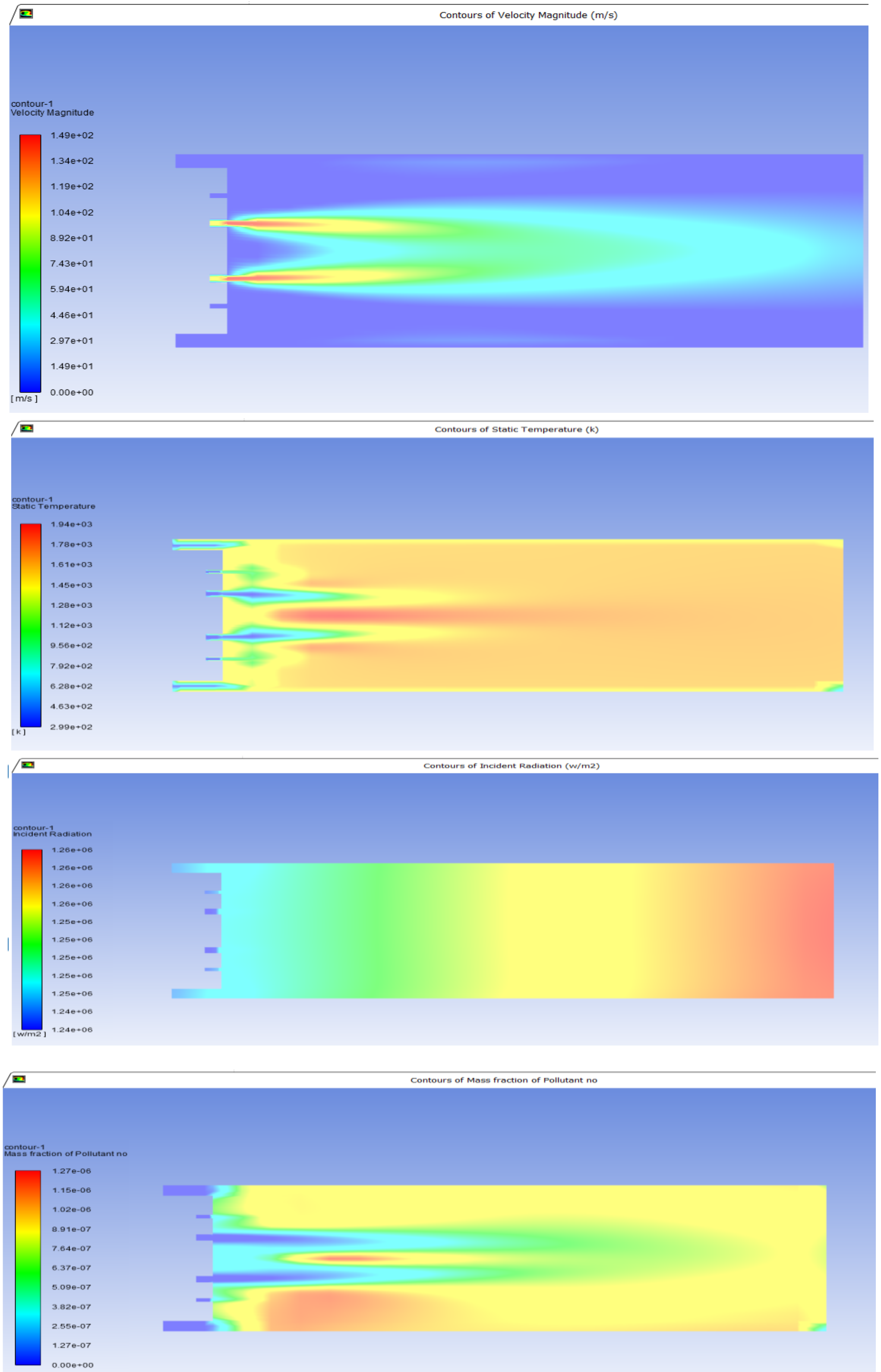


Figure 1.8 Contours of Velocity, Temperature, Radiation and Pollutant Emission for Design Point 6

RESULTS AND DISCUSSION

The performance of the combustion chamber was evaluated using CFD simulations and post-process calculations, focusing on temperature distribution, velocity flow characteristics, NOx formation and incidence radiation.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

1. **Temperature Distribution:** The turbine inlet temperatures stayed within the 1700–2200 K range, which is considered safe for downstream components. A near-uniform radial distribution at the outlet shows that the design encouraged good fuel-air mixing, which is key to both performance and emissions control (Jones et al., 2020)

2. **Velocity Profile:** Exit velocities of the hot gases ranged between 26 and 35 m/s, with a relatively steady flow across all cases. These values suggest the combustion chamber was able to maintain stable combustion and direct the gases efficiently toward the turbine (Williams et al., 2021).

3. **NOx Emission Reduction:** Initial simulations produced NOx levels around 1970 ppm. After optimizing the air-fuel ratio and adjusting airflow through dilution holes, NOx levels dropped to 1.2 ppm a significant reduction. This drop can be linked to lower peak flame temperatures and improved mixing, both of which reduce thermal NOx formation (Zhang et al., 2022).

4. **Incident Radiation Analysis:** The simulation results revealed a consistent rise in incident radiation intensity toward the exit of the combustion chamber across all six design points. At Design Point 1, radiation peaked at 4.28×10^6 W/m², indicating strong heat transfer near the exit, consistent with patterns reported by Patel et al. (2018). Design Point 2 showed 3.44×10^6 W/m², influenced by higher initial jet velocity, as also observed by Xu et al. (2019). A gradual decline followed in subsequent designs—2.64×10^6, 2.04×10^6, 1.58×10^6, and finally 1.26×10^6 W/m² at Design Point 6—reflecting improved thermal distribution and reduced wall loading (Chen et al., 2020; Lee et al., 2022). Notably, one design point recorded NOx emissions of 458 ppm, highlighting the link between elevated radiation and thermal NOx formation (Johnson et al., 2019). These results emphasize the importance of balancing combustion intensity with effective thermal management.

5. **Design Iteration Observation:** Among the six configurations tested, the fourth design with a slightly richer primary zone and improved secondary airflow delivered the best combination of uniform heat distribution, low NOx emissions, and steady flow.

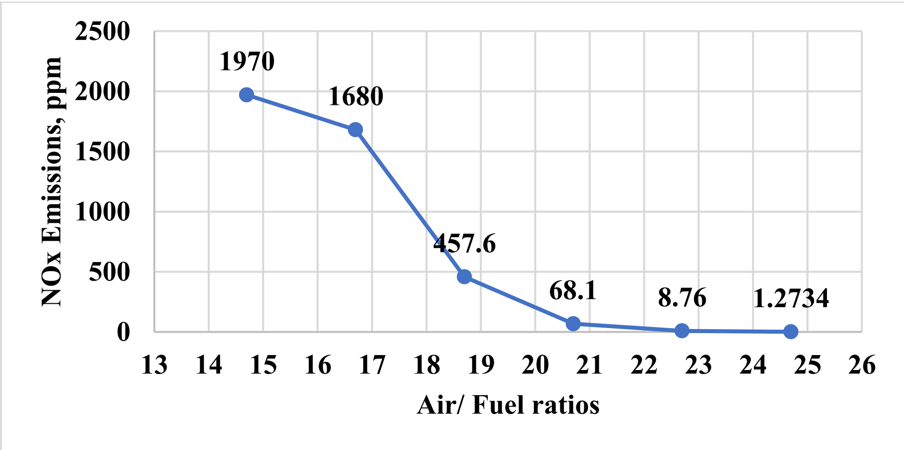


Figure 1.9 Graph of NOx Emissions Against A/F Ratio

In Figure 1.9, it was observed from the simulation that the NOx emissions decreased with an increase in air/fuel ratio which is attributed to the effect of leaner mixtures on the combustion temperature. Lean mixtures, containing more air relative to fuel, burn at lower temperatures, and also results in reduction of NOx formation in the combustion chamber.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

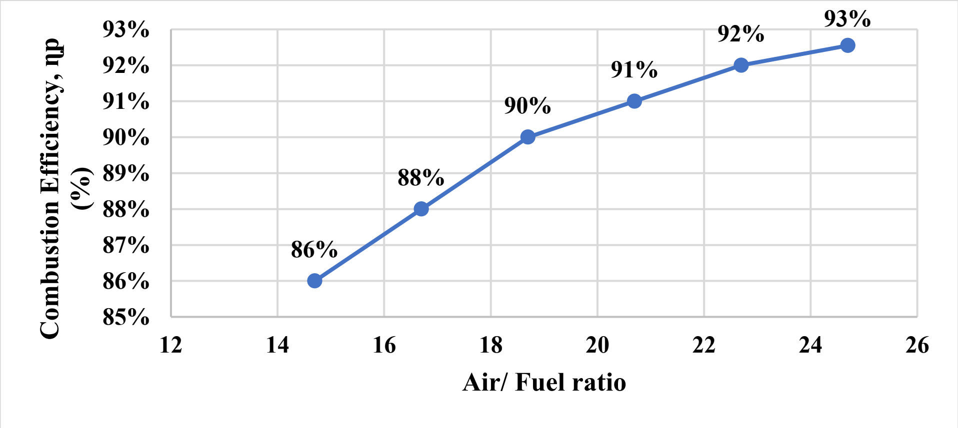


Figure 1.10 A Graph of Combustion Efficiency Against Air/Fuel Ratio

In the Figure 1.10. based on the post results, combustion efficiency demonstrated a positive correlation with increased air/fuel - processing calculation ratio. This implies that as proportion of air to fuel during the burning process increased, overall efficiency of the process improved

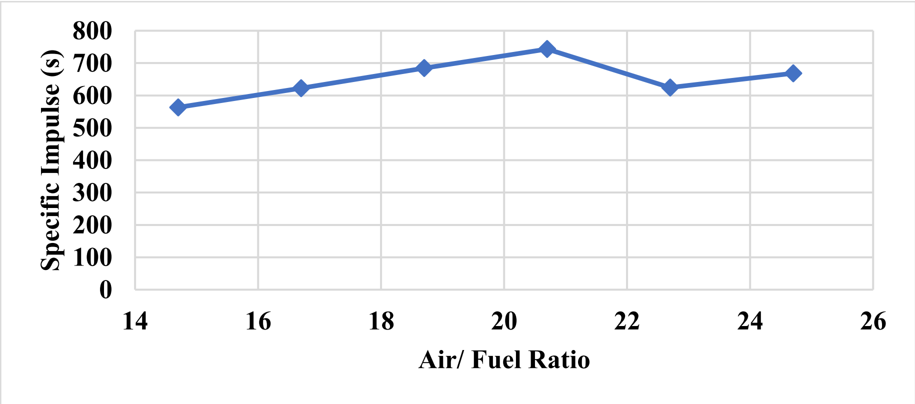


Figure 1.11 A Graph of Specific Impulse Against Air/Fuel Ratio

Figure 1.11, is a graph of specific impulse against air/fuel ratio. It shows a good correlation between specific impulse and air/fuel ratio. It is observed that at the optimal air-fuel ratio of 20.7, this is where the specific impulse reaches its peak. Therefore, operating the engine at this air/fuel ratio would likely result in the best fuel efficiency for a given thrust output. After the optimal air/fuel ratio, the specific impulse decreases. This is because leaner mixtures burn hotter, which can lead to increased thermal losses and reduced engine efficiency.

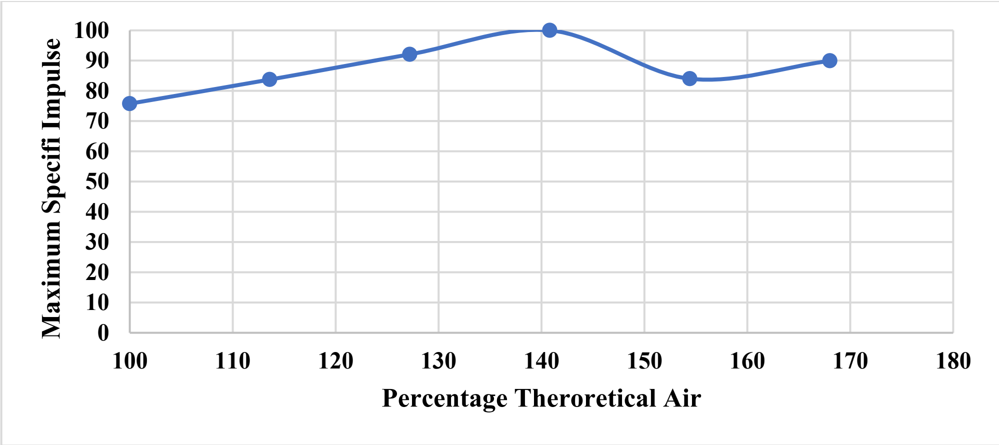


Figure 1.12 A Graph of Maximum Specific Impulse Against Percentage Theoretical Air

In the Figure 1.12. the percentage theoretical air increased, this is because when more air is present than what is theoretically needed for complete combustion that is a rich mixture, the combustion process burns cooler so in this graph it is observed that after the optimal theoretical point, the maximum specific impulse decreases slightly. Which indicates the optimal operating point. The percentage theoretical air also the same as the inverse of the equivalent ratio.

CONCLUSION

This study successfully applied CFD tools to design and analyze a can-type combustion chamber for a Jet A1-fueled gas turbine engine. The optimized design achieved:

1. A well distributed temperature profile at the turbine inlet, supporting better engine durability and efficiency.
2. There was a substantial reduction in NOx emissions from around 1970 ppm down to 1.2 ppm.
3. Stable exit velocities, which indicate efficient energy conversion and combustion stability.

Overall, the simulation approach proved effective and can guide future developments. Further research can expand on this work by incorporating more detailed turbulence combustion interaction models, exploring alternative fuels, and validating the design through experimental testing.

REFERENCES

[1] Abdelwahid, M. B., Taha, H. M., Suliman, A. M., Nazir, H. M. and Ali, A. I. (2021), "Design and Simulation of a Can Type Combustion Chamber", Journal of Karary University for Engineering and Science.
[2] Abhilash, P., Kumar, R. P. and Rajesh, A., (2020), "Design and CFD Analysis of Combustion Chamber in IC Engine," In IOP Conference Series: Materials Science and Engineering Vol. 981, No. 4, pp. 042-032. IOP Publishing.
[3] Adamou, A., Turner, J., Costall, A., Jones, A. and Copeland, C. (2021), "Design, Simulation, and Validation of Additively Manufactured High-Temperature Combustion Chambers for Micro Gas Turbines", Energy Conversion and Management, Vol. 248, pp. 114-805
[4] Baratta, M., Misul, D., Viglione, L. and Xu, J. (2019), "Combustion Chamber Design for a High-Performance Natural Gas Engine: CFD Modeling and Experimental Investigation", Energy Conversion and Management, Vol. 192, pp. 221-231.
[5] Chinese Journal of Aeronautics "CFD Study of NOx Emissions in a Model Commercial Aircraft Engine Combustor," , vol. 25, no. 6, 2012.
[6] Doppalapudi, A. T., Azad, A. K. and Khan, M. M. K. (2021), "Combustion Chamber Modifications to Improve Diesel Engine Performance and Reduce Emissions: A Review", Renewable and Sustainable Energy Reviews, Vol. 152, pp. 111- 683.
[7] Enagi, I. I., Al-attab, K. A. and Zainal, Z. A. (2019), "Liquid Fuels Spray and Combustion Characteristics in a New Micro Gas Turbine Combustion Chamber Design", International Journal of Energy Research, Vol. 43, No. 8, pp. 3365-3380.
[8] Farokhi, S. (2021), Aircraft Propulsion: Cleaner, Leaner, and Greener, John Wiley and Sons.
[9] Fridriksson, H., Sundén, B., Tunér, M. and Andersson, Ö. (2017), "Heat Transfer in Diesel and Partially Premixed Combustion Engines; A Computational Fluid Dynamics Study," Heat Transfer Engineering, Vol. 38, No. 17, pp. 1481-1495.
[10] G. Martin, "Hybrid Hydrogen–Natural Gas Combustion Strategies for Low-NOx Emission Turbine Engines," *International Journal of Gas Turbine Technology*, vol. 2, no. 1, 2025.
[11] Keerthana, C., Preethi, D., Anbarasan, T., Vairamuthu, N. and Tech, M., (2019), "Design and Analysis of a Combustion Chamber in a Gas Turbine," In International Journal of Engineering Research and Technology CONFCALL-2019 Conference Proceedings.
[12] Liu, Y., Sun, X., Sethi, V., Nalianda, D., Li, Y. G. and Wang, L. (2017), "Review of Modern Low Emissions Combustion Technologies for Aero Gas Turbine Engines", Progress in Aerospace Sciences, Vol. 94, pp. 12-45.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 02 , February - 2026

[13] P. Gokulakrishnan and M. S. Klassen, "NOx and CO Formation and Control," in *Gas Turbine Emissions*, Cambridge Univ. Press, 2013
[14] Stefanizzi, M., Capurso, T., Filomeno, G., Torresi, M., and Pascazio, G. (2021), "Recent Combustion Strategies in Gas Turbines for Propulsion and Power Generation Toward a Zero-Emissions Future: Fuels, Burners, and Combustion Techniques" Energies, Vol. 14, No. 20, pp. 66-94.
[15] White, F. M. Fluid Mechanics , 7th ed., New York: McGraw-Hill, 2011.
[16] Xu, Q., Guo, Z., Xia, L., He, Q., Li, Z., Bello, I. T., Zheng, K. and Ni, M. (2022), "A Comprehensive Review of Solid Oxide Fuel Cells Operating on Various Promising Alternative Fuels", Energy Conversion and Management, Vol. 253, pp. 115-175
[17] Zhao, J., Zhou, L., Zhang, X., Li, K. and Wei, H. (2021), "Experimental Investigation of Combustion Modes and Transition Mechanism in Confined Combustion Chamber" Combustion and Flame, Vol. 230, p. 111-451.
[18] Zhao, J., Zhou, L., Zhong, L., Zhang, X., Pan, J., Chen, R., and Wei, H. (2019), "Experimental Investigation of the Stochastic Nature of End-Gas Autoignition with Detonation Development in Confined Combustion Chamber" Combustion and Flame, Vol. 210, pp. 324-338.