American Society of Mechanical Engineers

ASME Accepted Manuscript Repository

Institutional Repository Cover Sheet

Cranfield Collection of E-Research - CERES

ASME Paper Title: Preliminary CFD Study on the effect of fuel injector coking on fuel spray characteristics

Authors: Agarwal P, Sethi V, Gauthier PQ, Sun X, Liu Y

ASME Conf Title: ASME 2017 Gas Turbine India Conference

Volume/Issue: ____ Volume 1_____ Date of Publication (VOR* Online) ___ 02/02/2018__

ASME Digital Collection URL: https://asmedigitalcollection.asme.org/GTINDIA/proceedings-abstract/GTINDIA2017/58509/V001T04A015/243928

DOI: 10.1115/GTINDIA2017-4838

*VOR (version of record)

Proceedings of the ASME 2017 Gas Turbine India Conference
GTINDIA 2017
December 7-8, 2017, Bangalore, INDIA
DOI: 10.1115/GTINDIA2017-4838

GTIndia2017-4838

PRELIMINARY CFD STUDY ON THE EFFECT OF FUEL INJECTOR COKING ON FUEL SPRAY CHARACTERISTICS

Parash Agarwal *
PhD Researcher
SATM Cranfield University
Cranfield, Bedfordshire MK43 0AL
Email: p.agarwal@cranfield.ac.uk

Dr. Vishal Sethi
Gas Turbine Combustion Lecturer
SATM Cranfield University
Cranfield, Bedfordshire MK43 0AL
Email: v.sethi@cranfield.ac.uk

Dr. Pierre Q. Gauthier
Visiting Academic
SATM Cranfield University
Cranfield, Bedfordshire MK43 0AL
Email: p.q.gauthier@cranfield.ac.uk

Xiaoxiao Sun
PhD Researcher
SATM Cranfield University
Cranfield, Bedfordshire MK43 0AL
Email: x.sun@cranfield.ac.uk

Yize Liu
PhD Researcher
SATM Cranfield University
Cranfield, Bedfordshire MK43 0AL
Email: yizeliu.liu@cranfield.ac.uk

ABSTRACT

Fuel injector coking involves deposit formation on the external or the internal surfaces of an injector or nozzle. This deposition of carbonaceous particles can result in uneven fuel-spray characteristics or localised burning (hot spots), which may eventually lead to mechanical failure or simply have a detrimental effect on the combustion system. This study focuses on the use of numerical methods to investigate the effect of coke formation on both the atomiser internal flow passages and its spray characteristics. Three different cases are examined; one investigating the clean injector; the second investigating the effect of internal coking; and the third investigating the effect of nozzle tip coking. A pressure swirl atomiser was considered for the purpose of the study. Validation of the numerical results for the clean injector condition is carried out against published experimental data. Two arbitrary geometries of coke deposits were created. The Volume of Fluid (VOF) multiphase model has been used in conjugation with a Geometrical Reconstruction Scheme (GRS) to simulate the interface representing the two phases. Spray cone angle, and the liquid film thickness for the clean injector condition, predicted by numerical ~~simulation~~ agreed well with the experimental data. Instabilities in the air core and the spray angle were also observed because of the presence of coke layers. Fouling present on the injector tip resulted in the earlier breakup of the film thereby affecting the flame lift-off length. These stated observations can have significant implications both on the performance as well as the life of the combustion systems and hence the significance of this study.

Keywords: *Atomisers, Jet Fuel Thermal Stability, Multi Phase Flow, Volume of Fluid*

NOMENCLATURE

α Volume Fraction
ρ Density
A Cross Section Area
d Diameter
L Film Thickness
P Pressure
r Radius
t Time

*Address all correspondence to this author.

Published by The American Society of Mechanical Engineers. This is the Author Accepted Manuscript issued with:
Creative Commons Attribution License (CC BY). The final published version is available online at DOI: 10.1115/GTINDIA2017-4838.
Please refer to any applicable publisher terms of use.

SUBSCRIPT

r Radial Direction
sw Swirl Chamber
1 Original Condition (3-D
2 Equivalent Condition (2-D)

LIST OF ABBREVIATIONS

CFD Radial Direction
CST Swirl Chamber
DPIV Original Condition (3-D)
GRS Equivalent Condition (2-D)
LDV Laser Doppler Velocimetry
LES Large Eddy Simulation
PIV Particle Image Velocimetry
VHBPR Very High By-Pass Ratio
VOF Volume of Fluid

INTRODUCTION

Historical trends in the improvement of engine performance have been characterised by an increase in engine bypass ratio and overall core pressure ratios. Increases in these cycle parameters has had a direct influence on enhancing the propulsive efficiency of the engines. This has led to the design concept of the Very High By-Pass Ratio (VHBPR) engines. One drawback of increasing these parameters is the impact on thermal stressing of the fuel. When the temperature of the fuel is increased beyond a certain threshold limit, the fuel is said to be thermally stressed. Thermal stressing of the fuel can lead to an accelerated rate of deposition of carbonaceous particles on the injector surfaces; the term "coking" has been coined to refer to this phenomenon [1]. This effect is further manifested with the increasing use of fuel as a heat sink to cool some of the aircraft systems. This applies in particular to VHBPR geared turbofan engine architectures. A significant amount of heat is generated in the gearbox connecting the low-pressure shaft to the slowly rotating fan, further increasing the amount of heat being dumped to the fuel. It also becomes of concern for modern lean burn combustion system where the fuel passages, which are typically arranged in two separate circuits i.e. main and pilot, are often designed as an integral part of the fuel injector heat management. While more heat is bound to be dumped to the fuel, the improvement in cycle efficiency provided by these VHBPR concepts will lead to decreased levels of fuel flow at a given thrust. This decrease in levels of fuel flow per unit thrust leads to an increase in the amount of heat being dump per unit fuel to further increase, thereby exaggerating the issue of coking. It can be further argued that the issue of coking will be of greater concern in the future because of the trend of increasing flight speed. The temperature of the ram air taken on board increases, making the bypass air less effective as a coolant and hence increasing dependency on the use of the fuel as a primary coolant. However, from a combustion point of view, this elevation of fuel temperature is not altogether undesirable, because it reduces fuel viscosity, thereby promoting finer atomisation.

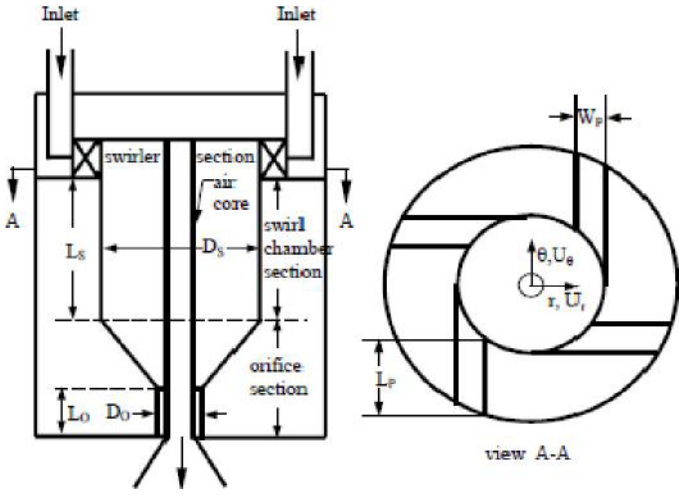

FIGURE 1. Pressure Swirl Atomiser Schematics [8]

Literature on the study of jet-fuel thermal stability is extensive. There is also a growing body of literature that recognises the importance of the study of this topic. Numerous experiments have been carried out to examine various aspects of the subject. Formation of these insoluble deposits under thermal stress can broadly be classified as an outcome of two mechanisms, autoxidation and pyrolysis. Each of these mechanisms consists of a series of endothermic reactions, ultimately leading to the precipitation of "gum like substances" [1]. Studies carried out by Andresen [2] concluded that formation of these deposits because of autoxidation is dominant from the fuel temperature of about 150˚C to 350˚C and pyrolysis is dominant for a temperature exceeding 300˚C. These temperature ranges are not absolute; different research efforts have shown a slight variation in these values [3] [4]. Some studies have also shown that fuel features such as high viscosity, low volatility and reactivity of unsaturated hydrocarbon chains such as olefins and aromatics can exaggerate the formation of carbon deposits on the injector tip [5].

Emission characteristics will also be affected because of the presence of local hot spots [6]. The lift-off length of the fuel is also reduced, mainly due to additional heat transfer from the nozzle-tip deposit to the fuel jet, thereby affecting combustion stability [7]. A pressure swirl atomiser is considered for the purpose of this study; since they contain small internal passages making it more susceptible to plugging or blockage [7]. In pressure swirl atomiser swirling motion is imparted to the fuel, leading it to spread out, under the action of centrifugal force in the form of a hollow cone as soon as it leaves the exit orifice. Schematics of the pressure swirl atomiser is shown in Figure 1. Owing to its use a large number of studies are available in published literature. The focus of earlier studies has been on the phenomenon taking place after the liquid exits the atomiser, breakup of a simple form of bulk fluid on the statistical nature of the fuel sprays and the development of correlations and operating pa-

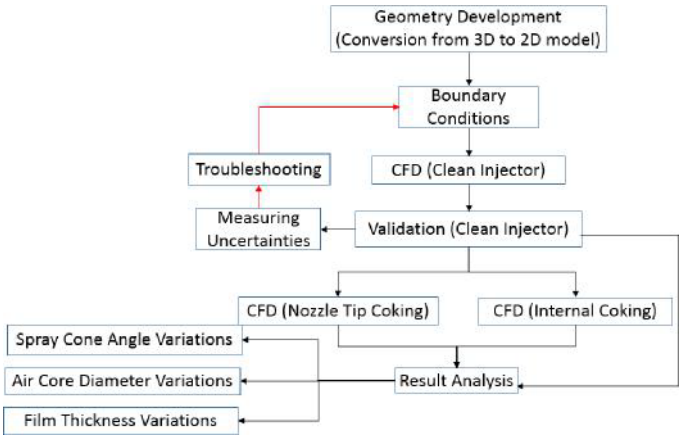

FIGURE 2. Methodology Flow Chart

rameters relevant to spray characteristics. Rizk and Lefevbre [9], Ashraf and Milliand [10], and Nonnenmacher and Piesche [11] studied the interface between the liquid and the surrounding gas. However, due to difficulties in measurement inside a small scale atomiser and the challenges in modelling two-phase flow, study on the impact of the flow phenomenon inside the atomiser have not received much attention. The interface between the fuel and the air is not a known a *priori* and hence must be determined as a part of the solution.

The synopsis of the methodology adopted for the study is shown in Figure 2. Three different simulation setup were devised, one studying the fuel spray characteristics of a clean injector, other studying the effect of internal coking and finally studying the effect of nozzle tip coking on the fuel spray characteristics. Literatue published by Ma [8] has has been used to develop the geometry for the fuel injector.

Experimental Setup

As a compliment to the limitations of the experimental techniques Computational Fluid Dynamics (CFD) provides insight into the dynamics of the internal flow characteristics of a pressure swirl atomiser. However, upon applying CFD to model fluid-dynamic behaviour, some form of validation is required to establish the accuracy of the model set-up. Hence in order to validate the CFD model setup, a comparative study was carried out between the clean injector condition with the experimental data available in public domain. The experimental data reported by Ma [8] has been used for this study. In this study, detailed flow structure inside simplex atomiser was studied using Digital Particle Image Velocimetry (DPIV) and Laser Doppler Ve-locimetry (LDV). The effect of changes to flow Reynolds number, density ratio, and the orifice geometry on the internal flow field of the atomiser were examined and their relationship with external spray characterisation. The atomiser used for the study had three interchangeable parts: inlet slot section, swirl chamber, and the exit orifice. All these three parts were made out of plexiglass. Water is fed through the tangential inlet ports into the swirl chamber as shown in Figure 1. Images obtained from particle image velocimetry are shown in Figure 3. The blank area in the centre is because of the presence of the air core. The presence of both the positive and the negative flow fields resulted in fairly sophisticated flow field patterns within the atomiser.

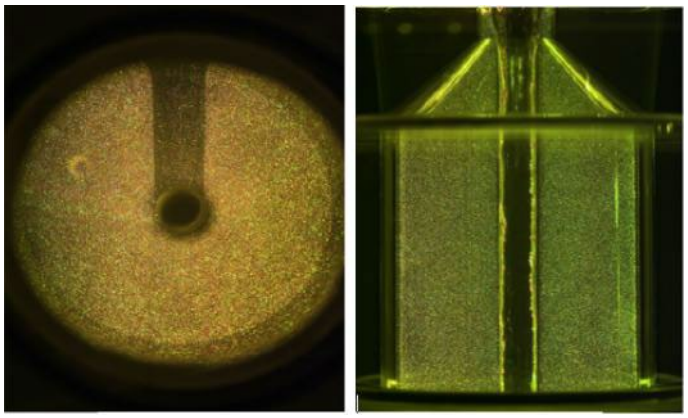

FIGURE 3. PIV Side and Top View of the Swirl Chamber [8]

Numerical Methods

Previous numerical studies have relied upon various models and techniques to successfully capture a sharp and accurate interface and hence analyse the dependency of spray parameters on boundary conditions such as swirl strength and the inlet pressure. The computational domain in this study was simplified to a 2-D axisymmetric domain so as to reduce the computational efforts. This simplification was carried out in accordance to a similar work by Ashraf [10]. The two-dimensional model of the flow assumes the flow to be axisymmetric in nature. This fundamental assumption requires the determination of a hypothetical “annular” slot. The width of this“annular” slot, along with radial and tangential velocity at the inlet are evaluated by matching the annular momentum, total mass flow rate, and the kinetic energy at the inlet of the “real” port. Using the advantage of the periodicity of the flow, only the upper half of the atomiser is modelled as shown in Figure 4. Several systematic reviews have been made comparing the difference between laminar and turbulent flow in order to get a deeper understanding of the nature of the flow. Hansen *et al*. [12] conducted his study using Large Eddy Simulations (LES) and laminar model amalgamating VOF and Eulerian multiphase model. Hansen *et al*. [12] concluded that combination of a laminar-VOF method was the best in regards to computational time and accuracy. Similarly, studies carried out by Baharanchi [13] and Hansen *et al*. [12] using realizable k- model, reported that the model was initially able to capture the interface, but it began to diverge once the turbulent viscosity started to escalate unboundedly in a region inside the swirl cham-

ber. This occurred because of the excess vorticity present on the outer edge of the interface inside the swirl chamber, hence deforming the film inward and destabilising and ultimately diverging the numerical solution. Detailed description of VOF model and different tracking schemes could be found in publications by Hirt and Nichols [14], Rider and Kothe [15], Gueyffier *et al*. [16] and Gorokhovski and Herrmann [17].

In accord to the work carried out by Hansen *et al*. [12] laminar-VOF multiphase model has been used for this study. VOF has been used in conjugation with a GRS to simulate the interface representing the two phases *i.e.* water and the air. GRS ensures that the two-phase boundary are captured in one computational cell hence avoiding the smearing effect [18]. Continuum Surface Tension (CST) modelling scheme was used to model the surface tension acting between the water and air interface. Assuming the flow to be incompressible, velocity inlet boundary condition was applied to the inlet port. The density of water was estimated to be 998.2 kgm^{-3}. Assuming the conservation of mass, momentum and energy the inlet port area was estimated to be 406 mm^2 and the magnitude of the total velocity was estimated to be 2.346 ms^{-1}. The swirl velocity component was estimated to be -1.21ms^{-1} and tangential velocity was estimated to be 2.0 ms^{-1}. Gauge pressure of 1 atm was implemented at the inlet port. It is noteworthy, the inlet static pressure pressure has been the fundamental criteria for comparisons as an indication of resistance to flow due to the existence of a volumetric force introduced by the CST model. Wall boundary condition were implemented for the boundaries of the swirl chamber, convergent section and exit orifice; this was done assuming a no slip condition on the respective surfaces. In order to improve the resolution of flow parameters because of the variation in flow fluxes adaptive grid refinement techniques have been used. Adaptive time step size setting has also been implemented to satisfy the global CFL condition. This feature is enabled for both the explicit and implicit volume fraction formulations when using the VOF model. Pressure Staggered Option (PRESTO) was employed for the computation of face pressures. A pressure outlet boundary condition was implemented to define the outflow condition. Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm was chosen for the pressure velocity coupling while momentum, volume fraction, and all other flow variables are discretized using the second order accurate scheme for attaining higher accuracy in modelling.

RESULTS AND DISCUSSIONS

CFD model setup is validated by comparing the variations in spray cone angle, film thickness and the discharge coefficient with the results obtained from the experiment. The spray cone angle was measured by drawing a tangent to the outer spray interface. Thickness of the film formed was measured at station x=70.05 mm (centre of the orifice). The pressure differential (ΔP) of 0.8776 atm was obtained from the CFD model. It can

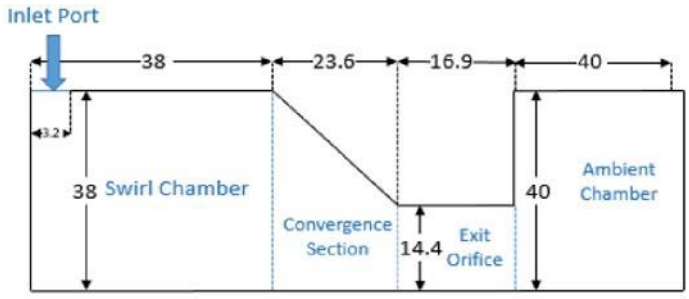

FIGURE 4. Equivalent 2-D Simplex Nozzle Design

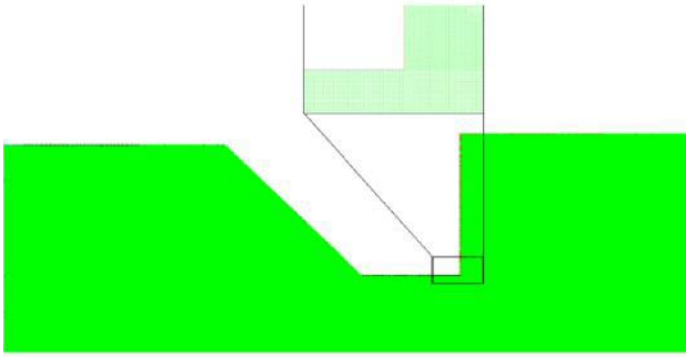

FIGURE 5. Mesh Generated for Clean Injector Condition

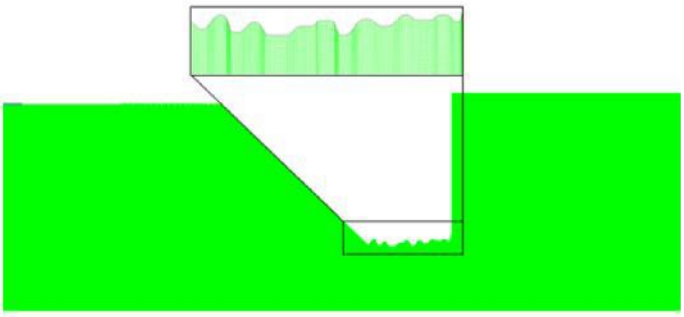

FIGURE 6. Mesh Generated for Internal Coking Condition

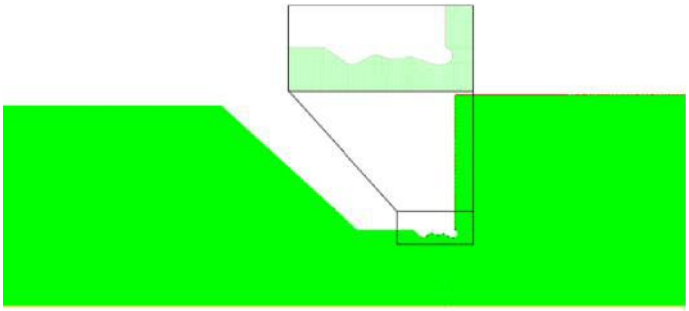

FIGURE 7. Mesh Generated for Nozzle Tip Coking Condition

TABLE 1. Mesh Size Parameters

	Number of Elements	Minimum Mesh Size (10^{-6}m)	Maximum Mesh Size (10^{-6}m)
Clean Injector	348810	0.200	38.3
Internal Coking	363252	0.152	38.6
External Coking	333286	0.178	37.4

TABLE 2. Comparison of Experimental and CFD results

Parameter	Experiment	CFD
Spray Cone Angle	76.9	78
Film Thickness	3.06	3.08
Discharge Coefficient	0.20	0.206

TABLE 3. Variations in Spray Cone Angle vs Time

	Case 1	Case 2	Case 3
t= 0.268 s	81.3^0	85.7^0	83.1^0
t= 0.285 s	80.6^0	80.8^0	82.3^0
t= 0.3054 s	77.8^0	84.0^0	82.6^0

be seen from Table 2 that the computational results agree well with the experimental measurements with the variation in spray cone angle obtained from CFD results being about +1.4%. The variation in film thickness being 0.0065% and the variation in discharge coefficient being 3%.

Figure 9 shows the computed velocity vectors on a planar cross section in the pressure swirl atomiser for the clean injector condition. The velocity vectors have both radial and tangential velocity components. The swirl velocity is perpendicular to the cross section and hence not shown in the plot. As a result of this swirling motion within the swirl chamber, the pressure decreases towards the centre axis. Due to the presence of this low pressure near the centre axis an air core is formed along the centre line as shown in Figure 10. The air enters the atomiser near the centre line and is pulled along the liquid sheet exiting the atomiser. As a result, the air axial velocity is negative near the centre and becomes positive closer towards the sheet. As expected the axial velocity decreases from the atomiser exit to the swirl chamber as shown in Figure 11. In the liquid phase, the negative and positive velocities indicate the existence of a number of small-scale vortices and therefore exhibit the flow patterns with recirculation.

The air core diameter is approximately constant in the swirl chamber and this increases as the flow enters the exit orifice. This is similar to the findings reported by Ma [8]. Photographic evidence of the liquid sheet breakup obtained from numerical simulation is also in accordance to the mechanism proposed by Fraser [19]. Fraser [19] stated three different modes of liquid sheet breakup namely; the formation of a thick rim, perforation of the liquid sheet, and the breakup of the sheet as shown in Figure 12, 13 and 14.

Surface tension forces cause the free edge of the liquid to contract into a thick rim, which then breaks up by a mechanism leading to the disintegration of the liquid sheet. Contour plot of this rim formation for all three cases are shown in Figures 12, 13 and 14 respectively. Solution time for the contour plot taken was 0.2052 s. From Figure 14 it is observed that the presence of coke on the surface of the orifice resulted in an early breakup of the film hence suppressing its formation. This suppression of the rim formation or an early breakup of the liquid film, could significantly vary the spray symmetry and thereby reducing its lift-off length [7]. Similar research efforts to study the effect of diesel injector coking on fuel spray characteristics, have attributed this effect to the additional heat transfer that take place from the fuel injector passages to the liquid film as a result of the increased surface area due to the formation of the coke layer [6]. Variation in spray cone angle for all the three cases at t=0.3054 s is shown in Table 3. Variations in spray cone angle for clean injector condition was estimated to be below 5%. This small variation in spray cone angle can be attributed to the turbulence present in the flow. Variation in spray cone angle for the internal coking condition was estimated to be about 10%. This large variation in spray cone can be attributed to the instabilities created by the presence of a coke layer at the orifice exit and hence greater turbulence. These instabilities present in the liquid film can lead to enhanced disintegration of the sheet thereby promoting finer atomisation. Furthermore, the spray cone angle is increased which can lead to an increase in the exposure of the droplets to the surrounding air further improving atomisation. This process is however disrupted due to the presence of coke deposits. Presence of these foreign particles on the orifice passage amplifies the growth of disturbances occurring within the injector passages. The disintegration of this liquid sheet at t= 0.285 s for all three cases, is shown in Figures 16, 19 and 22 respectively. It can be seen from Figure 18 that the presence of coke in the fuel injector passage results in an early breakup of the film. This can also be attributed to the instabilities present in the liquid film. These instabilities arise primarily due to the presence of disturbances in the orifice passage.

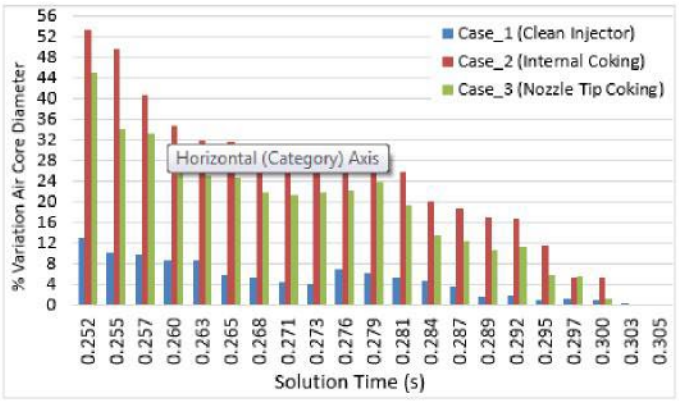

FIGURE 8. Variation in Air Core Diameter (Reference - Air Core Diameter t=0.3054)

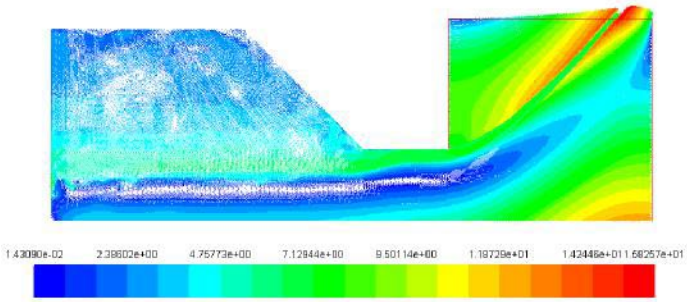

FIGURE 9. Velocity Magnitude Vector Plot (t=0.3054 s)

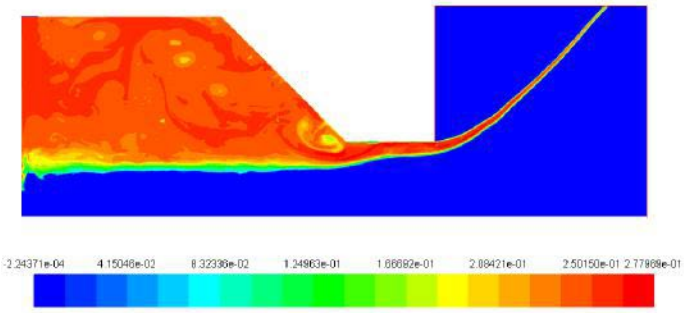

FIGURE 10. Total Pressure (atm) Contour Plot (t=0.3054 s)

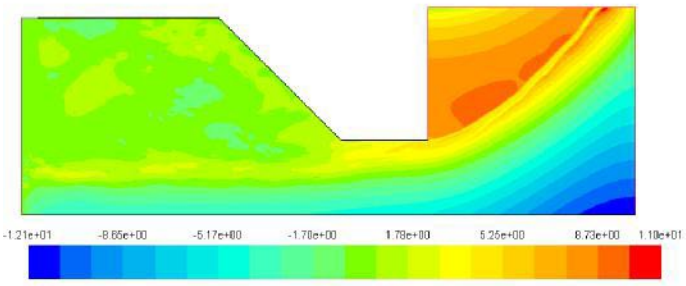

FIGURE 11. Axial Velocity Contour Plot (t=0.3054 s)

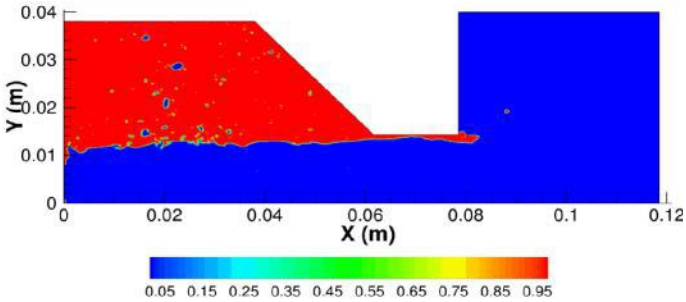

FIGURE 12. Formation of Rim (t = 0.2052 s)

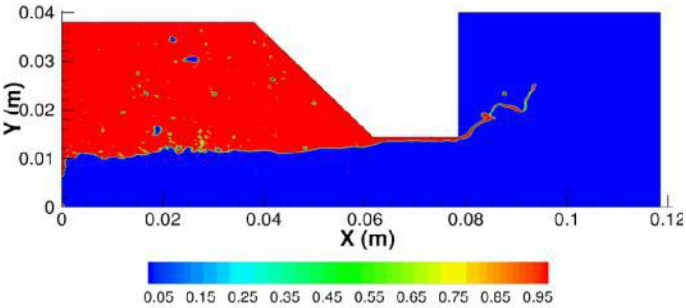

FIGURE 13. Perforation of the Liquid Sheet (t = 0.2212 s)

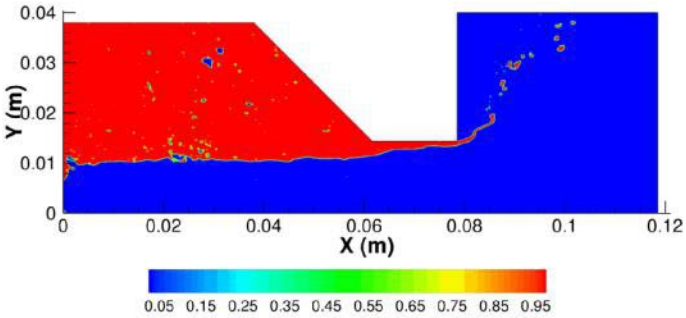

FIGURE 14. Disintegration of the Liquid Sheet (t = 0.2358 s)

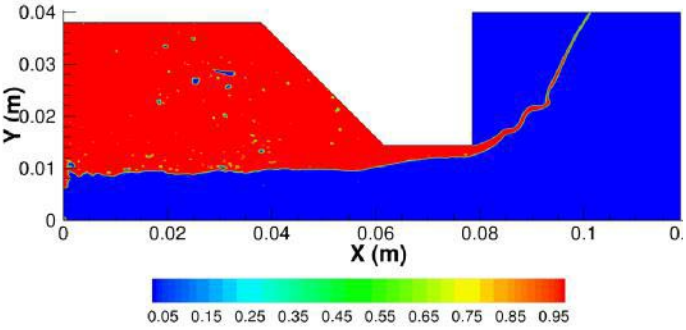

FIGURE 15. Clean Injector - Solution Time 0.268 s

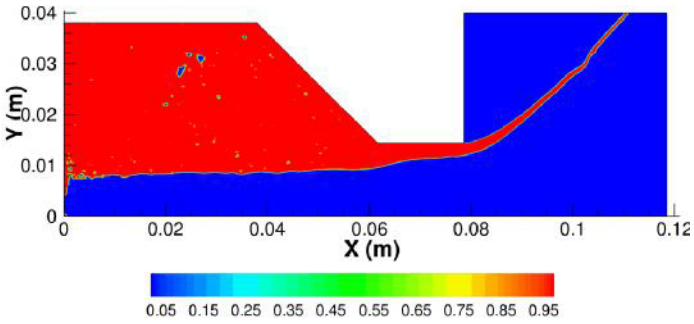

FIGURE 16. Clean Injector - Solution Time 0.285 s

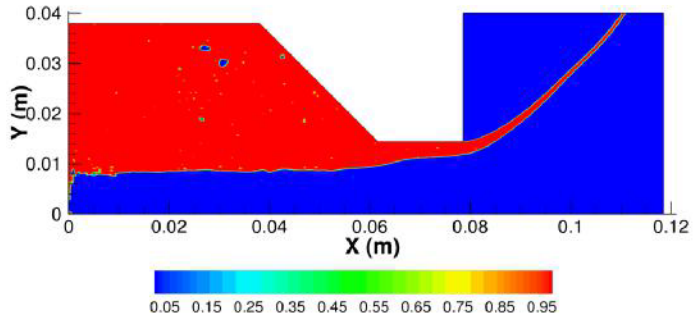

FIGURE 17. Clean Injector - Solution Time 0.300 s

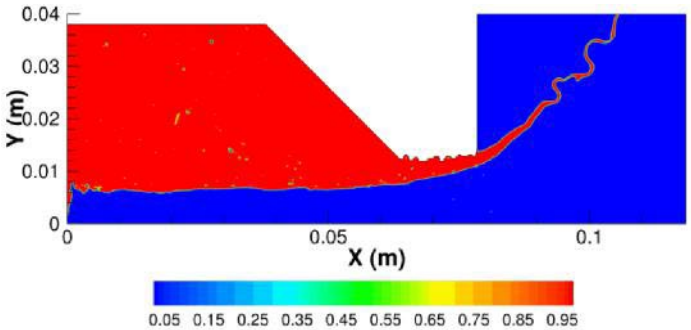

FIGURE 18. Internal Coking - Solution Time 0.268 s

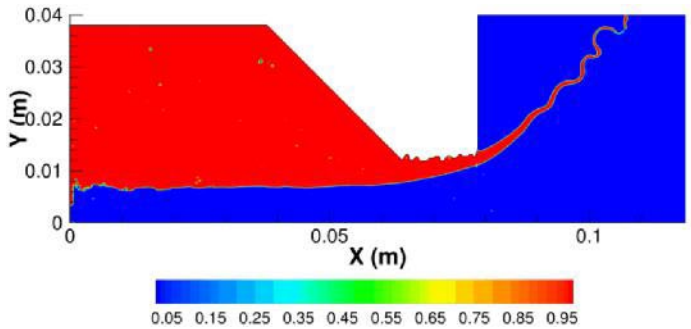

FIGURE 19. Internal Coking - Solution Time 0.285 s

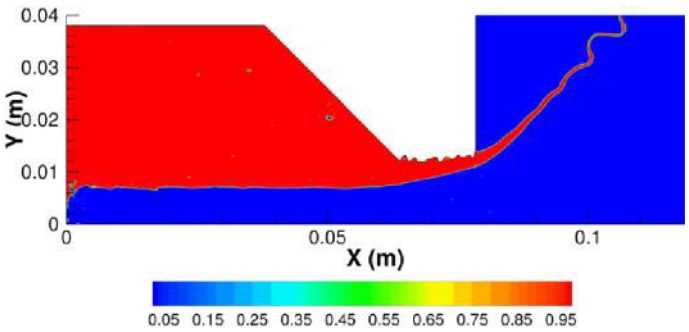

FIGURE 20. Internal Coking - Solution Time 0.300 s

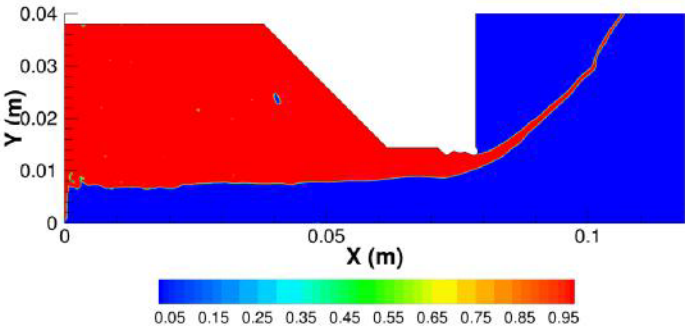

FIGURE 21. Nozzle Tip Coking - Solution Time 0.268 s

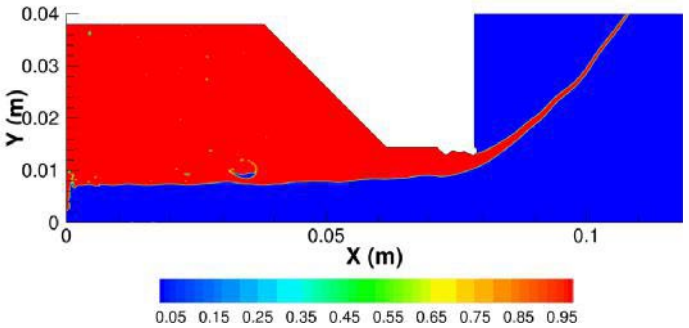

FIGURE 22. Nozzle Tip Coking - Solution Time 0.285 s

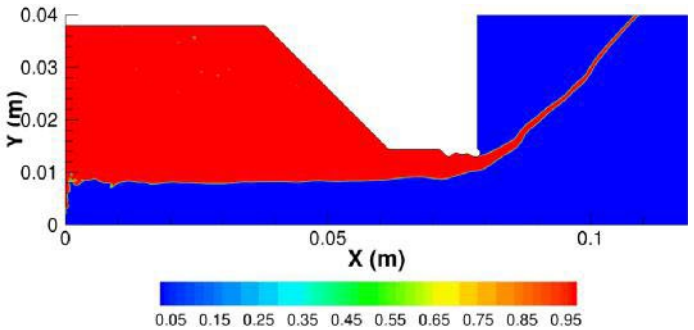

FIGURE 23. Nozzle Tip Coking - Solution Time 0.300 s

CONCLUSION

The formation of coke in the orifice exit creates a disturbance to the flow. These disturbances are what lead to the variation in discharge parameters of the injector. Variations in the air core diameter were also observed. These variations however, seem to diminish with increasing simulation time as shown in Figure 8, indicating that the flow within the atomiser tends to stabilize with increasing flow time. Variations in spray cone angle was also observed because of the instabilities in the flow condition at the air core. This variation in spray cone angle can directly affect the combustor liner life. The combustor liner life cannot be quantitatively evaluated directly as it depends on a range of factors including the design of the combustor liner itself. Variations in liquid film thickness were also observed as a consequence of the instability in the air core due to the presence of coke layer. This instability however resulted in the formation of a thicker film at the orifice exit, hence resulting in a coarser atomisation which is contrary to the effect observed for the spray cone angle. The laminar flow model was able to accurately capture the atomiser flow features like the spray cone angle, formation of liquid film *etc.* it was however not able to capture the disintegration of liquid sheet into finer droplets over a prolonged period of simulation time. Hence considering the use of scale resolving turbulence model like LES might be a more appropriate option. Apart from this, other simplifications were also carried out which include the conversion of the 3-dimensional model to a 2-dimensional axisymmetric domain in which only the upper half of the injector was modeled, in order to reduce the computational efforts. However, geometry of the swirl chamber as well as the orifice exit is very short and hence the flow can have three-dimensional features which a two dimensional domain is unable to predict therefore this can have severe implication to the accuracy of the results obtained. Hence this study can be further extended to study the 3-D nature of the flow both within the injector passages and within the nozzle discharge.

REFERENCES

[1] Lefebvre, A. H., 1989. *Atomisation and Sprays*. Hemisphere Publication Corporation, New York.

[2] Andrésen, J. M., Strohm, J., Sun, L., and Song, C., 2001. "Relationship between the formation of aromatic compounds and solid deposition during thermal degradation of jet fuels in the pyrolytic regime". *Energy & Fuels, 15*(3), pp. 714–723.

[3] Hazlett, R. N., 1991. "Thermal oxidation stability of aviation turbine fuels".

[4] Roan, M. A., and Boehman, A. L., 2004. "The effect of fuel composition and dissolved oxygen on deposit formation from potential jp-900 basestocks". *Energy & Fuels, 18*(3), pp. 835–843.

[5] Pehan, S., Svoljsak, M., Kegl, M., and Kegl, B., 2009. "Biodiesel influence on tribology characteristics of a diesel engine". *Fuel, 88*(6), June, pp. 970–979.

[0] Arpaia, A., Catania, A. E., dAmbrosio, S., A Ferrari, S. P. L., and Spessa", E., 2009. "Injector coking effects on engine performance and emissions". In ASME 2009 Internal Combustion Engine Division Fall Technical Conference, American Society of Mechanical Engineers, pp. 229–241.

[1] Lefebvre, A. H., 1998. *Gas turbine combustion*. CRC press.

[2] Ma, Z., 2002. "Investigation on the internal flow characteristics of pressure-swirl atomizers". PhD thesis, University of Cincinnati.

[3] Rizk, N., and Lefebvre, A., 1980. "The influence of liquid film thickness on airblast atomization". *Journal of Engineering for Power, 102*(3), pp. 706–710.

[4] Ibrahim, A., and Jog, M., 2007. "Nonlinear breakup model for a liquid sheet emanating from a pressure-swirl atomizer". *Journal of Engineering for Gas Turbines and Power, 129*(4), pp. 945–953.

[5] Nonnenmacher, S., and Piesche, M., 2000. "Design of hollow cone pressure swirl nozzles to atomize newtonian fluids". *Chemical Engineering Science, 55*(19), pp. 4339–4348.

[6] Hansen, K. G., Madsen, J., Trinh, C. M., Ibsen, C. H., Solberg, T., and Hjertager, B. H., 2002. "A computational and experimental study of the internal flow in a scaled pressure-swirl atomizer". *Zaragoza, 9*, p. 11.

[7] Baharanchi, A. A., Gokaltun, S., and Eshraghi, S., 2013. "A numerical approach for the simulation of internal nozzle flow in a pressure swirl atomizer using different turbulent models and towards an effective inlet weber number". In ASME 2013 International Mechanical Engineering Congress and Exposition, American Society of Mechanical Engineers, pp. V07AT08A068–V07AT08A068.

[8] Hirt, C. W., and Nichols, B. D., 1981. "Volume of fluid (vof) method for the dynamics of free boundaries". *Journal of Computational Physics, 39*(1), pp. 201–225.

[9] Rider, W. J., and Kothe, D. B., 1998. "Reconstructing volume tracking". *Journal of Computational Physics, 141*(2), pp. 112–152.

[10] Gueyffier, D., Li, J., Nadim, A., Scardovelli, R., and Zaleski, S., 1999. "Volume-of-fluid interface tracking with smoothed surface stress methods for three-dimensional flows". *Journal of Computational Physics, 152*(2), pp. 423–456.

[11] Herrmann, M., 2013. "On simulating primary atomization". *Atomization and Sprays, 23*(11).

[12] Scardovelli, R., and Zaleski, S., 1999. "Direct numerical simulation of free-surface and interfacial flow". *Annual Review of Fluid Mechanics, 31*(1), pp. 567–603.

[13] Fraser, R. P., 1957. "Liquid fuel atomization". In Symposium (International) on Combustion, Vol. 6, Elsevier, pp. 687–701.

https://dspace.org/

Preliminary CFD Study on the effect of fuel injector coking on fuel spray characteristics

Agarwal, Parash; Sethi, Vishal; Gauthier, Pierre Q.; Sun, Xiaoxiao; Liu, Yize

2018-02-02
American Society of Mechanical Engineers (ASME)
https://doi.org/10.1115/GTINDIA2017-4838
Agarwal P, Sethi V, Gauthier PQ, et al., (2017) Preliminary CFD Study on the effect of fuel injector coking on fuel spray characteristics. In: ASME 2017 Gas Turbine India Conference, 7-8 December 2017, Bangalore, India

Downloaded from a DSpace repository. https://dspace.org/