International Polytechnical Journal in Science and Engineering

Special Issue in Automotive Technologies Progress, Volume 1, Issue 1, 2021

EFFECTS OF THE COMBUSTION CHAMBER GEOMETRY ON COMBUSTION AND EMISSION OF A LIGHT-DUTY CI ENGINE

R. Sener[1*], M.Z. Gul[2]

[1] *Batman University, Automotive Engineering Department, Batman, Turkey.*
[2] *Marmara University, Mechanical Engineering Department, Istanbul, Turkey.*
**ramazan.sener@batman.edu.tr*

ABSTRACT

Compression-ignition engine is widely used due to its thermal efficiency, reliability, and fuel economy, while the engine emissions are harmful to the environment and human health. Therefore, emission standards (EPA, Tier, NRE-v / c standards, etc.) limit the exhaust emissions of engines around the world. The most successful method of reduction of exhaust emissions is to optimize the combustion chamber and fluid motion inside the engine. In this study, the CFD method was used to analyze fluid motion, spray, combustion process, and exhaust emissions in a CI engine. A new type of swirl piston bowl was adopted for the test engine. The results were compared to the baseline engine piston bowl on combustion and emission. The results show that piston bowl shape has a critical impact on engine performance and emissions. The swirl piston bowl design contributes to reducing emissions and improving the combustion process, as it increases in-cylinder swirl and turbulence. The swirl piston bowl improves the combustion process but also increases NO_X emissions due to an increase in-cylinder temperature. On the other hand, NO_X emissions can also be reduced when the injection parameters of the engine are optimized to provide the same power as the swirl piston bowls.

Keywords: *Combustion, piston bowl, diesel engine, emission.*

INTRODUCTION

The diesel engine is widely used for non-road purposes, as well as transportation purposes because of its thermal efficiency, reliability, and fuel economy. Diesel engine technology with higher efficiency and lower emission is being developed. These developments have led to an increase in the interest in diesel engines for agriculture and maritime sectors where fuel economy is an important parameter.

Modeling is a very useful method for the development of the internal combustion engine and optimization of its parameters. Modeling of an internal combustion engine will develop as our understanding and knowledge of the physics and chemistry mechanism of the phenomena and as computers continue to increase their ability to solve complex equations. The models describe the thermodynamics, fluid flow, heat transfer, combustion, and emission formation events of the engines [1–3].

The combustion chamber geometry has an important effect on the performance and emission characteristic of the direct injection compression engine [4]. It depends on the combustion chamber design to ensure the most efficient combustion in the cylinder. The piston bowl geometry, which provides the fuel injection to mix with the charge inside the cylinder increases the performance. In order to reduce the exhaust emission and maximize the performance of the single-cylinder diesel, three different piston bowl (i.e., base, reentrant, swirl) to the test engine. The results are compared in terms of in-cylinder pressure, temperature distribution, in-cylinder fluid motion and equivalence ratio, and also, NO_X, CO, CO_2, and soot emission.

CFD MODELING

The test engine is a single-cylinder, water-cooled naturally aspirated four-stroke diesel engine. A summary of the engine's operating parameter is listed in table 1. The engine has a bore diameter of 85-mm, a stroke of 90-mm, and displacement volume of 510 cm^3. The engine geometries with three different piston bowls have the same volume and hence the same compression ratio. They were simulated using the same operating conditions to compare their effectiveness.

Table 1. Main characteristics and operating parameters of the test engine

Engine Type	Single cylinder, naturally aspirated, and direct injection
Total displacement	510 cm^3
Bore	85 mm
Stroke	90 mm
Con. rod length	144.5 mm
Compression ratio	17.5
Max. Power	12 HP
Max. Torque	32 Nm
Injection pressure	20000 kPa, mechanical
Spray angle	120 deg.

The existing geometry was named the baseline or base piston bowl. It is a primitive design and not used commonly. Reentrant piston bowl is widely used in literature due to its improving the combustion and thermal efficiency [5,6]. While swirl piston bowl (DSB) has lateral swirl effect and improve the air-fuel mixing with wall-flow-guide [7–9]. These piston bowls were adopted to the test engine according to the base geometry and characteristics, and the total in-cylinder volume and compression ratio were kept constant. The piston bowl geometries are shown in fig. 1.

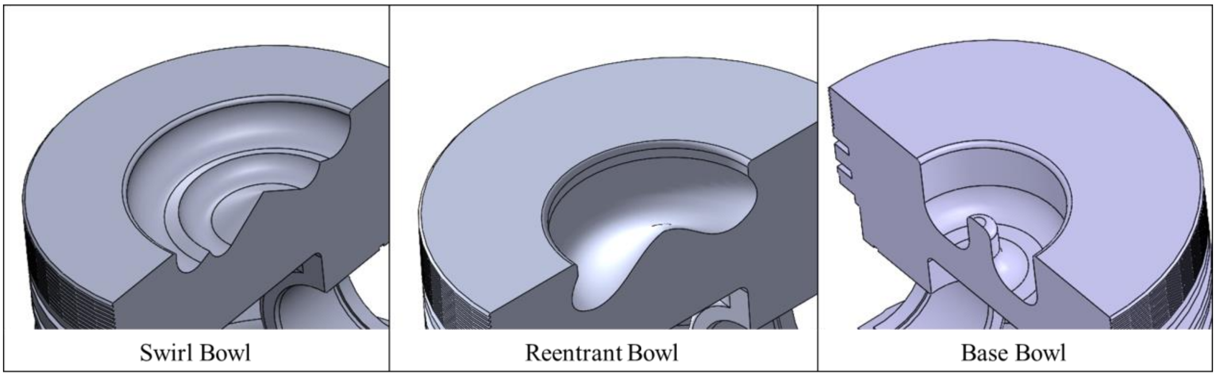

Figure 1. Geometries of the different piston bowls.

The numerical simulations were performed using Converge commercial CFD code [10]. The modified cut-cell Cartesian grid method has been used for meshing. The domain is re-meshed at each time-step for moving boundaries. The adaptive mesh refinement (AMR) is used to refine the grid based on fluctuating and moving conditions such as temperature or velocity.

The Renormalization Group (RNG) k-ε model [11] has been preferred to calculate in-cylinder turbulent flow for solution accuracy, simplicity, and effectiveness [12,13]. The Extend Coherent Flame Model, 3-Zone (ECFM-3Z Model) was selected for the combustion process inside the cylinder. ECFM-3Z model divide into three zones of unmixed, the mixed air and fuel zone, and unmixed fuel. Flame propagation of this model proceeds from the burned gas to the unburned gas section and takes into account the combustion modes of self-ignition, premixed flame, and diffusion flame [14,15]. Sub-models of the simulation are listed in table 2. Surfaces of the engine were prepared as a full engine with valves (fig. 2).

Table 2. CFD simulation Sub-models.

Turbulence model	RNG k-ε
Combustion model	ECFM-3Z
Evaporation model	Frossling
Collision model	NTC
Spray-wall interaction model	Rebound/Slide
Spray Breakup model	KH + RT
NOx model	Extended Zeldovich
Soot model	Hiroyasu

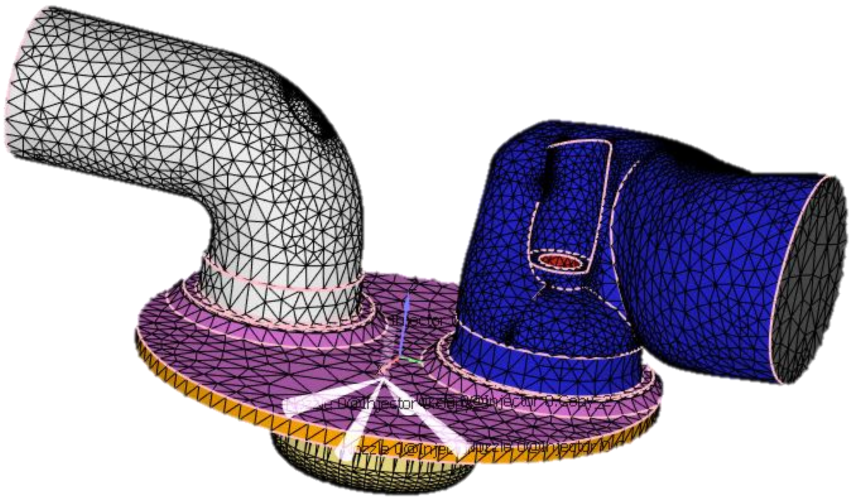

Figure 2. The surface of the CFD model at TDC with baseline piston bowl.

The mesh independence study was performed for the baseline piston bowl. Three different meshes were generated as coarse (4 mm grid size), medium (2 mm grid size), and fine (1.5 mm grid size) with the adaptive mesh refinement and the fixed embedding for cylinder, walls, the spray region. All operating conditions of three different meshes were the same except the grid size. In-cylinder pressures of the different meshes (i.e. 1.5, 2, and 4 mm) were compared in fig. 3. The in-cylinder pressure trends for the three meshes are very similar to each other. The 4-mm mesh produced inconsistent results at high pressures, but the 1.5-mm and 2-mm mesh results were very close. Therefore, the medium size mesh, which has about 1.2×10^6 cells, is considered for simulation in accordance with the results of the mesh independence studies.

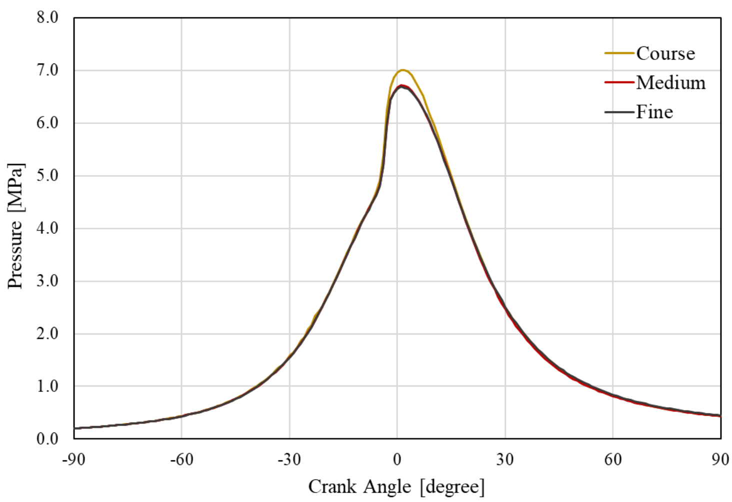

Figure 3. In-cylinder pressure curves of the mesh independency studies.

RESULTS AND DISCUSSION

The simulations have been performed at a constant engine speed of 2800 RPM and full load condition. In-cylinder pressure and temperature values for the base bowl, reentrant bowl, and swirl bowl were compared as shown in fig. 4, in order to understand the combustion process. It has been found that the in-cylinder pressure and temperature values of the swirl bowl is higher than the base bowl and reentrant bowl. This situation may be attributed to the better mixing and faster evaporation of fuel due to its bowl shape. The base bowl has the lowest in-cylinder pressures and temperatures. Since the in-cylinder pressures are directly linked to the performance of the engine, the swirl bowl will achieve higher power among three-piston bowl designs.

Due to the shape of the swirl bowl, the temperature can rise in the cylinder head. The fuel hits the convex surfaces, a large portion of the fuel moves upwards, and a high concentration of the fuel-air mixture is formed around the cylinder head. When the mixture burns, the temperature and the thermal load of the cylinder head increases. A similar scenario occurs in the reentrant bowl, but this effect is more significant for the swirl bowl (fig. 5). Better combustion efficiency results in higher temperatures. Therefore, it increases NO_X emissions for the swirl bowl. In addition, because of the soot-NOx trade-off, soot emissions are lower for the swirl bowl. Better mixing formation also reduces CO emissions. This situation causes by the conversion of CO emissions into CO_2 as the product of complete combustion (fig. 6).

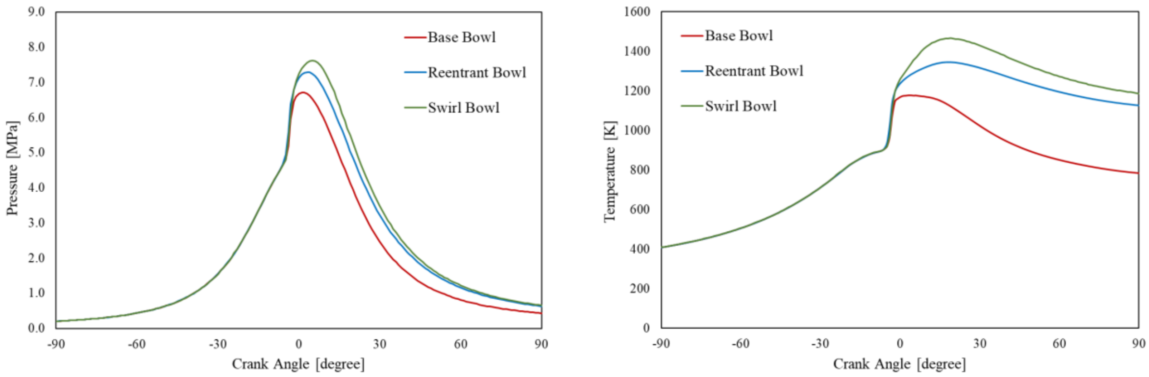

Figure 4. In-cylinder pressure and temperature curves for three different piston bowls.

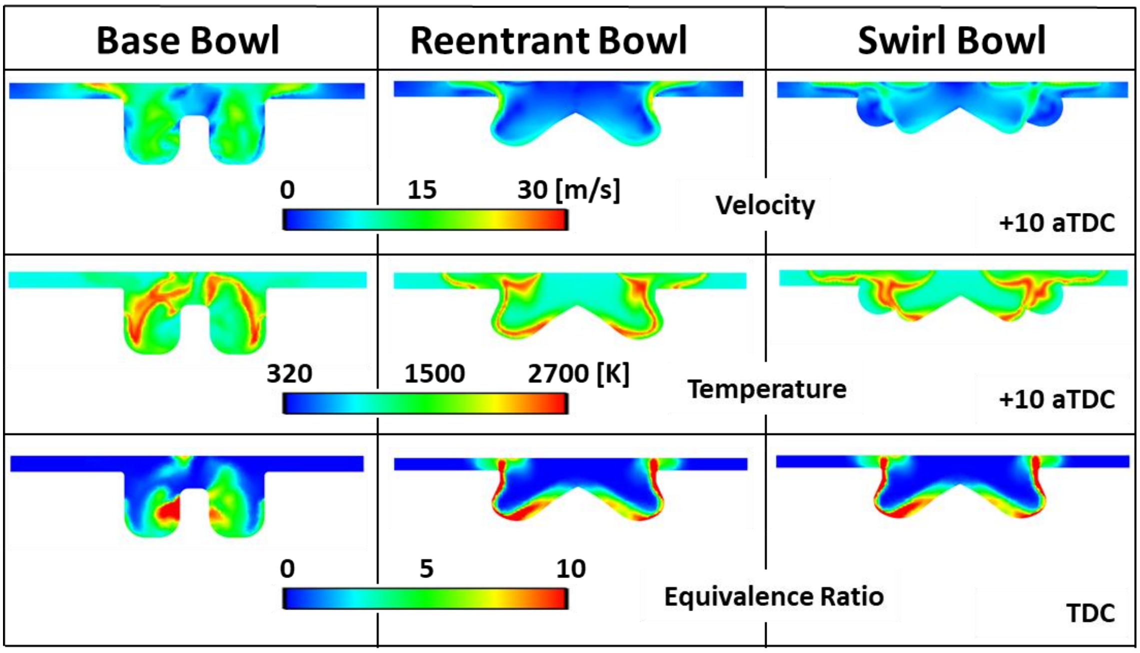

Figure 5. In-cylinder velocity, temperature, and equivalence ratio distributions of three different piston bowls.

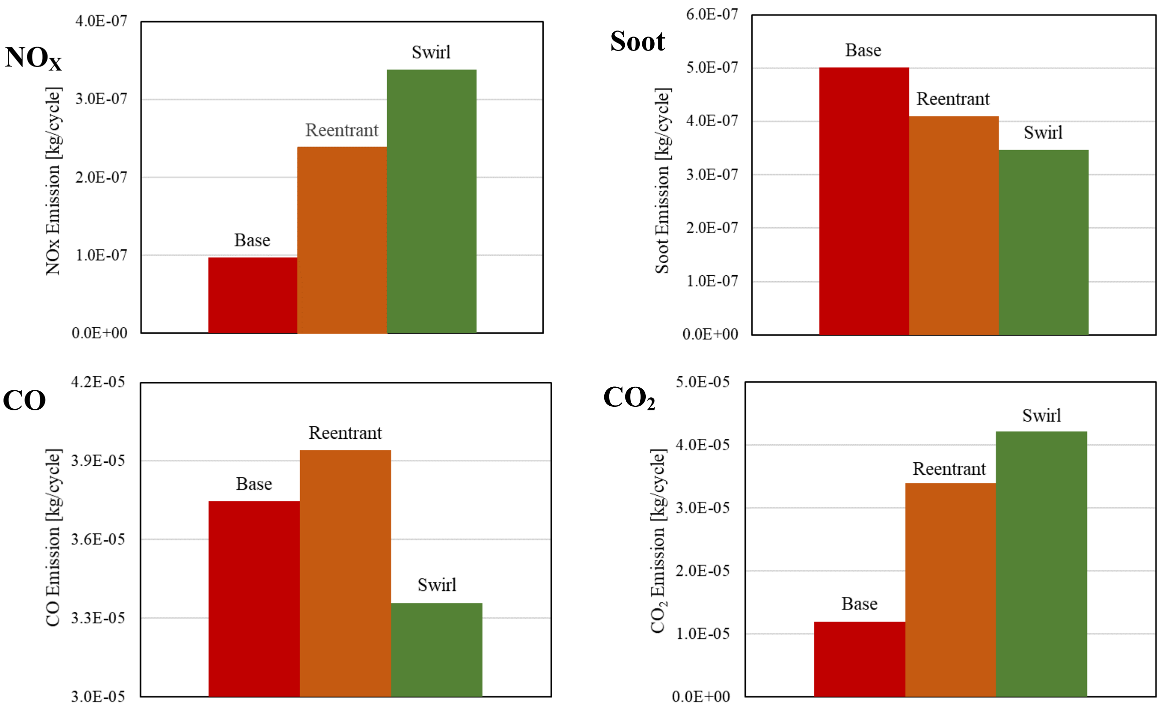

Figure 6. Exhaust emissions of three different piston bowls.

CONCLUDING REMARKS

In this study, the effects of the bowl shape on the combustion process and emissions are investigated using CFD simulation methods. The base, reentrant, and swirl piston bowls have been examined to consider replacing the base bowl. The swirl piston bowl improves the air/fuel mixing formation among three bowls in a direct injection diesel engine. These piston bowl geometries reduce the fuel-rich regions. Despite its simpler geometry, the swirl bowl achieves better results in the most present cases. The swirl bowl which guides the movement of the fuel inside the cylinder ensures high thermal efficiency and low soot emissions.

Based on the simulation results, it can be said that the swirl combustion chamber can be used especially for decreasing soot emissions in diesel engines. Significant improvements on exhaust emissions and combustion characteristics can be obtained by modifications such as optimizing the design parameters of the bowl and injection parameters.

Acknowledgment

The research was supported by the Marmara University Scientific Research Commission through the projects FEN-C-DRP-131217-0676 and CoHE faculty training program (YÖK ÖYP). We gratefully acknowledge the Convergent Science Company for providing the academic version of the CONVERGE CFD software.

NOMENCLATURE

AMR	Adaptive mesh refinement
aTDC	After top dead center
BDC	Bottom dead center
bTDC	Before top dead center
CA	Crank Angle
CFD	Computational Fluid Dynamics
RANS	Reynold Averaged Navier-Stokes
TDC	Top dead center

REFERENCES

[1] Broatch A, Olmeda P, Margot X, Escalona J. New approach to study the heat transfer in internal combustion engines by 3D modelling. International Journal of Thermal Sciences 2019;138:405–15. https://doi.org/10.1016/j.ijthermalsci.2019.01.006.

[2] Rajak U, Nashine P, Verma TN, Pugazhendhi A. Performance, combustion and emission analysis of microalgae Spirulina in a common rail direct injection diesel engine. Fuel 2019;255:115855. https://doi.org/10.1016/j.fuel.2019.115855.

[3] Shu J, Fu J, Liu J, Ma Y, Wang S, Deng B, et al. Effects of injector spray angle on combustion and emissions characteristics of a natural gas (NG)-diesel dual fuel engine based on CFD coupled with reduced chemical kinetic model. Applied Energy 2019;233–234:182–95. https://doi.org/10.1016/j.apenergy.2018.10.040.

[4] Sener R, Yangaz MU, Gul MZ. Effects of injection strategy and combustion chamber modification on a single-cylinder diesel engine. Fuel 2020. https://doi.org/10.1016/j.fuel.2020.117122.

[5] Benajes J, Novella R, Pastor JM, Hernández-López A, Kokjohn SL. Computational optimization of the combustion system of a heavy duty direct injection diesel engine operating with dimethyl-ether. Fuel 2018;218:127–39. https://doi.org/10.1016/j.fuel.2018.01.020.

[6] Turkcan A, Altinkurt MD, Coskun G, Canakci M. Numerical and experimental investigations of the effects of the second injection timing and alcohol-gasoline fuel blends on combustion and emissions of an HCCI-DI engine. Fuel 2018. https://doi.org/10.1016/j.fuel.2018.01.061.

[7] Lim SS, Vos T, Flaxman AD, Danaei G, Shibuya K, Adair-Rohani H, et al. A comparative risk assessment of burden of disease and injury attributable to 67 risk factors and risk factor clusters in 21 regions, 1990–2010: a systematic analysis for the Global Burden of Disease Study 2010. The Lancet 2012;380:2224–60. https://doi.org/10.1016/S0140-6736(12)61766-8.

[8] Su L, Li X, He X, Liu F. Experimental research on the diffusion flame formation and combustion performance of forced swirl combustion system for DI diesel engines. Energy Conversion and Management 2015;106:826–34. https://doi.org/10.1016/j.enconman.2015.10.027.

[9] Li X, Chen Y, Su L, Liu F. Effects of lateral swirl combustion chamber geometries on the combustion and emission characteristics of DI diesel engines and a matching method for the combustion chamber geometry. Fuel 2018;224:644–60. https://doi.org/10.1016/j.fuel.2018.03.063.

[10] Richards KJ, Senecal PK, Pomraning E. CONVERGE 2.4 Manual 2019:1078.

[11] Han Z, Reitz RD. Turbulence Modeling of Internal Combustion Engines Using RNG k-ε Models. Combustion Science and Technology 1995;106:267–95. https://doi.org/10.1080/00102209508907782.

[12] Huang H, Lv D, Zhu J, Chen Y, Zhu Z, Pan M, et al. Development and Validation of a New Reduced Diesel/ n -Pentanol Mechanism for Diesel Engine Applications. Energy & Fuels 2018;32:9934–48. https://doi.org/10.1021/acs.energyfuels.8b02083.

[13] Li Z, Wang Y, Geng H, Zhen X, Liu M, Xu S, et al. Parametric study of a diesel engine fueled with directly injected methanol and pilot diesel. Fuel 2019;256:115882. https://doi.org/10.1016/j.fuel.2019.115882.

[14] Taghavifar H, Mardani A, Mohebbi A, Taghavifar H. Investigating the effect of combustion properties on the accumulated heat release of di engines at rated EGR levels using the ANN approach. Fuel 2014. https://doi.org/10.1016/j.fuel.2014.07.073.

[15] Colin O, Benkenida A. The 3-zones Extended Coherent Flame Model (ECFM3Z) for computing premixed/diffusion combustion. Oil and Gas Science and Technology 2004. https://doi.org/10.2516/ogst:2004043.

[16] Frassoldati A, D'Errico G, Lucchini T, Stagni A, Cuoci A, Faravelli T, et al. Reduced kinetic mechanisms of diesel fuel surrogate for engine CFD simulations. Combustion and Flame 2015;162:3991–4007. https://doi.org/10.1016/j.combustflame.2015.07.039.

[17] Reitz RD. Mechanisms of Atomization Processes in High-Pressure Vaporizing Sprays. Atomization and Spray Technology 1987;3:309–37.

[18] Sánchez PS. Modeling the Dispersion and Evaporation of Sprays in Aeronautical Combustion Chambers. Institut National Polytechnique de Toulouse, 2012.

[19] Abramzon B, Sirignano WA. Droplet vaporization model for spray combustion calculations. International Journal of Heat and Mass Transfer 1989. https://doi.org/10.1016/0017-9310(89)90043-4.

[20] Sirignano WA. Fluid Dynamics and Transport of Droplets and Sprays. Cambridge University Press; 1999. https://doi.org/10.1017/CBO9780511529566.

[21] Amsden AA, O'Rourke PJ, Butler TD. KIVA-II: A Computer Program for Chemically Reactive Flows with Sprays. Los Alamos National Lab, 1989. https://doi.org/10.1007/BF01054829.