LEAN BLOW-OUT ANALYSIS FOR THE PERFORMANCE ASSESSMENT OF DIFFERENT BURNER DESIGNS THROUGH A HIGH-FIDELITY CFD APPROACH

G. Lemmi*, S. Castellani*, S. Galeotti*, A. Picchi*, R. Becchi*, A. Andreini*, G. Babazzi, R. Meloni****
gianmarco.lemmi@unifi.it
*Department of Industrial Engineering, University of Florence, Italy
**Baker Hughes, Via F. Matteucci 2, 50127, Florence, Italy

Abstract

Carbon Capture and Storage (CCS) systems play an increasingly essential role in reducing carbon footprint in Gas Turbine (GT) power generation. However, efficient GT-CCS integration requires high Exhaust Gas Recirculation (EGR) rates to boost CO_2 content at the CCS inlet, posing challenges to conventional combustion systems by reducing oxygen levels. Developing innovative technical solutions is critical to extending combustor operability under high EGR rates. This study employs a high-fidelity Computational Fluid Dynamics (CFD) approach to conduct a comprehensive Lean Blow-Out (LBO) analysis, identifying burner designs with broader operability limits at high EGR levels. Due to the computational intensity, cost-effective accuracy is crucial. Hence, all simulations use a validated extended FGM turbulent combustion model.

Introduction

In the ongoing global energy transition towards carbon-neutral alternatives, the urgency to curb emissions from existing power plants is paramount [1]. Within the realm of GT power generation, CCS systems may serve as a viable temporary solution for reducing CO_2 emissions [2]. However, their integration with GT systems requires high EGR rates to maximise CO_2 concentration at the CCS inlet, thus lowering oxygen levels and potentially limiting combustion system operability [3]. Overcoming flame instability under high EGR conditions requires innovative technical solutions to broaden operability limits. In this context, CFD has the potential to play a pivotal role in identifying flame stability limits across various EGR levels and burner configurations. However, industry demands pose a dual challenge: the need for a robust turbulent combustion model capable of accurately addressing near-blow-out phenomena, and the requirement for extensive CFD simulations during the design phase to evaluate technical solutions' impact on flame stability under varying EGR levels, emphasising the imperative to manage computational costs.

In this study, an enhanced version of the FGM model, previously validated by the authors, is employed to address both objectives. The primary objective of this work is to conduct a comprehensive LBO analysis through a Large Eddy Simulation (LES) approach, assessing flame stability between different burner design solutions under heavy CO_2-diluted air conditions. The computational setup will be outlined in the subsequent section, followed by a description of the numerical LBO procedure. The Results and Discussion will offer a direct comparison of volume-averaged chamber temperature and flame topology among the different design burners investigated.

Experimental Rig and Numerical Setup

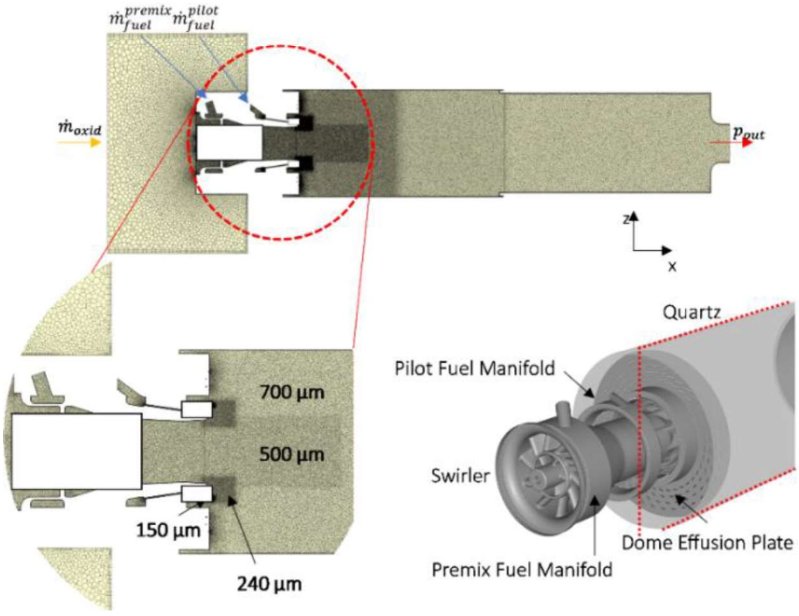

Figure 1. Detailed view of the burner-dome assembly and view of the grid in a longitudinal plane. The employed local sizings are also reported.

Tests are conducted in an optically accessible rig housing an industrial burner supplied by two fuel lines [4]. Fuel is injected by the main premixer through a counter-rotating swirler to ensure uniform mixture delivery into the combustor. In contrast, the pilot line facilitates flame stabilisation by directly delivering gas into the primary zone. The rig operates at atmospheric pressure, with heavily CO_2-diluted air to simulate a real EGR system set to 573 K and Natural Gas with high C2+ content as fuel, featuring an inlet temperature of 291 K, defining test point 95 (TP95) for lean blow-out analysis.

The computational domain encompasses the entire burner geometry (Figure 1), employing a fully unstructured mesh comprising 22 polyhedral cells. Local refinements are applied within the chamber to accurately capture local mixing and flow dynamics. Simulations utilise the commercial pressure-based code ANSYS Fluent® 2022 R1, employing an unsteady high-fidelity LES approach. The WALE subgrid-scale model addresses unresolved turbulence effects [5], while the PISO algorithm governs pressure-velocity coupling with a constant time-step of 1e−5s [6].

Second-order schemes are utilized for both space and time discretization. The outlet maintains constant atmospheric pressure, with wall temperatures prescribed based on thermocouple data. For turbulent combustion modelling, this study utilizes an enhanced version of the FGM approach, incorporating flame stretch and heat loss effects in a pre-calculated table (Γ-table), influencing combustion reactivity directly. For detailed model description and validation, readers are referred to [7].

Numerical Lean-Blow-Out Procedure

The primary aim of this analysis is to determine the burner geometry with the widest flame operability under high CO2 dilution levels in the air. The analysis workflow involves systematically reducing the mass flow rate until encountering lean blow-out [8]. The first step of the analysis consists of an LES of the given geometry under the TP95 operating conditions, here identified as S0. Following stabilization, a 10% reduction in fuel mass flow rate is applied, maintaining the pilot split constant at 30%. Subsequent reductions of 5% in fuel mass flow rate are iteratively implemented until flame extinction occurs. Volume-averaged temperature and volume integral of heat release monitors are employed to identify the onset of LBO conditions. This procedure is repeated for four different burner geometries, denoted as G0, G1, G2, and G3, with specific design characteristics as follows:

- G0: N holes of equal diameter D, with a non-axisymmetric distribution.
- G1: N holes of equal diameter D, with an axisymmetric distribution.
- G2: 2N holes of equal diameter (0.7 D), with an axisymmetric distribution.
- G3: Similar to G1 but with holes tilted in the direction of the swirled flow.

Results and Discussion

Figure 2a displays the area-weighted averaged mixture fraction plots computed on a cross-sectional plane of the flame tube, illustrating the fuel drop steps applied to each geometry. Similar flow times were employed for each geometry during corresponding steps to facilitate visual comparison, as shown in Figure 2a. Small differences arise based on the timing of flame stabilisation occurrence. Each LBO simulation required a total flow time of approximately 0.55s. Considering an estimated flow-through-time (FTT) of about 0.014s, this corresponds to approximately 40 FTTs. The dashed line represents the applied boundary conditions' steps. When reducing the fuel mass flow rate, it takes approximately 2.5 FTTs to uniformly fill the chamber with the new mixture.

Figure 2b illustrates the volume-averaged normalised temperature plots as a function of flow time for each simulated geometry. Trends indicate higher temperatures for G1-G2-G3 configurations compared to G0 under operating conditions with a 15% reduction in fuel mass flow rate (S2). During this phase, G1, G2, and G3 geometries exhibit stable pilot-anchored flames, whereas G0 displays a heavily lifted and elongated flame, as depicted in Figure 3. Conversely, a trend reversal is observed with a 20% reduction in fuel mass flow rate (S3), with the G0 configuration maintaining higher temperatures inside the flame tube compared to other geometries.

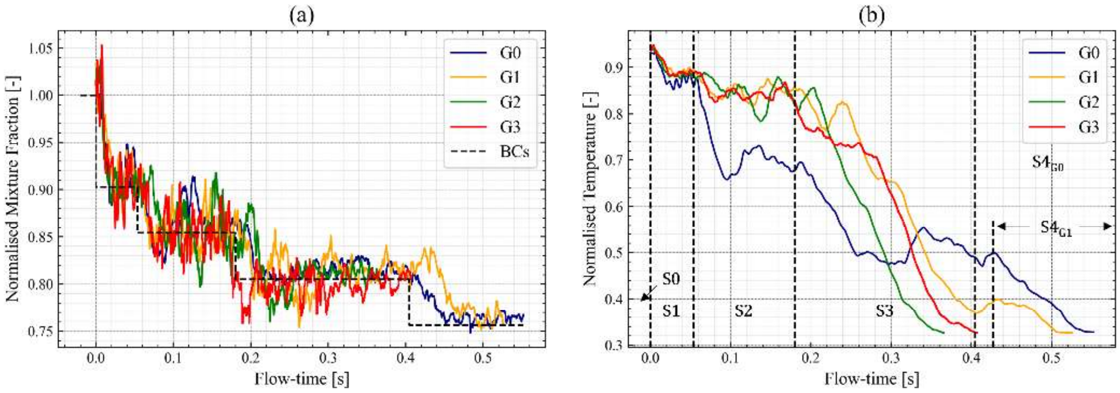

Figure 2. Area-weighted and volume-averaged normalised mixture fraction (a) and temperature (b) plots as a function of the flow time for each simulated geometry.

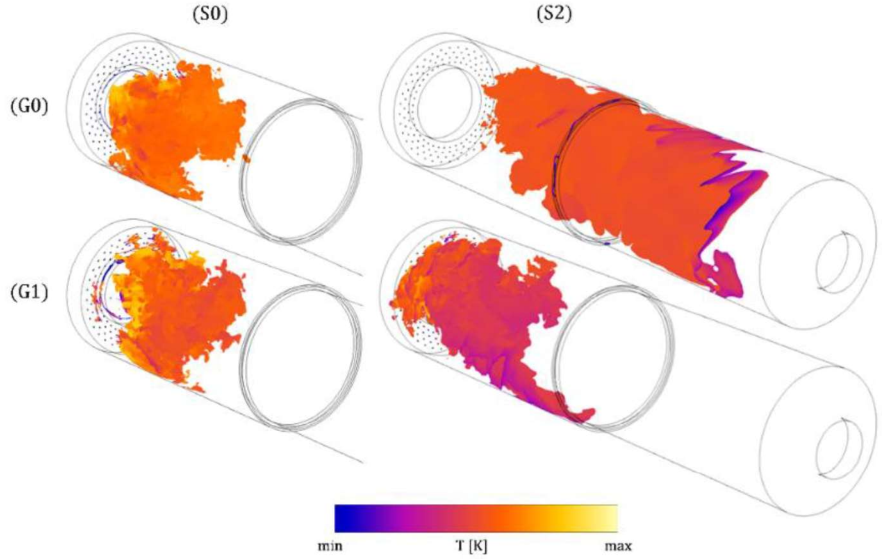

Figure 3. Iso-surface of Progress Variable = 0.5 coloured by temperature for G0 and G1 geometries under 0% (S0) and 15% (S2) fuel mass flow rate reduction.

Under these conditions, the G0 solution sustains stable near-blow-out (BO) conditions, while both G2 and G3 encounter lean blow-out (LBO), with G1 approaching it but not completely experiencing it.

Figure 4 illustrates instantaneous temperature fields on the midplane of the burner for the G0 and G1 design solutions at each fuel reduction step. Both geometries exhibit clear flame length elongation as the fuel mass flow rate decreases. Notably, the G1 geometry demonstrates less susceptibility to this behaviour during the initial two fuel drops (S1-S2), with the flame remaining anchored to the pilots and a high-temperature field observed immediately after the dome. However, a significant decrease in flame stability is observed thereafter, with the flame localised exclusively in the latter portion of the flame tube. In contrast, the G0 geometry displays a more linear trend with the reduction in fuel mass flow rate, with a higher average temperature during the S3 operating condition.

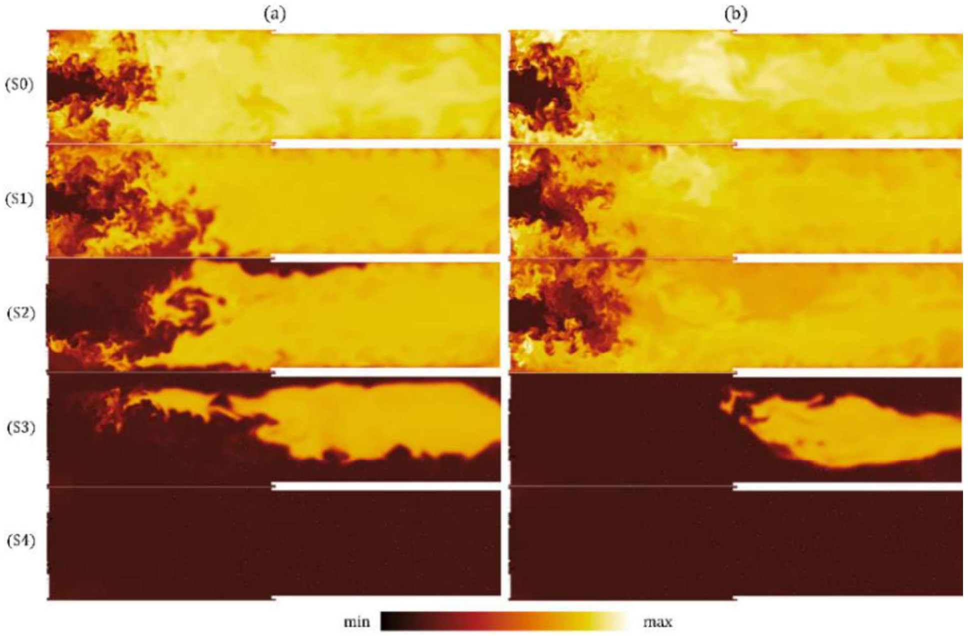

Figure 4: Instantaneous temperature contours on the midplane of the burner for the G0 (a) and G1 (b) design solutions at each fuel reduction step.

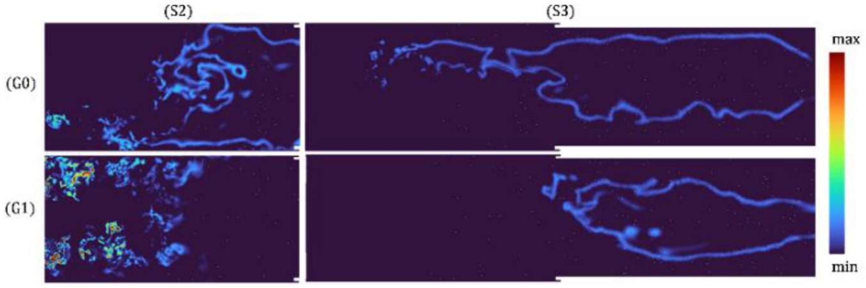

Figure 5: Instantaneous heat-release contours on the midplane of the burner for the G0 and G1 design solutions at steps 2 and 3.

This is further supported by the heat-release contours (Figure 5), indicating that the G0 still maintains a peak of heat release immediately after the pilot jets' region, while the G1 confines it at the end of the flame tube, suggesting an approaching lean blowout (LBO) condition. All solutions ultimately experience lean blowout after a 25% fuel drop. These results closely align with experimental data, with differences ranging from approximately 0.5% to 3% depending on the definition of LBO.

Conclusion

The study delved into the influence of burner geometry on flame stability under highly CO_2-diluted air conditions using LES-based LBO numerical analysis. Through systematic fuel mass flow rate reductions, flame behaviour was analyzed across various burner configurations. Results highlight certain geometries' superior flame stability, showing reduced susceptibility to elongation and LBO. These findings emphasise the pivotal role of burner design in addressing combustion challenges linked to high EGR rates. Further validation against experimental data underscores the effectiveness of the approach, with the enhanced FGM model maintaining consistency while managing computational costs.

Acknowledgement
The present research activity was carried out in the framework of TRANSITION (fuTure hydRogen Assisted gas turbiNeS for effective carbon capTure IntegratiON). This project has received funding from the Europen Union’s Horizon Europe research and Innovation programme under Grant Agreement No 101069665. The Authors gratefully acknowledge this financial support.

Funded by
the European Union

References

[1] A. Pape-Salmon et al., “Low-Impact Renewable Energy Policy in Canada: Strengths, Gaps and a Path Forward”, Pembina Institute, 2003 [Online], Available: https://www.jstor.org/stable/resrep00232.5. [Accessed 08 12 2023].

[2] M. Boot-Handford et al., “Carbon capture and storage update”, *Energy & Environmental Science,* vol. 7, 2014, https://doi.org/10.1039/C3EE42350F.

[3] A. M. ElKady et al., "Exhaust Gas Recirculation in DLN F-Class Gas Turbines for Post-Combustion CO2 Capture", *ASME Turbo Expo 2008,* vol. 3, pp. 847-854, 9-13 June 2008, https://doi.org/10.1115/GT2008-51152

[4] S. Romano et al., "Modeling of Natural Gas Composition Effect on Low NOx Burners Operation in Heavy Duty Gas Turbine", *Journal of Engineering for Gas Turbines and Power,* vol. 143, 2021, https://doi.org/10.1115/1.4049819

[5] F. Ducros et al., “Wall-Adapting Local Eddy-Viscosity Models for Simulations in Complex Geometries”, *Numerical Methods for Fluid Dynamics VI,* 1998.

[6] R. Issa et al., "The computation of compressible and incompressible recirculating flows by a non-iterative implicit scheme", *Journal of Computational Physics,* vol. 62, pp. 66-82, 1986, https://doi.org/10.1016/0021-9991(86)90100-2

[7] G. Lemmi et al., “FGM vs ATF: a comparative LES analysis in predicting the flame characteristics of an industrial lean premixed burner for gas turbine applications”, JFUECO*,* 2024, https://doi.org/10.1016/j.jfueco.2024.100117

[8] P. C. Nassini et al., "Lean blow-out prediction in an industrial gas turbine combustor through a LES-based CFD analysis", *Combustion and Flame,* 2021, https://doi.org/10.1016/j.combustflame.2021.02.037