A comprehensive methodology for computational fluid dynamics combustion modeling of industrial diesel engines

Citation for published version (APA):

Lucchini, T., Della Torre, A., D'Errico, G., Onorati, A., Maes, N. C. J., Somers, L. M. T., & Hardy, G. (2017). A comprehensive methodology for computational fluid dynamics combustion modeling of industrial diesel engines. *International Journal of Engine Research*, 18(1-2), 26-38. <https://doi.org/10.1177/1468087416679570>

DOI:

[10.1177/1468087416679570](https://doi.org/10.1177/1468087416679570)

Document status and date:

Published: 01/02/2017

Document Version:

Accepted manuscript including changes made at the peer-review stage

Please check the document version of this publication:

- A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
- The final author version and the galley proof are versions of the publication after peer review.
- The final published version features the final layout of the paper including the volume, issue and page numbers.

[Link to publication](#)

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the "Taverne" license above, please follow below link for the End User Agreement:

www.tue.nl/taverne

Take down policy

If you believe that this document breaches copyright please contact us at:

openaccess@tue.nl

providing details and we will investigate your claim.

A Comprehensive Methodology for CFD Combustion Modeling of Industrial Diesel Engines

T. Lucchini¹, A. Della Torre¹, G. D’Errico¹, A. Onorati¹
N. Maes², L. M. T. Somers², G. Hardy³

¹Department of Energy, Politecnico di Milano, Italy.

²Technische Universiteit Eindhoven, The Netherlands.

³FPT Motorenforschung AG, Switzerland.

Abstract. Combustion control and optimization is of great importance to meet future emission standards in Diesel engines: increase of bmep at high loads and extension of the operating range of advanced combustion modes seem to be the most promising solutions to reduce fuel consumption and pollutant emissions at the same time. Within this context, detailed CFD tools are required to predict the different involved phenomena such as fuel-air mixing, unsteady diffusion combustion and formation of noxious species. Detailed kinetics, consistent spray models and high quality grids are necessary to perform predictive simulations which can be used either for design or diagnostic purposes. In this work, the authors present a comprehensive approach which was developed using an open-source CFD code. To minimize the pre-processing time and preserve results accuracy, algorithms for automatic mesh generation of spray-oriented grids were developed and successfully applied to different combustion chamber geometries. The Lagrangian approach was used to describe the spray evolution while the combustion process is modeled employing detailed chemistry and, eventually, considering turbulence/chemistry interaction. The proposed CFD methodology was first assessed considering inert and reacting experiments in a constant volume vessel, where operating conditions typical of heavy duty Diesel Engines were reproduced. Afterwards, engine simulations were performed considering two different load points and two piston bowl geometries, respectively. Experimental validation was carried out by comparing computed and experimental data of in-cylinder pressure, heat release rate and pollutant emissions (NO_x , CO and soot).

1 Introduction

High efficiency and reliability make Diesel Engines the favorite choice of heavy-duty applications for road transportation, agriculture and many industrial applications. Fulfillment of emission standards and, at the same time, fuel consumption reduction currently drive the design and development of new CI engines.

Possible improving areas are represented by turbocharging, air management, combustion and after-treatment systems [9, 30]. When focusing on combustion, currently efforts are focused on the increase of compression ratio and injection pressure [8]. In order to exploit all the advantages of such solutions, combustion chamber geometry optimization is also necessary and this requires a deep understanding of the complex interplay of the complex physical processes governing fuel-air mixing, flame propagation and formation of pollutants [29].

To this end, computational fluid dynamics is applied for combustion system design. However, to be predictive and massively usable for engine design, a CFD code must fulfill different pre-requisites with the most important ones being the capability to automatically generate the computational mesh and the availability of accurate and robust numerical methods. Furthermore, extensively validated physical models are necessary to describe spray evolution, combustion with turbulence chemistry interaction and pollutant formation [3, 2].

This work is focused on the development of a comprehensive methodology for CFD simulation of heavy duty Diesel engines using the Lib-ICE code, which is based on the OpenFOAM® technology. Suitable algorithms were developed to automatically generate spray oriented grids on the basis of engine geometry data. The proposed mesh structure minimizes numerical diffusivity where fuel-air mixing process takes place, creating the possibility to perform simulations with an acceptable grid size [19]. The spray is modeled with the Lagrangian approach, including suitable sub-models for turbulent jet atomization and secondary breakup [14]. Diesel combustion is assumed to be represented by a multiple number of diffusion flames evolving in the mixture fraction space with turbulence/chemistry interaction governed by the scalar dissipation rate. Suitable sub-models for prediction of pollutant emissions were also introduced and their coupling with the combustion was extensively discussed.

A comprehensive validation of the proposed approach was carried out considering two separate steps. The main objective of the first one was to verify the capability of the employed set of models to reproduce spray evolution and flame structure. To this end, experiments were carried out at the Eindhoven Technical University using a combustion vessel in which it is possible to reach ambient conditions typical of heavy duty engines at full load. The fuel is delivered by a single hole, large nozzle and a set of different operating conditions were used for a full characterization of the fuel-air mixing and combustion processes. Afterwards, engine simulations were carried out considering two different engine geometries, producing the same amount of power and NO_x for the two selected operating points. Validation is carried out by a comparison between computed and experimental data of in-cylinder pressure, heat release rate and pollutant emissions (NO_x , CO, soot).

2 Computational models

Fuel-air mixing simulations were carried out by using the Lib-ICE code, which is a set of libraries and solvers for IC engine modeling based on the OpenFOAM technology. Over the years it was successfully applied to simulation of spray and combustion in direct-injection engines [6, 17, 19]

2.1 Spray Model

Due to the relatively large nozzle sizes employed in Heavy Duty Engines, higher spray penetrations are expected and to achieve realistic results the CFD setup is crucial in terms spray sub-models, mesh size and turbulence. In particular, atomization and breakup spray sub-models regulate the droplet size evolution once they have left the nozzle with a consequent effect on mass and momentum transfer to the gas phase. Due to longer spray penetrations, Lagrangian and Eulerian phases are expected to interact over a larger portion of the computational domain. Hence, mesh size and structure must be carefully chosen due to the well-known grid dependency problem. Following previous works,[17, 18], separate models were applied to predict atomization and secondary breakup processes. This is expected to better reproduce the morphology and the evolution of sprays emerging from large nozzles. The Huh-Gosman model was used [14] for spray atomization: primary parcels (blobs) are injected into the computational mesh with the same nozzle diameter and their velocity is function of the injected mass flow rate profile. Both Kelvin-Helmholtz and turbulence induced breakup on the jet surface are taken into account by the model, describing the diameter reduction of the injected parcels as function of the characteristic atomization length and time scales L_a and τ_a whose values are computed at nozzle exit and then change according to the jet turbulence decay. As a consequence of the parent droplet diameter reduction, new droplets are created whose size is computed from a PDF distribution, whose properties follows the one of the expected turbulence length-scale spectrum. The atomization process is supposed to cease as soon as one of the following conditions is satisfied: parent droplet diameter lower than the atomization length scale, Weber number (We) lower than 40 or Ohnesorge number (Oh) greater than 2. Parent droplets are not subject to drag, evaporation and heat transfer. To better reproduce the atomization process, primary parcels velocity has the same direction of the nozzle axis. At the time the stripping process takes place, secondary droplets are deflected with a radial velocity $v_r = L_a \tau_a$, which takes into account both turbulence at the nozzle exit and its progressive decay when traveling downstream. This is expected to better predict the spray morphology resulting from atomization.

The model originally proposed by Pilch and Erdman was applied in this work to predict the secondary breakup process [28]. According to their approach, there is a maximum stable diameter D_s below which breakup does not take place. The value of D_s is affected in two ways: (i) the decrease of droplet Weber number because of the new smaller droplet diameter and (ii) the decrease in relative velocity between the droplets and the flow-field, due to the changes in droplet acceleration (as a results of the decreased droplet diameter). The droplet breakup occurs if the decrease in Weber number is greater than a critical value We_c , accounting for the viscous effects parameterized by the Ohnesorge number

2.2 Representative Interactive Flamelet (RIF) Combustion Model

This model is based on the laminar flamelet concept, assuming that the smallest turbulent time and length scales are much larger than the chemical ones and there exists a locally undisturbed sheet where reactions occur [25]. This sheet

can be treated as an ensemble of stretched counter-flow diffusion flames, called *flamelets*. The advantage of such treatment is that all reacting scalars only depend on the mixture fraction variable, Z , which is related to the local fuel-to-air ratio for non-premixed combustion. Hence, local chemical composition can be estimated from the Z field in the CFD domain, assuming that its sub-grid distribution can be represented by a β -pdf. To this end, transport equations for both Z and its variance $\widetilde{Z''^2}$ need to be solved. The Z transport equation includes a source term related to spray evaporation, while such effects are neglected in the mixture fraction variance equation since they do not significantly affect the computed results.

The local flame structure is defined by the flamelet equations for chemical species and enthalpy which are solved assuming unity Lewis number in the mixture fraction space [2] where effects of turbulence and flow field are grouped into the scalar dissipation rate term:

$$\chi_z = \widehat{\chi_{st,j}} \frac{f(Z)}{f(Z_{st})} \quad (1)$$

$f(Z)$ has an ln-profile [11], while scalar dissipation rate at stoichiometric mixture fraction conditions $\widehat{\chi_{st,j}}$ for each flamelet is computed as an average of the local values in each computational cell and accounting for flamelet marker distribution M_j :

$$\widehat{\chi_{st,j}} = \frac{\int_V M_j \chi_{st,l}^{3/2} \rho P(Z_{st}) dV'}{\int_V M_j \chi_{st,l}^{1/2} \rho P(Z_{st}) dV'} \quad (2)$$

where P is the β -pdf of the mixture fraction, whose parameters depend on mixture fraction and its variance [10]. In each cell $\chi_{st,l}$ is computed following the Hellstrom formulation [31]:

$$\chi_{st,l} = \frac{\chi}{\int_0^1 \frac{f(Z)}{f(Z_{st})} \tilde{P}(Z) dZ} \quad (3)$$

The chemical composition in each cell of the CFD domain is thus computed from mixture fraction and its variance distribution as follows:

$$Y_i(\vec{x}) = \sum_{j=1}^{N_f} M_j \int_0^1 Y_{j,i}(Z) P(Z, \widetilde{Z''^2}) dZ \quad (4)$$

In case a multiple number of flamelets is employed, M_j in Eqn. 4 represents the so-called *flamelet marker* field in the CFD domain [2, 5].

One of the advantages of the *RIF* model compared to other ones based on detailed chemistry is represented by the fact that the reaction-diffusion problem is solved in the mixture fraction space, which is approximated as a one-dimensional grid with a limited number of points (100-200). This drastically reduces the CPU time required for chemistry integration and makes possible to use large mechanisms (more than 100 species) with a better prediction of both combustion and pollutant emissions. A single flamelet was used to represent Diesel combustion and this choice was motivated by the very short ignition delays which are typical of the chosen operating conditions. The use of a single flamelet was also supported by a preliminary sensitivity analysis carried out by

the authors: no relevant changes in computed cylinder pressure and NO_x results were found when using up to 20 flamelets. At the beginning of the simulation, the temperature profile in the mixture fraction space is initialized from enthalpy balance assuming oxidizer temperature at $Z = 0$ and 380 K temperature at $Z = 1$:

$$h(Z) = (1 - Z) \cdot h(Z = 0) + Z \cdot h(Z = 1) \quad (5)$$

It is also possible to take the fuel evaporation into account during initialization but for the tested conditions in this work no significant changes were found and for this reason this effect was not considered.

2.3 Prediction of pollutant emissions

Suitable sub-models to estimate the main pollutant emissions formed during the combustion process were also implemented in the proposed framework. In particular, the possibility to predict CO, NO_x and soot was included. Carbon monoxide concentration is directly estimated from the flamelet domain. This choice is justified by the fact that, in the diffusive combustion process, CO is formed where the mixture is rich and then convected by flow and turbulence in lean regions where it is oxidized. Hence, CO mass fraction is evaluated consistently with 4 as follows:

$$Y_{CO}(\vec{x}) = \sum_{j=1}^{N_f} M_j \int_0^1 Y_{j,CO}(Z) P(Z, \widetilde{Z''^2}) dZ \quad (6)$$

NO_x emissions are considered to be only NO and such species is assumed to be formed with the Extended Zeldovich mechanism as follows:

Reaction rate constants are taken from [12] after verifying that suggested values from newer works produce very similar results. Incorporation of NO formation mechanism inside the RIF combustion model is still an open-issue since the time-scales for NO formation are much longer than the ones of the other species involved in the diffusive combustion process. Evolution of NO inside the cylinder is related to fuel and temperature distributions with the maximum concentrations expected in the high-temperature regions and where the mixture fraction is close to the stoichiometric value. To take such aspects into account, different approaches were proposed in the past [7] and three different methods were implemented in Lib-ICE and compared in this work.

- **Model 1:** A transport equation for NO mass fraction Y_{NO} is solved in the CFD domain and its reaction rate is directly taken from the flamelet domain presuming a β -PDF. Advantage of such approach is represented by its consistency with the RIF combustion model and the possibility to take in-cylinder mixture fraction distribution effects into account. Temperature stratification is not considered, since NO reaction rate depends only on the flamelet temperature profile. As a consequence, NO formation rate depends mainly on in-cylinder pressure: as soon as it starts to decrease

NO formation is reduced as well irrespectively of its local concentration and the in-cylinder temperature distribution.

- **Model 2:** A transport equation for NO mass fraction Y_{NO} is solved in the CFD domain with a source term computed in any cell accounting for the average temperature and species concentrations estimated from Eq. 4 except NO for which the local concentration is used. This model can take in-cylinder temperature distribution into account but, on the other hand, it does not consider turbulence/chemistry interaction. For this reason, it is expected that NO will form only in cells where the mixture fraction is stoichiometric and cell temperature is high enough.
- **Model 3:** The concentration of NO is directly estimated from Eq. 4. In such case, evolution of NO depends on flamelet temperature history and mixture fraction distribution. This model neglects the time-scales which are typical of NO formation: as a consequence of this, predicted NO concentrations are higher than the ones of the other tested approaches and also of experimental data as reported in [7].

2.3.1 Soot emissions

The semi-empirical model proposed by Lindstedt and co-workers [16] is used to estimate soot emissions: two transport equations for soot particle number density N_p and volume fraction f_v are solved, with source terms related to nucleation, coagulation, surface growth and oxidation processes as follows:

$$\dot{\omega}_{N_p} = \dot{\omega}_{inc} - \dot{\omega}_{coag} \quad (10)$$

$$\dot{\omega}_{f_v} = \dot{\omega}_{inc} + \dot{\omega}_{grow} - \dot{\omega}_{oxi,O_2} - \dot{\omega}_{oxi,OH} \quad (11)$$

Inception and surface growth source terms ($\dot{\omega}_{inc}$ and $\dot{\omega}_{grow}$, respectively) depend linearly on the soot precursor concentration which is considered to be acetylene, consistently with other papers in which the same model was successfully applied to Diesel combustion [4]. As it has been experimentally observed, the reactivity of the soot particles decrease in time; in this work, in a consistent way with the original work from Lindstedt [16], this aspect was accounted for by assuming the soot surface growth rate proportional to the square root of the specific surface area, S_{soot} . Coagulation of soot particles, $\dot{\omega}_{coag}$ is modeled using the normal square dependence [16]. Soot oxidation depends on O_2 and OH concentrations, following [16]. It is well known that the formation of soot is rather a slow process compared to the other species involved in the combustion chemistry, thus the fast chemistry assumption, solving f_v and N_p equations in the mixture fraction space might questionable when it is necessary to predict the effects of mixture distribution, injection pressure and combustion chamber geometry [15]. For this reason transport equations for the soot model are solved in the CFD domain and source terms are computed on the basis of the local species concentrations and thermodynamic conditions.

3 Experimental validation

Two different operating points, named *A75* and *Cruise*, were chosen as representative of the conditions of interest for the design of a heavy-duty truck engine.

Minimum fuel consumption is reached in the A75 point, while the Cruise load is typical of constant speed operation in highways. Details related to injection pressure, conditions at SOI, engine speed and bmep are illustrated in Tab. 1.

Table 1: Details about A75 and Cruise operating points.

		Cruise	A75
Engine speed	[rpm]	1200	1200
bmep	[bar]	9.5	19
EGR rate	[%]	8	3-6
Injection pressure	[bar]	900-1000	1100-1200
Density @ SOI	[kg/m ³]	30	45
Estimated T @ SOI	[K]	950	1000
Nozzle diameter	[μm]	~ 200	

Preliminary assessment and validation of the proposed methodology is necessary with well documented experiments in order to successfully apply it also to engine simulations. For this reason, constant volume conditions were first considered and, in particular, data sets from two different laboratories were used. Fuel-air mixing and combustion experiments carried out in the TU-Eindhoven (TU/e) vessel allowed to assess the proposed set of spray and combustion models at ambient conditions and with nozzle sizes which are typical of Heavy-Duty Diesel engines at start of injection (SOI) time. Measurements recently performed in the SANDIA Combustion Vessel and available in the context of the Engine Combustion Network ECN [22] were used to validate the soot model.

All the simulations were run using the standard $k - \varepsilon$ turbulence model whose $C_{\varepsilon 1}$ was slightly increased from 1.44 to 1.5 to better predict the penetration of fuel jets following the practice known as *round jet correction* [5, 32]. Diesel fuel in engine simulations was assumed to be n-dodecane, whose oxidation is computed by using the mechanism proposed in Chisty et al. [24] which was combined with the Zeldovich mechanism to compute NO. The mechanism has 58 species and 272 reactions. **The RIF model equations are solved using the finite volume method in the mixture fraction space which is discretized with 130 grid points, most of them located around the stoichiometric mixture fraction value.**

3.1 TU/e combustion vessel

Fuel jet experiments were conducted in a constant-volume vessel with optical access where it is possible to reach thermodynamic conditions which similar to those encountered in heavy-duty Diesel engines at full-load conditions [1]. A single hole common-rail injector is mounted at the center of one of the metal side-ports. The injector in this work has an orifice outlet diameter of 205 μm , a converging hole with k-factor 1.5. The high pressure n-dodecane jets can be visualized through sapphire windows with a diameter of 100 mm from all side ports of the combustion vessel. The vessel is equipped with a pressure transducer and three different line-of-sight diagnostic techniques were used in this study to obtain ensemble average high-speed recordings. High-speed liquid-

and vapor-phase fuel penetration were measured for non-reacting experiments using DBI and Schlieren, respectively. Further details about the employed optical techniques can be found here [20, 27]. The experimental results reported in this work were constructed using ensemble averages of at least 10 individual recordings. To determine the heat release rate, gas pressure measurements were conducted at 50 kHz using a Kistler model 6045A pressure transducer, placed in one of the upper corners opposite of the injector. Four different operating conditions were considered, displayed in Tab. 2. All them are well representative of the engine operating points displayed in Tab. 1 and they differ in terms of injection pressure. C1 and C2 are non-reacting, while for C3 and C4 the ambient oxygen concentration is 21%.

Table 2: Details of the operating conditions tested in the TU/e high-pressure vessel.

	C1	C2	C3	C4
Ambient density [kg/m ³]	40	40	40	40
Ambient temperature [K]	900	900	900	900
Ambient pressure [bar]	105	105	105	105
Ambient oxygen concentration [%by vol]	0	0	21	21
Fuel injection pressure [bar]	800	1600	800	1600
Estimated injection duration [ms]	5.2	4.9	5.2	4.9

Simulations were carried out using a three-dimensional cubical computational mesh whose volume is very close to the one of the TU/e vessel. Figs. 1(a)-(b) illustrate the structure of the computational mesh used in the simulations which intends to reproduce the topology that is commonly adopted to model Diesel engine combustion chambers. To better predict the fuel-air mixing process, local refinement was used in the region where the spray evolves, where mesh size ranges from 0.3 to 1 mm. The spray model presented in this work was extensively validated in [20], where a suitable methodology for a consistent comparison between computed and DBI experimental data of liquid penetration was also developed. In particular, a light scattering model was implemented by the authors following the recent methods suggested by Magnotti and co-authors [21]. Axial profiles of non-dimensional optical thickness τ^* were computed assuming the spray to be composed only of spherical liquid droplets. For the computation of the steady-state liquid length, experimental and computed optical thickness profiles were processed in the same way: a line is fitted through the decreasing computed τ^* profile along the injector axis and the intersection of that line with the abscissa represents the steady-state liquid length. Fig. 1(c) compares computed and experimental optical thickness maps for the C1 and C2 conditions. Steady-state spray penetration is correctly estimated by the simulations, despite its angle looks smaller compared to experiments.

For further details related to the computation of the liquid penetration value and its comparison with experimental data, the reader is referred to [20], where the effects of spray model constants on computed extinction profiles are presented in detail.

Consistently with Fig. 1(b), computed and experimental data of steady-

Figure 1: (a) Computational mesh used for fuel-air mixing and combustion simulations in the TU/e vessel; (b) computational mesh structure in the spray region ; (c) Left panel: 2-D optical thickness maps obtained with DBI experiments. Right panel: numerical reproduction of the optical thickness maps using simulated liquid spray data.

state liquid penetration agree well as it is illustrated in Fig. 2(a) where it is possible to see that variation of injection pressure from 800 to 1600 bar produces a slight increase of liquid length for the C2 condition. However, due to the much higher momentum transferred to the gas phase, C2 has a higher vapor penetration compared to C1 and this aspect is correctly predicted by the proposed CFD setup as it can be seen in Fig. 2 (b). The capability to reproduce the vapor distribution was also verified in Fig. 2 (c) for the C1 condition where computed contours of mixture fraction equal to 10^{-3} were superimposed to ensemble averaged Schlieren images of fuel vapor at different instants after the start of injection. Computations seem to overestimate the radial vapor diffusion and this aspect can be related to the employed turbulence model.

Figure 2: Spray model validation for the C1 and C2 conditions: (a) Comparison between experimental and calculated liquid penetration values; (b) Comparison between computed (dashed lines) and experimental (solid lines) data of vapor penetration; (c) Time sequence of ensemble averaged Schlieren experiments at C1 conditions. The contours represent a numerical mixture fraction threshold of 10^{-3} .

After a proper assessment of the spray model, combustion simulations were carried out for the C3 and C4 operating points. When using a single flamelet, combustion model validation can be performed only in terms of vessel pressure evolution and heat release rate profiles. **This is because, in each computational cell, the chemical composition is only related to mixture fraction value and its**

variance. Hence, after ignition, a diffusion flame will be established in the whole computational domain and it will be almost stabilized at the nozzle exit. For a proper prediction of the lift-off length, it is necessary to employ a multiple number of flamelets [5].

To remove the uncertainties related to material properties, thermal inertia and radiation, it was decided to compare normalized profiles of experimental apparent heat release rate (AHRR) and computed heat release rate (HRR) directly estimated from the chemical species reaction rate. Such comparison was performed after verifying that in simulations the ratio between the cumulative heat released by combustion and the injected fuel mass corresponded to the n-dodecane lower heating value. Figs. 3(a)-(b) illustrate a comparison between normalized HRR for both the C3 and C4 conditions. Ignition delay time is very short and, in agreement with experimental data, it is longer (0.32 vs 0.24 ms) for the C4 condition due the higher injection pressure which is probably responsible for producing scalar dissipation rate values greater than the extinction one for a longer time [2]. Experiments carried out with a smaller nozzle (0.9 mm) show the opposite trend in terms of injection pressure effect on ignition delay [6], to better understand this aspect further investigations are required concerning fuel-air mixing in the early part of the injection process and they will be matter of investigation in a future work. Despite injected mass flow rate profiles reach their steady-state value approximately at 0.25 ms, the rate of heat release has a progressive increase up to 3 ms then it stabilizes for the C3 condition while it starts decreasing for C4. The RIF model correctly captures these features which can be ascribed .

Figure 3: (a) Comparison between computed and experimental values of the normalized heat release rate for the C3 operating condition; (b) Comparison between computed and experimental values of the normalized heat release rate for the C4 operating condition;

3.2 SANDIA combustion vessel

Assessment and validation of the proposed methodology for soot prediction was carried out with experimental data from the SANDIA combustion vessel which is extensively studied in the context of the Engine Combustion Network [23]. The experimental setup is widely described in [26] and the so-called *Spray-A* exper-

iment was simulated where n-dodecane is delivered through a single-hole nozzle with a $90\text{ }\mu\text{m}$ diameter and K-factor equal to 1.5. Recently, diffused, back-illumination, extinction imaging (DBIEI) was used to acquire time-resolved images of soot optical thickness (KL) in n-heptane spray combustion experiments [22]. DBIEI maps of KL are processed to obtain the dimensional extinction coefficient K , which can then be related to the radial distribution of the soot volume fraction f_v [22]. Assuming an axi-symmetric jet, the evolution of the total amount of soot can be estimated directly from f_v . Two operating conditions, named O3 and T3 were chosen to test the soot model from Leung, Lindstedt and Jones implemented in Lib-ICE. Details of such conditions are provided in Tab. 3. They can be considered good candidates for a preliminary validation of the soot model before carrying out engine simulations because of they have different ambient temperature and oxygen concentration values.

Table 3: Simulated operating points in the Sandia combustion vessel.

	O3	T3
Oxygen concentration [% by vol]	21	15
Injection pressure [bar]	1500	1500
Ambient temperature [K]	900	1000
Ambient density [kg/m^3]	22.8	22.8
Injection duration [ms]	5	5

First, it was verified that combustion simulations reproduce correctly the experimental heat release rate. In Fig. 4(a) - (b) computed and experimental normalized HRR profiles are compared for the T3 and O3 operating conditions. Highest injection pressure and smaller nozzle size are responsible for a different development of the combustion process, reaching the steady state HRR very fast compared to the TU/e vessel. For both the tested conditions, the RIF model correctly reproduce the experimental trend in terms of ignition delay time and transition to steady *injection rate controlled* combustion mode.

The soot model was tuned on the O3 condition by slightly modifying only the pre-exponential factor of the oxidation reactions and using all other constant as originally suggested by Lindstedt [16]. Then the model capabilities were verified also for the T3 operating point, having a higher ambient temperature and lower oxygen concentration. Computed and experimental evolutions of soot mass as function of time are reported in Fig. 5 for both the considered conditions. Despite only one flamelet was used and, consequently, flame is attached to the nozzle, **the soot trend is reproduced fairly well for the tested conditions because of the following two separate reasons:**

- Acetylene, used as soot precursor species, is formed mainly in the very rich core of the jet which is located far from the nozzle;
- Soot transport equations are solved in the CFD domain and this makes its distribution more consistent with local flow and species concentration.

Despite the authors are aware of the well-known relation between soot distribution and lift-off length and the need to use multiple flamelets for a proper

Figure 4: (a) Comparison between computed and experimental values of the normalized heat release rate for the T3 operating condition; (b) Comparison between computed and experimental values of the normalized heat release rate for the O3 operating condition.

prediction of flame stabilization, they consider this setup suitable for conventional combustion simulations in Heavy Duty engines, where very short lift-off lengths are expected due to the high cylinder pressure and temperature values which are generally found.

Figure 5: Computed vs experimental evolution of soot mass as function of time for the Spray-A O3 and T3 operating conditions.

Finally, Figs. 6(a)-(b) display computed and experimental distributions of soot volume fraction f_v for the O3 and T3 operating conditions, respectively. Compared to O3, the increase of the ambient temperature and, at the same time, a reduction of the oxygen concentration in T3 are responsible for a higher amount of soot mass. Both experiments and simulations show in Fig. 6(b) that in the T3 condition there is a larger zone where soot is formed and also that maximum f_v values are higher. These results are very encouraging, since they

prove that combination of the Lindstedt semi-empirical and RIF model allow a rather good description of the soot structure within the flame and of the transient and steady evolution of the soot mass for operating conditions which are typical of conventional Diesel combustion, where the reactivity of the fuel is high. Particularly the latter is considered to be a significant validation, requiring an equilibrium condition among the four sub-mechanisms of soot formation and the occurring mixing and transport phenomena of gaseous species and soot. Further validation is necessary at lower oxidizer temperature and considering higher charge dilution, also to verify if the proposed setup based on a single flamelet is still valid.

Figure 6: Comparison between experimental (top) and computed (bottom) distributions of soot volume fraction f_v for the O3 operating condition (a) and the T3 operating condition (b).

3.3 FPT Cursor 11 Engine

After the model validation at constant-volume conditions, combustion simulations were carried out for the FPT Cursor 11 engine. Effects of combustion chamber geometry were evaluated at different operating conditions. In particular, two different layouts for the piston bowl were tested under the A75 and the Cruise load points. Piston bowl geometry details are provided in Fig. 7: the first one is named *O-Bowl* and presents the well-known Mexican Hat layout. The second one has a deeper bowl and it is called *H-Bowl*. It was designed to enhance the air entrainment inside the fuel spray with expected positive effects on combustion efficiency and pollutant emissions. Different cylinder heads and injectors were used: with the O-Bowl, the engine runs with a 0.5 swirl ratio and a 9-hole nozzle while H-Bowl was tested with a 8-hole nozzle and higher swirl (1.3). The compression ratio from the H-Bowl is 20.5 which is slightly higher than the one of the O-Bowl (20): for this reason, SOI time was adjusted in order to achieve the same break mean effective pressure (bmep) for any load with the two bowl configurations. Same NO_x levels were experimentally achieved from the A75 and Cruise load points, respectively. Details of the operating conditions tested are illustrated in Tab. 4.

In Fig. 7 the computational grids employed for the simulations are reported. They were generated automatically with the algorithm described in [19]. On the basis of user parameters related to main engine geometry data, combustion chamber layout and spray axis, a spray-oriented mesh is automatically

Table 4: Tested operating conditions in the FPT C11 engine using the *O-Bowl* and *H-Bowl* configurations.

Bowl type	H-Bowl	O-Bowl	H-Bowl	O-Bowl
Operating condition	A75	A75	Cruise	Cruise
bmep [bar]	19	19	9.5	9.5
NO/NO _{ref}	1.3	1.3	1	1
Relative air/fuel ratio λ	1.6	1.6	2.0	2.0
SOI [deg BTDC]	9.1	6.3	8.0	6.5
Injection pressure [bar]	1100	1200	900	1000
EGR [%]	3.0	6.0	8.0	8.0
Bore [mm]	128			
Stroke [mm]	144			

generated. Such grid layout is expected to minimize the numerical diffusivity with positive effects on the prediction of the fuel-air mixing, combustion and pollutant formation processes. Simulations were carried out in a sector of the combustion chamber (1/9 for the O-Bowl and 1/8 for the H-Bowl) and, to keep a fine enough mesh resolution in the whole domain, the grid is progressively refined in the polar direction. At TDC, the O-Bowl mesh has approximately 50000 cells while the 60000 cells are used for the H-Bowl. Simulation starts from IVC where a swirl motion was imposed assuming a wheel-flow velocity profile. Wall-heat transfer is modeled by using the Huh-Chang temperature wall functions [13]. The spray model setup was the same employed for the simulations carried out in the TU/e vessel.

Figure 7: Details of the computational grids used for the combustion simulations in the *O-Bowl* and *H-Bowl* combustion chamber layouts.

The validity of the combustion model was first verified for the A75 load condition and Figs. 8(a)-(d) report a comparison between computed and experimental data of in-cylinder pressure and heat release rate profiles for both the

tested piston bowl configurations. Figs. 8(a)-(b) show that calculated cylinder pressure trace matches well the experimental one for both the H and O bowl geometries. In the H-Bowl, fuel is injected later and to compensate this delay the injection pressure is increased by approximately 100 bar. The result of this is a more rapid growth of the heat release rate profile after ignition delay, as well as higher values during the part where HRR reaches almost a constant value. Both these features are well reproduced by the RIF model as it can be seen comparing Figs. 8(c) and (d).

Figure 8: A75 Load point. Comparison between computed and experimental in cylinder pressure profiles for (a) O-Bowl configuration; (b) H-Bowl configuration; comparison between computed and experimental in heat release rate profiles for (c) O-Bowl configuration; (d) H-Bowl configuration

In the Cruise condition, due to the reduced engine load, injection is shortened and the experimental heat release rate profile assumes a sort of triangular shape with the maximum value located shortly after the top dead center. The results for the H-bowl, presented in Figs. 9(b)-(d) are satisfactory and qualitatively similar to what was achieved for the A75 load point. Computed HRR ramp is steeper than the experimental one, but location and magnitude of maximum heat release rate is very well predicted as well as the in-cylinder pressure trace. O-bowl results in the Cruise load point are not satisfactory and they need to be further analyzed. Fig. 9(a) shows that computed cylinder pressure is underestimated and the peak value is located before where it was found in experiments. The comparison between calculated and experimental values of the HRR in Fig. 9(b) shows that combustion is very slow after auto-ignition.

To improve O-Bowl results, a deeper analysis will be carried out in the

Figure 9: Cruise Load point. Comparison between computed and experimental in cylinder pressure profiles for (a) O-Bowl configuration; (b) H-Bowl configuration; comparison between computed and experimental in heat release rate profiles for (c) O-Bowl configuration; (d) H-Bowl configuration

future by analyzing with both experiments and simulations both fuel-air mixing and combustion processes.

The effect of engine load on CO emissions is reported in Figs. 10(a)-(b) for the O-Bowl and H-Bowl configurations. For sake of completeness, the benefits of using the H-Bowl to reduce CO emissions are also reported for experiments and simulations in Fig. 10(c) where the non-dimensional ratio:

$$r_{CO} = \frac{CO_{H-Bowl}}{CO_{O-Bowl}} \quad (12)$$

is shown. It is possible to see that, under the A75 operating condition, the H-Bowl is capable of an approximately 90% reduction of CO emissions compared to O-Bowl. The reduction of CO emissions for the O-Bowl from A75 to Cruise load point is correctly estimated by simulations in Fig. 10(a) and the main reason for this trend seems to be the increase of the relative air-fuel ratio λ . Fig. 10(b) shows a different behavior for the H-Bowl configuration where levels of CO for the two load points are very similar and much less depending on the relative air/fuel ratio. Even, CO emissions for the A75 load point are lower than the ones reported for the Cruise condition. Fig. 10(b) shows that simulations predict the opposite trend. However, analyzing only Figs. 10(a)-(b) would lead to wrong conclusions in terms of the model capabilities to predict CO. Looking again at Fig. 10(c), it is possible to see the r_{CO} parameter for the Cruise load

point is correctly predicted and also for the A75 condition simulations predict a reduction of CO emissions by approximately 65% instead of 90% reported in experiments. From this investigation, it is possible to conclude that the proposed approach is able to predict the effects of piston bowl geometry on CO emissions. To further improve the computed results probably a more detailed study of flow at IVC is necessary for the H-Bowl configuration since combustion simulations reveal that probably the estimated in-cylinder turbulence is higher than the expected one.

Figure 10: (a) Comparison between normalized computed and experimental CO emissions for the O-Bowl condition; (b) Comparison between normalized computed and experimental CO emissions for the O-Bowl condition; (c) Comparison between computed and experimental r_{CO} values for the A75 and Cruise load points.

The influence of the approach chosen to predict NO_x is reported in Figs. 11. In particular, Fig. 11(a) reports the normalized NO values with respect to the maximum one and in Fig. 11(b) the relative error with respect to experimental data is shown. Results provided by Model 1 are not satisfactory: reduction of load for the H-Bowl corresponds to an increase of NO_x . The trend is instead captured for the O-Bowl conditions. Model 1 always underestimates the experimental NO_x values consistently with what was discussed when it was presented: it can take into account turbulence/chemistry interaction but, on the other hand, it neglects in-cylinder temperature distribution. This seems to be an important drawback mainly for the H-Bowl configuration. The trend of NO_x from Model 2 is in better agreement with experimental data since the reduction of NO_x from A75 to Cruise load is correctly predicted. Computed data are underestimated also for this model, and the reason for this seems to be related to the fact that NO reaction rate depends on average cell temperature and composition. Except for the O-Bowl case under the cruise load, where cylinder pressure is underestimated, results from Model 3 overestimates experimental data by a 20% factor, but they are the ones which better reproduce the experimental trend either in terms of NO variation as function of load and the piston bowl design. For this reason, Model 3 is probably the best for engine design purposes. Unfortunately because of an unexpected lack of available measurements, it was not possible to carry on a validation of the proposed soot model in the engine conditions.

Figure 11: (a) Comparison between experimental and computed normalized NO_x emissions from Models 1, 2 and 3; (b) Computed relative error in NO_x estimation from models 1, 2 and 3.

4 Conclusions

This work was focused on the development of a comprehensive methodology for the simulation of Heavy Duty Diesel Engines. To this end, sub-models for spray and combustion and mesh management were implemented in the Lib-ICE code and assessed, aiming at the definition of the best numerical approach to estimate soot, NO_x and CO emissions. Dedicated constant volume experiments carried out at conditions similar to those encountered in Heavy Duty engines using an equivalent single-hole injector, to verify the capability of spray and combustion models to correctly reproduce both air/fuel mixing and heat release rate. The soot model instead was validated on the basis of the measurements of soot distribution taken in the ECN Spray-A experiments. Afterwards, engine simulations were carried out.

The achieved results illustrate that the proposed methodology can be successfully applied for design of Heavy Duty Diesel engines, since it is capable to reproduce the effects of piston bowl geometry and operating conditions on both combustion and pollutant emissions. However, there are some open issues which require further investigation and will be matter of study in future works, namely:

- **Mesh structure and resolution:** the proposed algorithm for automatic mesh generation produces grids of acceptable size and quality, ensuring accurate results. Possible further improvements could be focused on a better control of the mesh resolution where spray evolves and ignition takes place.
- **NO_x model:** the main implications due the way the NO_x chemistry is solved were extensively discussed. However, among the tested alternative approaches, none of them seems to be capable to reproduce the exact NO_x levels, while the correct qualitative trend was observed only assuming fast chemistry.

References

- [1] BAERT, R., FRIJTERS, P., SOMERS, L. M. T., LUIJTEN, C., AND

- DE BOER, W. Design and Operation of a High Pressure, High Temperature Cell for HD Diesel Spray Diagnostics: Guidelines and Results. *SAE Paper 2009-01-0649* (2009).
- [2] BARTHS, H., HASSE, C., AND PETERS, N. Computational fluid dynamics modelling of non-premixed combustion in direct injection diesel engines. *International Journal of Engine Research* 1 (3) (2000), pp. 249–267.
- [3] BODE, M., FALKENSTEIN, T., CHENADEC, V. L., KANG, S., PITSCH, H., ARIMA, T., AND TANIGUCHI, H. A New Euler/Lagrange Approach for Multiphase Simulations of a Multi-Hole GDI Injector. *SAE Paper 2015-01-0949* (2015).
- [4] BOLLA, M., FARRACE, D., WRIGHT, Y. M., BOULOUCHOS, K., AND MASTORAKOS, E. Influence of turbulencechemistry interaction for n-heptane spray combustion under diesel engine conditions with emphasis on soot formation and oxidation. *Combustion Theory and Modelling* 18, 2 (2014), 330–360.
- [5] D’ERRICO, G., LUCCHINI, T., CONTINO, F., JANGI, M., AND BAI, X.-S. Comparison of well-mixed and multiple representative interactive flamelet approaches for diesel spray combustion modelling. *Combustion Theory and Modelling* 18, 1 (2014), 65–88.
- [6] D’ERRICO, G., LUCCHINI, T., ONORATI, A., AND G.HARDY. Computational fluid dynamics modeling of combustion in heavy-duty diesel engines. *International Journal of Engine Research* 16 (1) (2015), pp. 112–124.
- [7] FELSCH, C., GAUDING, M., VANEGAS, A., WON, H., LUCKHCHOURA, V., PETERS, N., HASSE, C., AND EWALD, J. Evaluation of Modeling Approaches for NO_x Formation in a Common-Rail DI Diesel Engine within the Framework of Representative Interactive Flamelets (RIF). *SAE Paper 2008-01-0971* (2008).
- [8] FUNAYAMA, Y., NAKAJIMA, H., AND SHIMOKAWA, K. A Study on the Effects of a Higher Compression Ratio in the Combustion Chamber on the Diesel Engine Performance. *SAE Paper 2016-01-0722* (2016).
- [9] HASHIMOTO, M., AOYAGI, Y., KOBAYASHI, M., MURAYAMA, T., GOTO, Y., AND SUZUKI, H. BSFC Improvement and NO_x Reduction by Sequential Turbo System in a Heavy Duty Diesel Engine . *SAE Paper 2012-01-0712* (2012).
- [10] HASSE, C. *A Two-Dimensional Flamelet Model for Multiple Injections in Diesel Engines*. PhD thesis, Technischen Hochschule Aachen, 2004.
- [11] HASSE, C. *Modellierung der Zündung und Schadstoffbildung bei der dieselmotorischen Verbrennung mit Hilfe eines interaktiven Flamelet-Modells*. PhD thesis, RWTH Aachen, 1997.
- [12] HEYWOOD, J. B. *Internal Combustion Engine Fundamentals*. McGraw-Hill, 1988.

- [13] HUH, K. Y., CHANG, I. P., AND MARTIN, J. K. A comparison of boundary layer treatments for heat transfer in IC engines. *SAE Paper 900252* (1990).
- [14] HUH, K. Y., AND GOSMAN, A. D. A Phenomenological Model of Diesel Spray Atomization. *Proceedings of the International Conference on Multi-phase Flows, Tsukuba, Japan* (1991).
- [15] KIM, T., AND KIM, Y. Interactive transient flamelet modeling for soot formation and oxidation processes in laminar non-premixed jet flames. *Combustion and Flame*. 162 (2015), 1660–1678.
- [16] LEUNG, K., LINDSTEDT, R., AND JONES, W. A simplified reaction mechanism for soot formation in nonpremixed flames. *Combustion and Flame*. 87 (1991), 289–305.
- [17] LUCCHINI, T., D’ERRICO, G., ONORATI, A., BONANDRINI, G., VENTUROLI, L., AND GIOIA, R. D. Development and Application of a Computational Fluid Dynamics Methodology to Predict Fuel-Air Mixing and Sources of Soot Formation in Gasoline Direct Injection Engines. *International Journal of Engine Research* 15 (5) (2014), pp. 581–596.
- [18] LUCCHINI, T., ETTORRE, D., D’ERRICO, G., BRUSIANI, F., BIANCHI, G., MONTANARO, A., AND ALLOCCA, L. Experimental and Numerical Investigation of High-Pressure Diesel Sprays with Multiple Injections at Engine Conditions. *SAE Paper 2010-01-0179* (2010).
- [19] LUCCHINI, T., TORRE, A. D., D’ERRICO, G., MONTENEGRO, G., FIOCCO, M., AND MAGHBOULI, A. Automatic Mesh Generation for CFD Simulations of Direct-Injection Engines. *SAE Paper 2015-01-0376* (2015).
- [20] MAES, N., DAM, N., SOMERS, B., LUCCHINI, T., D’ERRICO, G., AND HARDY, G. Experimental and numerical analyses of liquid and spray penetration under heavy-duty diesel engine conditions. *SAE International Journal of Fuel and Lubricants* (2016, In press).
- [21] MAGNOTTI, G., AND GENZALE, C. A Novel Approach to Assess Diesel Spray Models using Joint Visible and X-Ray Liquid Extinction Measurements. *SAE International Journal of Fuel and Lubricants* 8, 1 (2015), 167–178.
- [22] MANIN, J., PICKETT, L., AND SKEEN, S. Two-Color Diffused Back-Illumination Imaging as a Diagnostic for Time-Resolved Soot Measurements in Reacting Sprays. *SAE International Journal of Engines* 6(1) (2013), 1908–1921.
- [23] MEIJER, M., SOMERS, B., JOHNSON, J., NABER, J., LEE, S.-Y., MALBEC, L. M., BRUNEAUX, G., PICKETT, L. M., BARDI, M., PAYRI, R., AND BAZYIN, T. Engine combustion network (ecn): Characterization and comparison of boundary conditions for different combustion vessels. *Atomization and Sprays* 22, 9 (2012), 777–806.

- [24] CHISHTY, M., BOLLA, M., HAWKES, E., PEI, Y., HOOK, S. Assessing the Importance of Radiative Heat Transfer for ECN Spray A Using the Transported PDF Method. *SAE Int. J. Fuels Lubr.* 9, 1 (2016), 100–107.
- [25] PETERS, N. Laminar Flamelet Concepts in Turbulent Combustion. *21st Symposium on Combustion (Int)* (1986), pp. 1231–1250.
- [26] PICKETT, L. M., GENZALE, C. L., BRUNEAUX, G., L.-M. MALBEC, L.-M., HERMANT, L., CHRISTIANSEN, C., AND SCHRAMM, J. Comparison of diesel spray combustion in different high-temperature, high-pressure facilities. *SAE International Journal of Engines* 3, 2 (2010), 156–181.
- [27] PICKETT, L. M., KOOK, S., AND WILLIAMS, T. Visualization of diesel spray penetration, cool-flame, ignition, high-temperature combustion, and soot formation using high-speed imaging. *SAE International Journal of Engines* 2, 1 (2009), 439–459.
- [28] PILCH, M., AND ERDMAN, C. A. Use of Breakup Time Data and Velocity History Data to Predict the Maximum Size of Stable Fragments for Acceleration-Induced Breakup of a Liquid Drop. *International Journal of Multiphase Flow* 13 (6) (1987), pp. 741–757.
- [29] SINGH, S., REITZ, R. D., AND MUSCULUS, M. P. B. Comparison of the Characteristic Time (CTC), Representative Interactive Flamelet (RIF), and Direct Integration with Detailed Chemistry Combustion Models against Optical Diagnostic Data for Multi-Mode Combustion in a Heavy-Duty DI Diesel Engine. *SAE Paper 2006-01-0055* (2006).
- [30] YU, W., LIU, B., LI, Y., SU, Q., PEI, Y., AND SU, W. A Hybrid Combustion Control Strategy for Heavy Duty Diesel Engines Based on the Technologies of Multi-Pulse Injections, Variable Boost Pressure and Retarded Intake Valve Closing Timing. *SAE Paper 2011-01-1382* (2011).
- [31] T. Hellstrom. RIF implementation and testing. *Technical Report 01.07.1996-31.12.1996*, 1997.
- [32] Pope, S. B. An Explanation of the Turbulent RoundJet/Plane-Jet Anomaly. *AIAA Journal* 16(3) (1978), pp. 279–281.