CFD Simulation and Validation of Hydrodynamic Instabilities Onset in Swirl Combustors

Ali Safa Alsaegh*[1], Fares Amer Hatem [1], Agustin Valera Medina[1], Richard Marsh[1]
[1]College of Physical Sciences and Engineering, Cardiff University, UK

Abstract
The objective of this paper is to employ a numerical approach to model a 150kW tangential swirl burner to investigate the consequence of central air injection on the flashback mechanism. The effects of diffusive air injection on flow field characteristics and how these can affect the lower instability limits by altering the flashback mechanism via CIVB are analysed in both experimental and theoretical approaches. Simulations under isothermal conditions are carried out using both premixed and partially premixed species models to compare the flow field behaviour with and without air injection. The experimental data includes LDA measurements for the same burner geometry. CFD and experimental results demonstrated that using diffusive air affects flashback propensity significantly by expanding the stability region in terms of both equivalence ratio and mass flow rate that lead to greater operability at higher power outputs compared to using only a central body injector. The CFD results were verified and correlated to experimental findings with very good agreement.

Introduction

Most of the global energy generation processes are based on combustion. Thus, the sector has faced real challenges due to requirements to decrease CO2 emissions concurrent with low NOx and other pollutants [1,2]. Emissions regulation and energy efficiency are of high interest due to the associated issues of environmental pollution and energy consumption, respectively. Optimum design and improvement of combustors is a step towards the answer to those considerations. Currently, industrial combustors aim to operate under lean premixed conditions to reduce harmful emissions due to its potential of low NOx production and enhance combustion performance. Moreover, using syngas instead of traditional fuels [3,4] or developing systems for hydrogen blends [5,6] are among the main means of solving this type of challenges. However, the reliability of lean premixed systems is complex as they operate close to the lean stability limit and are more sensitive to combustion instabilities. From a design point of view, it is critical to recognise and predict such instabilities [1,2].

Swirling flows have been used for many decades in many areas of engineering applications. The most significant improvements to the gas turbine combustion system are represented by using swirl combustors due to their flame stabilisation capabilities over a wide range of equivalence ratios thanks to the formation of coherent structures. Their high level of swirl creates vortex breakdown phenomena that lead to the appearance of the central recirculation zone (CRZ) and a vortex with an off-centre core known as the precessing vortex core (PVC) [7]. Vortex breakdown phenomena enhance the high turbulence and shear associated with this kind of burners. These features induce very powerful mixing and therefore improve combustion efficiency by ensuring that all unburned fuel molecules have plenty of oxygen molecules floating around them. The recirculation recycles hot combustion gases to the incoming air hence aiding ignition [8]. Furthermore, wide flame stability limits can be acquired allowing the system to burn of different fuel [8–12]. However, such combustors are frequently subjected to various combustion instabilities upstream the flame, producing phenomena such as flashback propagation from the stable flame position in the combustion chamber towards the premixing zone [2,7]. Consequently, flashback can hinder stable operation, produce a critical damage in the burners, increase the maintenance cost and push up the level of pollutants.

One mechanism of flashback propagation is through the Combustion Induced Vortex Breakdown CIVB, which is considered a fast-acting flashback mechanism that appears in swirl burners because of the formation of the CRZ [13]. This type of flashback receives special attention amongst other flashback mechanisms since it is one of the prevailing mechanisms in swirl combustors and represents an obstacle in developing combustion systems, especially those fed by high flame speed fuels such as high hydrogen blends [13,15]. Another mechanism that increases flashback trends is the appearance of highly turbulent combustion zones caused by fuel properties, flow turbulence and other barely understood phenomena [16,17]. These instabilities occur even when the incombustible mixture velocity is higher than the flame speed. Thus, they can have dramatic consequences when high turbulent flame speed fuels such as those based on highly hydrogenated blends are used [18,19].

Many techniques can effectively tackle CIVB and anchor CRZ downstream the burner nozzle either by doing some geometrical modifications or by raising flow field patterns. Firstly, using diffusive fuel injection to mitigate the flashback mechanism, many researchers found that the strong and coherent axial jet can effectively push downstream the vortex breakdown, consequently eliminating the possibility of CIVB [13,20]. Secondly, using bluff bodies as stabilisers to the jet and swirling flow is another option. However, despite the vitality of this flame stabilisation technique, it cannot totally mitigate flame flashback. Moreover, using central fuel injector

* Corresponding author: AlsaeghAS@cardiff.ac.uk

could lead to increase of NOx emission levels and degrade the degree of mixing. On the other hand, the existence of bluff bodies or central injectors in touch with high temperature flames for long times could lead to material degradation and hence increase in maintenance cost [1,14,21]. Thirdly, injecting air diffusively through the centre of the vortex core to change the defect of negative axial velocity and turbulence characteristics is another option. Using axial air injection instead seems to be more efficient in this context, as it can perform the required flame stabilisation and avoid increasing pollutant levels. Recently axial air injection as flame stabilisation technique has been investigated by many researchers [22,23]. This area of study still needs further investigations, in particular for the optimum amount and position of the axial air injection.

In parallel, Reynolds Averaged Navier-Stokes equations (RANS) are originally used to describe the turbulent flow numerically and then employed to predict the properties of swirl flows. The main point behind the RANS is represented by the Reynolds decomposition, where the instantaneous quantity is decomposed into its time-averaged and fluctuating quantities [24]. Thus, RANS is widely used to model combustion dynamics and swirl flows due to its low computation cost and high reliability [25]. For that aim, standard κ-ϵ equations have been assessed in swirling or recirculating flows with great success [26]. Authors have also employed the κ-ϵ viscous model in their simulation campaign to demonstrate the effects of low swirling flows in a sudden expansion chamber [28].

Some other works have been also conducted with other turbulence models with good sucess [27]. Recently, the large eddy dissipation (LES) turbulence model has been used widely by several researchers to model swirling combustion dynamics and study the interactions between different elements under isothermal and combustion conditions [15,29]. LES resolves the eddies of the turbulence itself. Consequently, it is the best CFD turbulence model to predict the three-dimensional unsteady nature of the actual swirling flow in gas turbine combustors and the flame instabilities and it is more reliable than classic RANS model. However, LES needs much higher transient resolution and is more costly, needing advanced specifications for the computer hardware. LES also requires a long integration time to attain the correct converged solution.

Currently, CFD codes suffer from the lack of proper validation in the combustion area. Vast amoutn of the work performed on swirling flows has been covered through experimental campaigns wich are more complicated and costly. Therefore, in this work experimental and numerical results are presented and analysed to examine the effect of axial air injection on flow field characteristics downstream the burner mouth, especially the turbulence profile and negative velocity defects within the CRZ. The use of central air injection was attempted, with results that show that the technique can be considered promising regarding operation flexibility because it enables switching to another fuel while maintaining full load operation. The rest of this paper is structured as follows: first, the test rig description and equipment used in the experimental work are presented. Next, the numerical approach used is described followed by some details about the ANSYS Fluent 17.2 simulation code settings; finally, discussion and conclusions are provided.

Test Rig and Equipment

A 150kW tangential swirl burner, previously designed at Cardiff University, was used in this work. The system is depicted in Figure 1.

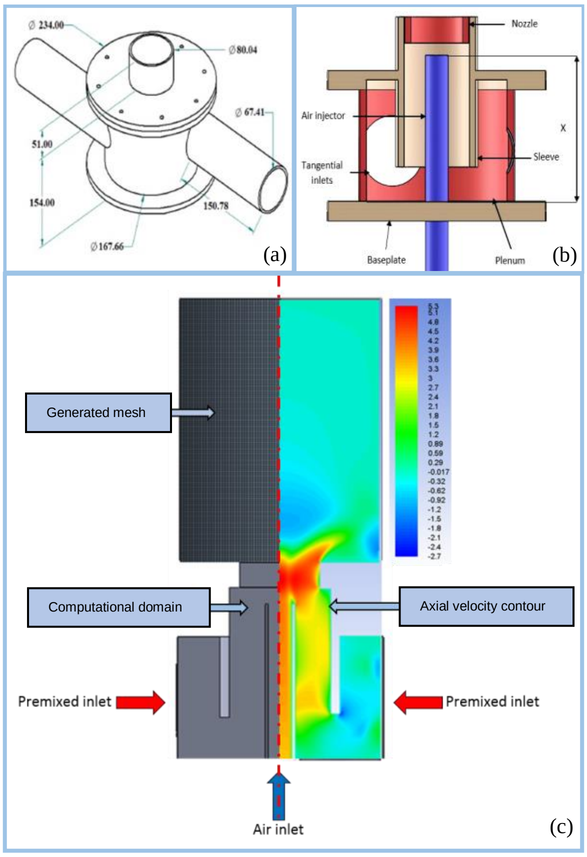

Figure 1: Schematic view of the computational domain, Physical model, mesh, and axial velocity contour (600 LPM with air injection)

Other investigations on swirling flow stability have been undertaken previously using this combustion system [19,30]. The diameter of the tangential inlets can vary using different inserts, while the exit diameter can be changed using different nozzle configurations. Thus, it is possible to have variable geometric swirl numbers from 0.913 up to 3.65. However, in this work, only a 0.913 swirl number has been used with two configurations, the swirl burner with no central air injection and that with the effect of air injection.

The air injector is connected to the burner baseplate; this allows for vertical movement inside the burner plenum to give different positions (X) with respect to the base plate. To start combustion, fuel is injected via the central injector firstly then this supply should be shut down slowly once the tangential premixed fuel valve is opened.

Instantaneous velocity components downstream the burner mouth have been measured with a DANTEC Dynamics Laser Doppler Anemometry (LDA). Velocity measurements have been done at three different levels downstream the burner dump plane. The system is connected to a PC to gather and analyse data via Dantec software.

Numerical Approach and Turbulence Model

One-dimensional LDA measurements of axial velocity and hence turbulence intensity values can provide a good prediction of flow behaviour in both cold and combustion cases. However, three-dimensional characterisation is still required and essential to emphasise the high complexity of the coherent structures of the swirling flow and the interaction between them. ANSYS Fluent 17.2 code has been used to simulate the cold swirl flow in the 150 kW tangential swirl burner. It is a computational fluid dynamics CFD code which involves broad physical modelling abilities and permits simulation of problems that include phenomena such as heat transfer, fluid flow, turbulence, and reactions within the computational models created by users. The premixed and the partial premixed combustion models of Fluent code were used to achieve the goal of this paper. Fluent partial premixed combustion model was used to study the effects of air injection on the swirl flow and to predict the turbulence behind this effect. On the other hand, the premixed combustion model was used to simulate the case without diffusive air.

Turbulent flows which occur in high Reynolds numbers are characterised by large, almost random fluctuations in velocity and pressure in both space and time. These variations result from instabilities that finally are dissipated (into heat) viscosity effects. The popular turbulence models to solve these conditions are the κ-ϵ or the κ-ω models which simplify the dilemma to the solution of two further transport equations and launch an eddy-viscosity (turbulent viscosity) constant to estimate the Reynolds Stresses [31,32]. In this paper, κ-ϵ turbulent models are utilised to illustrate the turbulent flow behaviour in the tangential swirl burner under consideration.

Computational Domain and Mesh Generation

The configuration and mesh generation of the system of interest must be performed in the ANSYS Workbench. Furthermore, the geometry could be imported from (CAD) software Packages. Due to our complex geometry, SolidWorks 2016 has been used to generate the computational domain for the tangential swirl burner under consideration as shown in the left part of figure 1-c. From a turbulence modelling side, the shear layers should be covered by a minimum of ~10 cells normal to it. Below this mesh size the model will not be capable of giving its calibrated performance particularly for free shear flows whose position is not known during the mesh generation [33].

In this stage, care was taken to construct a high-quality mesh to choose high-order discretization schemes and a robust equation solver, and to ensure adequate convergence. A finely structured mesh was used and independence mesh analyses were performed to examine the mesh sensitivity using some experimental data for validation. For the purposes of the grid sensitivity analysis, the axial velocity value at the centre of the nozzle and 5mm downstream burner exit was compared for different grid densities. The process was repeated numerically to examine the grid dependency for the same configuration and boundary conditions. As a result, the total number of nodes of the independent grid used is 11,117,541 with 10,985,610 elements.

Solver Solution Setup

After defining the fluid domain and the mesh size, the boundary condition location and its values in terms of different input parameters should be specified. In this study, different tangential inlet flow rates were used, i.e. 600, 800 and 1000 LPM while the diffusive air flow rate was 50 LPM. The thermo-physical properties were calculated at ambient pressure and temperature. The Pressure-Implicit with Splitting of Operators (PISO) pressure-velocity coupling scheme was employed as a solution method which is recommended for steady state and transient calculations on high skewed meshes. A suitable option for the discretization for the pressure, momentum, turbulent kinetic energy, and the turbulent dissipation rate are selected.

Accurate results were achieved after the convergence was done. In Fluent, throughout the path towards convergence, the governing equations are solved through iterations that depend on the mesh size, the numerical method used and problem physics. The convergence residuals represent an indication of the solution satisfaction to the discrete form of the governing equations. In this work, the absolute convergence criteria were set to 10^{-4}.

Results and discussion

The principle motivation of this paper is to investigate the applicability of the computational procedure to swirling flows and study the flow dynamics in swirl burners. To obtain an accurate CFD outcome and ensure that the simulation results are sensible and reasonable, the mathematical model should be verified and compared with some experimental findings. Once the numerical approach is validated, it could be expanded to study the effect of different boundary and initial conditions. Five planes were set out at the computational domain downstream the burner exit nozzle to calculate different unknown properties such as axial velocity, kinetic energy, turbulent intensity and pressure. These planes are P1=1mm, P2=5mm, P3=10 mm, P4=15 mm and P5=25 mm from the burner mouth.

Figures 2 shows the CFD results of the axial velocity profile for 600 LPM tangential inlet at plane P2 (5 mm downstream the nozzle exit) without air injection. The figure reveals an excellent agreement with the results using LDA measurements. Therefore, the numerical analysis will provide a good prediction of the effect of axial air injection on the burner stability map. Clearly, at the burner centreline the axial velocity is decaying due to the presence of the vortex breakdown.

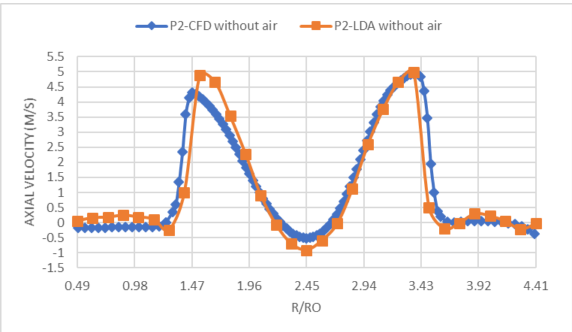

Figure 2: Comparison of axial velocity measured by LDA with CFD result at P2, 600 LPM tangential inlet, X=150 mm and no air injection

The central air injection promotes flame stability by affecting the aerodynamic characteristics of the flow field downstream the burner mouth. It reduces the defect of the axial velocity at the tip of the recirculation zone which is one of the main reasons leading to CIVB flashback. Figure 3 presents a comparison between the axial velocity profile at plane P2 with and without air injection for 600 LPM inlet tangential flow rate. In figure 4 contours of the mean axial velocity with air injection effects are revealed for the whole burner.

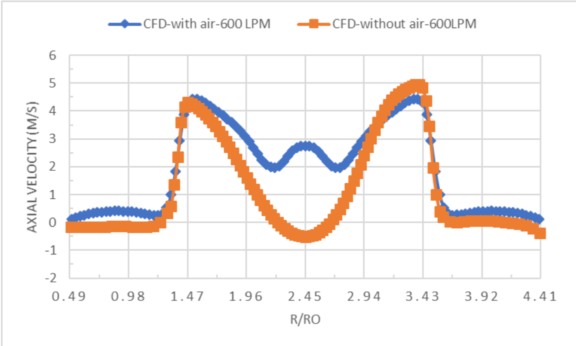

Figure 3. Effect of axial air injection on the defect of axial velocity at P2 with X = 150 mm, 600 LPM tangential inlet

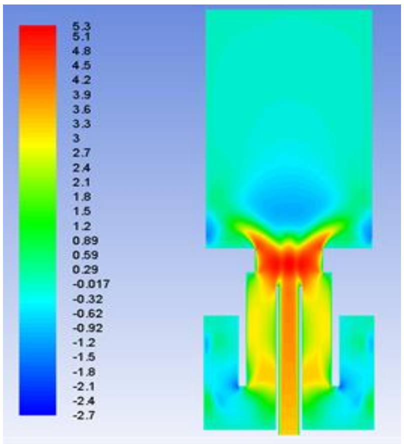

Figure 4. Axial velocity contour (m/s), 600 LPM tangential inlet, X=150 mm

It is clear that the central recirculation zone is pushed up downstream the burner mouth due to hydrodynamics effect of the diffusive jet.

Keeping the vortex core radius as constant as possible in the axial direction is recommended to achieve good stability conditions [34]. Thus, to achieve constant vortex core radius, the central air injection should still be effective at a certain distance downstream the burner centre. Figures 5 illustrates the effect at four different planes downstream the burner mouth (P2=5 mm, P3=10 mm, P4=15 mm and P5=25 mm). It can be seen that central air injection is still effective in those planes. Nevertheless, the degree of the diffusive air injection effect on axial velocity defects is less than that of P2.

A comparison between the axial velocities at P2 plane for different tangential flow rates, i.e. 600, 800 and 1000 LPM, is presented in Figure 6. Upon increasing inlet tangential flow rate, the effect of diffusive air became less pronounced. These findings suggest that the value of diffusive air injection must be proportional to the amount of tangential flow rates. For instance, at low tangential flow rates the amount of diffusive air injection is around 10% to achieve the required effect in terms of flame flashback resistance. However, at high flow rates the ratio is decreased to almost 4 %. Thus, keeping the diffusive to tangential flow ratio at about 8-10% could achieve the desired stability operation.

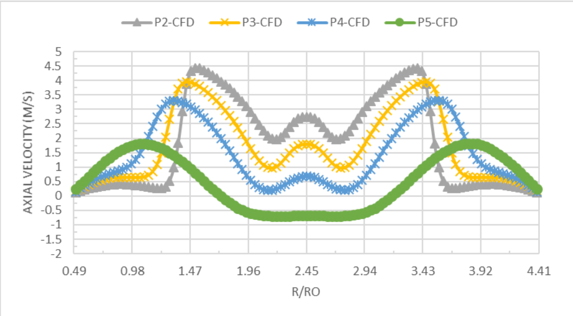

Figure 5. Axial velocity at different planes form burner mouth, 600 LPM tangential inlet, X=150 mm with air injection effect

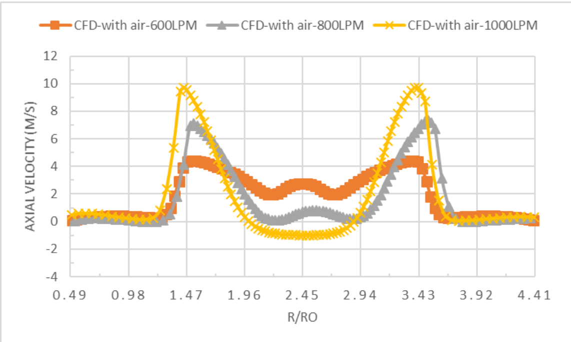

Figure 6. Axial velocity comparison for different tangential inlets at P2, X=150 mm

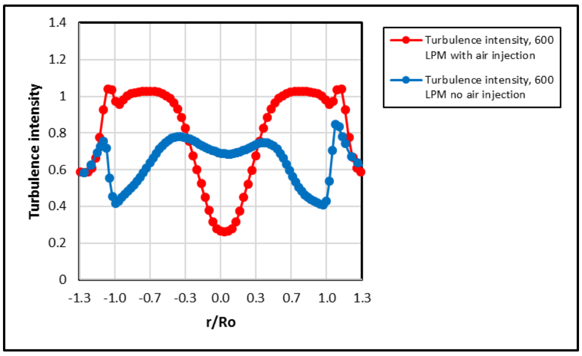

Figure 7. CFD results, three-dimensional turbulent intensity values, effect of central air injection, 600 LPM, at P2 plane, X=150 mm

Turbulence intensity fluctuations downstream the burner mouth have crucial effects on the stability regime because they have direct effect on turbulent flame speed, consequently flame flashback propensity [35]. It can be seen from figures 7 that without central air injection, high levels of turbulence intensity are observed at a 600 LPM tangential inlet flow rate. This reveals the existence of the tip of the central recirculation zone CRZ and its interaction with the incoming flow, producing high amounts of turbulence in this region, with the peaks of the turbulence intensity values indicating the presence of shear layers [35].

Conclusions

This project draws the following conclusions,

1. ANSYS Fluent produced reasonable results with good validation using RANS models, showing good correlation with experimental data. The approach can be used for different configurations and boundary conditions with a good mesh grid.
2. The use of central air injection is a promising technology that can provide further resistance to flashback for wider operability limits.
3. Investigating the swirling flow characteristics downstream burner mouth, especially axial velocity and turbulence, proved to be an important approach in determining the effects of diffusive injection on swirling flows and hence the operation stability regime of the combustor.

Acknowledgments

This paper is part of PhD work sponsored by University of Babylon and University of Diyala. Ali Alsaegh and Fares Hatem thanks the Iraqi government for their support. Thanks to the technician staff of the combustion lab at Cardiff University.

References

[1] N. Syred, M. Abdulsada, A. Griffiths, T. O'Doherty, and P. Bowen, "The effect of hydrogen containing fuel blends upon flashback in swirl burners," *Appl. Energy*, vol. 89, no. 1, pp. 106–110, 2012.

[2] C. Duwig and C. Fureby, "Large eddy simulation of unsteady lean stratified premixed combustion," *Combust. Flame*, vol. 151, no. 1–2, pp. 85–103, 2007.

[3] S. Taamallah, K. Vogiatzaki, F. M. Alzahrani, E. M. A. Mokheimer, M. A. Habib, and A. F. Ghoniem, "Fuel flexibility, stability and emissions in premixed hydrogen-rich gas turbine combustion: Technology, fundamentals, and numerical simulations," *Appl. Energy*, vol. 154, pp. 1020–1047, 2015.

[4] M. Chaos and F. L. Dryer, "Syngas Combustion Kinetics and Applications," *Combust. Sci. Technol.*, vol. 180, no. 6, pp. 1053–1096, 2008.

[5] V. Di Sarli and A. Di Benedetto, "Laminar burning velocity of hydrogen-methane/air premixed flames," *Int. J. Hydrogen Energy*, vol. 32, no. 5, pp. 637–646, 2007.

[6] J. Wang, Z. Huang, C. Tang, H. Miao, and X. Wang, "Numerical study of the effect of hydrogen addition on methane-air mixtures combustion," *Int. J. Hydrogen Energy*, vol. 34, no. 2, pp. 1084–1096, 2009.

[7] A. E. E. Khalil and A. K. Gupta, "Hydrogen addition effects on high intensity distributed combustion," *Appl. Energy*, vol. 104, pp. 71–78, 2013.

[8] N. Syred and J. M. Beér, "Combustion in swirling

flows: A review," *Combust. Flame*, vol. 23, no. 2, pp. 143–201, Oct. 1974.

[9] R. C. Chanaud, "Observations of oscillatory motion in certain swirling flows," *J. Fluid Mech.*, vol. 21, no. 1, pp. 111–127, Jan. 1965.

[10] S. Leibovich, "The Structure of Vortex Breakdown," *Annu. Rev. Fluid Mech.*, vol. 10, no. 1, pp. 221–246, Jan. 1978.

[11] O. Lucca-Negro and T. O'Doherty, "Vortex breakdown: A review," *Prog. Energy Combust. Sci.*, vol. 27, no. 4, pp. 431–481, 2001.

[12] N. Syred, "A review of oscillation mechanisms and the role of the precessing vortex core (PVC) in swirl combustion systems," *Prog. Energy Combust. Sci.*, vol. 32, no. 2, pp. 93–161, 2006.

[13] G. Baumgartner and T. Sattelmayer, "Experimental Investigation of the Flashback Limits and Flame Propagation Mechanisms for Premixed Hydrogen-Air Flames in Non-Swirling and Swirling Flow," in *Volume 1A: Combustion, Fuels and Emissions*, 2013, p. V01AT04A010.

[14] T. Lieuwen, V. McDonell, D. Santavicca, and T. Sattelmayer, "Burner Development and Operability Issues Associated with Steady Flowing Syngas Fired Combustors," *Combust. Sci. Technol.*, vol. 180, no. 6, pp. 1169–1192, 2008.

[15] E. Tangermann, M. Pfitzner, M. Konle, and T. Sattelmayer, "Large-Eddy Simulation and Experimental Observation of Combustion-Induced Vortex Breakdown," *Combust. Sci. Technol.*, vol. 182, no. 922482307, pp. 505–516, 2010.

[16] M. Konle, F. Kiesewetter, and T. Sattelmayer, "Simultaneous high repetition rate PIV-LIF-measurements of CIVB driven flashback," *Exp. Fluids*, vol. 44, no. 4, pp. 529–538, 2008.

[17] T. Sattelmayer, C. Mayer, and J. Sangl, "Interaction of Flame Flashback Mechanisms in Premixed Hydrogen-Air Swirl Flames," *J. Eng. Gas Turbines Power*, vol. 138, no. January, pp. 1–13, 2014.

[18] C. Heeger, R. L. Gordon, M. J. Tummers, T. Sattelmayer, and A. Dreizler, "Experimental analysis of flashback in lean premixed swirling flames: Upstream flame propagation," *Exp. Fluids*, vol. 49, no. 4, pp. 853–863, 2010.

[19] A. Valera-Medina, N. Syred, P. Kay, and A. Griffiths, "Central recirculation zone analysis in an unconfined tangential swirl burner with varying degrees of premixing," *Exp. Fluids*, vol. 50, no. 6, pp. 1611–1623, 2011.

[20] M. Konle and T. Sattelmayer, "Interaction of heat release and vortex breakdown during flame flashback driven by combustion induced vortex breakdown," *Exp. Fluids*, vol. 47, no. 4–5, pp. 627–635, 2009.

[21] T. Reichel and C. O. Paschereit, "Experimental Investigation of Flame Stability of Swirl-Stabilized , Lean Pre- mixed Hydrogen Flames," 2014.

[22] T. G. Reichel, S. Terhaar, and O. Paschereit, "Increasing Flashback Resistance in Lean Premixed Swirl-Stabilized Hydrogen Combustion by Axial Air Injection," *J. Eng. Gas Turbines Power*, vol. 137, no. 7, p. 71503, 2015.

[23] J. Lewis, A. Valera-Medina, R. Marsh, and S. Morris, "Augmenting the structures in a swirling flame via diffusive injection," *J. Combust.*, vol. 2014, 2014.

[24] H. K. Versteeg and W. Malalasekera, *An introduction to computational fluid dynamics: the finite volume method*. Pearson Education, 2007.

[25] P. Fokaides, M. Weiß, M. Kern, and N. Zarzalis, "Experimental and numerical investigation of swirl induced self-excited instabilities at the vicinity of an airblast nozzle," *Flow, Turbul. Combust.*, vol. 83, no. 4, pp. 511–533, 2009.

[26] D. G. Sloan, P. J. Smith, and L. D. Smoot, "Modeling of swirl in turbulent flow systems," *Prog. Energy Combust. Sci.*, vol. 12, no. 3, pp. 163–250, 1986.

[27] T. O'Doherty, a. J. Griffiths, N. Syred, P. J. Bowen, and W. Fick, "Experimental Analysis of Rotating Instabilities in Swirling and Cyclonic Flows," *Dev. Chem. Eng. Miner. Process.*, vol. 7, no. 3–4, pp. 245–267, 2008.

[28] B. Guo, T. A. G. Langrish, and D. F. Fletcher, "CFD simulation of precession in sudden pipe expansion flows with low inlet swirl," *Appl. Math. Model.*, vol. 26, no. 1, pp. 1–15, 2002.

[29] Y. Sommerer, D. Galley, T. Poinsot, S. Ducruix, F. Lacas, and D. Veynante, "Large eddy simulation and experimental study of flashback and blow-off in a lean partially premixed swirled burner," *J. Turbul.*, vol. 5, no. January 2015, pp. 37–41, 2004.

[30] N. Syred, A. Giles, J. Lewis, A. Valera-Medina, P. Bowen, and A. Griffiths, "Tangential Velocity Effects and Correlations for Blow-Off and Flashback in a Generic Swirl Burner and the Effect of a Hydrogen Containing Fuel," in *51st AIAA Aerospace Sciences Meeting*, 2013.

[31] A. . Fallis, "Fundamentals of fluids mechanics," *J. Chem. Inf. Model.*, vol. 53, no. 9, pp. 1689–1699, 2013.

[32] B. E. Launder and D. B. Spalding, "The numerical computation of turbulent flows," *Comput. Methods Appl. Mech. Eng.*, vol. 3, no. 2, pp. 269–289, 1974.

[33] ANSYS Inc. (US), "ANSYS Fluent Theory Guide," vol. 15317, no. January, pp. 724–746, 2016.

[34] S. Burmberger and T. Sattelmayer, "Optimization of the Aerodynamic Flame Stabilization for Fuel Flexible Gas Turbine Premix Burners," *J. Eng. Gas Turbines Power*, vol. 133, no. 10, p. 101501, 2011.

[35] D. Beerer, V. McDonell, P. Therkelsen, and R. K. Cheng, "Flashback and Turbulent Flame Speed Measurements in Hydrogen/Methane Flames Stabilized by a Low-Swirl Injector at Elevated Pressures and Temperatures," *J. Eng. Gas Turbines Power*, vol. 136, no. 3, p. 31502, 2013.