17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

# ENCIT2018-0458
# ANALYSIS OF THE COMBUSTION PROCESS IN AN ENGINE ADAPTED WITH PRE-CHAMBER USING A ZERO DIMENSIONAL NUMERICAL MODEL AND A THREE-DIMENSIONAL MODEL

**Bruno Silva de Lima, brunosilvadelima@hotmail.com [1]**
**Kilder Campos Fagundes, kilder.fagundes@gmail.com [1]**
**Gabriel Racilan Campos Faria, bielracilan@gmail.com [2]**
**Miguel Humberto Barrientos Sandoval, miguel.barrientossandoval@gmail.com [1]**
**Carlos Eduardo Castilla Alvarez, ing.carlos.ufps@gmail.com [1]**

Universidade Federal de Minas Gerais, Departamento de Engenharia Mecânica, Universidade Federal de Minas Gerais, Belo Horizonte, Brazil

***Abstract**. The reduction in the emission of gases combustion products and the reduction of the specific consumption of fuel make an important part in the design of new internal combustion engines, aiming at protecting the environment. One of the methods of achieving this goal is the burning of poor mixtures, using a pre-chamber ignition system. In this work a zero dimensional mathematical model developed in in a Matlab® routine starting from mass and energy balance and a three-dimensional CFD model in commercial software were used to analyze the combustion of the engine. A comparison of the results between the two models made possible the analysis of the behavior of the zero dimensional model, identifying it as a strong tool to design and characterize the combustion process in a fast way and a low computational cost. On the other hand, although the three-dimensional model has a higher computational cost, it was possible to analyze the flow and propagation of the flame in regions of interest and complex engine geometries, as well as to analyze flame front parameters. In general, the models showed good agreement when it comes to overall engine parameters, and it is possible to identify the advantages and disadvantages of each one. It is up to the designer to analyze which tool should be used, depending on the purpose of the modeling and the design phase of the engine.*

*Keywords**: Numerical simulation, Internal combustion engines, Pre-chamber***

## 1. INTRODUCTION

Control of the emission of pollutants in internal combustion engines presents a particular challenge, since the point of operation for maximum efficiency coincides with the air-fuel ratio for maximum NOx emission, which is one of the most harmful contaminants to health. (Raine et al. (1995))

An efficient strategy for the control of emissions in spark ignition (SI) engines is the use of poor mixtures in the combustion chamber, since as identified by Heywood (1988), emissions in SI engines are sensitive to the air-fuel ratio.

According to the authors (Dunn-Rankin et al., 2011; Kerimov et al., 1978 and Uyehara et al., (1995)) for poor mixtures, with $\lambda$ greater than 1.0, excess air and lower combustion temperatures cause emissions of $NOx$ and $CO$ become very low, while emissions of $(HC)$ with $\lambda$ close to 1.1. Shrestha (1999) explains that the thermal efficiency of the engine also increases with the use of poor mixtures, and Ma et al. (2007) report that due to the reduction of heat losses and the increase of gross indicated fuel conversion efficiency.

As discussed earlier, burning of poor blends in spark ignition internal combustion engines brings many benefits. However, as shown by (Ayala et al., 2006, Getzlaff et al., 2007, Getzlaff, 2014, Wang et al., 1736 and F. Taylor, 1985) limitations are also present, such as the increase in cyclical variation in combustion, increase the energy required to initiate combustion and reduce the rate of flame propagation.

One solution to overcome the difficulties of burning poor blends is the prechamber ignition system (PCI). In the PCI system the combustion has its beginning in the pre-chamber, by means of an electric spark, where the increase of pressure causes the burning gases to be expelled towards the main chamber through the interconnection holes. According to Ryu (1987), each combustion mixture jet from the pre-chamber represents a high energy source for the ignition of the mixture inside the main chamber. Moreira (2014) shows that this ignition system can work with homogeneous or stratified load.

Rodrigues Filho (2015) explained that when working with homogeneous load, part of the mixture that feeds the main combustion chamber is pushed into the pre-chamber during the compression stroke, operating at the same air-fuel equivalence ratio, both in the pre-chamber and in the main combustion chamber. In ignition systems with stratified pre-chambers a small amount of fuel or air-fuel mixture is injected into the pre-chamber by means of an auxiliary system. This makes a richer mixture near the spark plug than in the main chamber.

The constructive characteristics of the pre-chambers exert a great influence on the operation of the internal combustion engine with ignition by flamethrowers, as several studies have shown (Ryu et al., 1987; Wall et al., 1978; Sakai, 1974; al., 2013 and Adams, 1978). Proper determination of the internal volume and diameter of the communication port with the main combustion chamber is critical for the engine to perform well. In addition, a theoretical study of pre-chamber characteristics and their effect on various engine variables can reduce design costs and deadlines by reducing the number of high-computation simulations and prototype machine versions.

Lima (2015) has shown that zerodimensional models are efficient for the calculation of global parameters such as temperature and pressure, but they present considerable errors in relation to velocity and mass distribution fields, due to the fact that these models consider the density evenly distributed in the system and only one component of the speed.

Braga (2015) points out in his work that the three-dimensional numerical simulations are fundamental for optimization of the project, since they generate complete results of the flow field. So it is possible to characterize the flow and then optimize it for better engine performance.

Thus zerodimensional models are used to analyze global properties, with a lower computational cost at the beginning of the project, and the three-dimensional tools are used in later phases of design for system optimization.

As discussed, the geometry of the pre-chamber becomes a very important variable in the ignition system with pre-chambers, for this reason, this work aims to make a comparison of a zerodimensional model and a three-dimensional CFD model to achieve a viable methodology in the pre-chamber design.

## 2. MATERIALS AND METHODS

In this chapter the models used for the zerodimensional and three-dimensional simulations of the engine will be presented. The equations for both models will be presented as well as the results to be compared.

The engine used for this simulations is the Ford Sigma 1.6 Flex, four-cylinder and 16-valve. This engine has a PFI injection and a compression ratio of 11: 1.

### 2.1. Zerodimensional Modeling

In this paper we will discuss the zerodimensional model constructed by Castilla et al. (2015) who proposed a methodology for calculating the geometry of the pre-chamber and discuss the impact of the same on the energy release capacity for the main combustion chamber. In their work they used the control volume shown in Figure 1. In this figure it is possible to observe that the pre-chamber is a control volume separate from the main chamber and these control volumes communicate through the area An.

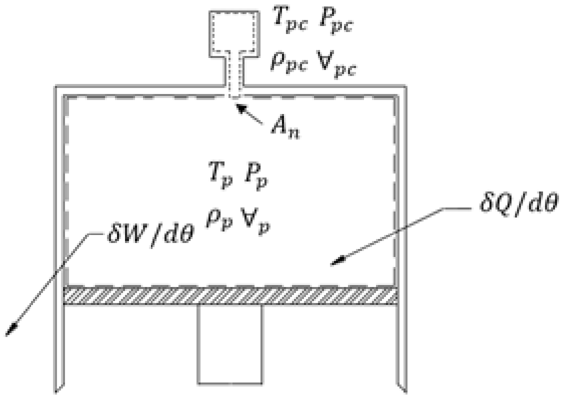


Figura 1. Control volumes and their main variables
Source: Castilla et al 2015

The analysis was restricted to the period between the closing of the intake valve and the opening of the exhaust, were not considered entry or exit of mass by the fuel injection or blow-by, thus considering only the mass of the two volumes without exchanges with the outside. Calculations start in PMI towards PMS and return to PMI when calculations close.

Gas passage through the orifice was modeled assuming the fluid as the ideal gas, specific heats were considered as a function of the temperature in each control volume and a zerodimensional isentropic flow through a restriction. The discharge coefficient used was 1.0, which means that the load losses due to restriction of the pre-chamber nozzle were not considered, so the effective area is equal to the nominal area.

The combustion in the combustion chambers is modeled, independently, from the Wiebe Law of an area. Due to the stop of the calculation after the energy release in the pre-chamber, the model disconsiders the heat exchanges with the wall.

The working fluid is supposed to obey the Law of Ideal Gases. In addition, the specific heats of the fluid are considered as a function of the temperature in each control volume.

Based on this methodology, Castilla et al. (2015) developed a code using the MATLAB program to solve the equations in the mentioned control volumes.

## 2.2. Three-dimensional Modeling

For the three-dimensional modeling the internal geometry of the engine was inserted in the commercial software Converge CFD, along with the initial and contour conditions necessary for the simulation. The SAGE chemical kinetics model was used to solve the combustion model.

The mesh used has a base size of 4mm which is recommended by the Converge user manual. Converge-CFD has a mesh optimization algorithm that automatically generates the mesh according to local parameters. This adapted mechanism was used in regions where the speed or temperature are higher. In addition to this mechanism, some regions have a fixed refinement criterion, such as the region of the ignition that is refined at the moment of spark, the valves are refined at the time of opening and closing, and the combustion chamber and the pre-chamber within the range of combustion occurs. (Converge CFD Version 2.3: CONVERGE Studio 2.3 Manual)

Computational Fluid Dynamics, CFD, is the analysis involving heat transfer, fluid mechanics, and other phenomena, such as chemical reactions, within a system. Mathematical modeling of differential equations that represent the physics of the system under study are used. (VERSTEEG & MALALASEKERA (2007))

Turbulent flow generates vortices in the fluid, which are responsible for more efficient energy, mass and momentum changes. The larger vortices withdraw energy from the stream and carry that energy into smaller vortices until it is converted into thermal energy. The methods that exist for the turbulence calculations and consequently the vortices can be grouped into three groups, DNS (Direct numerical simulation), LES (Large eddy simulation) and RANS (Reynolds-averaged Navier-Stokes equations). RANS has the lowest computational cost, this model was chosen for the simulations performed in this paper.

In order to simulate turbulent flows with the mean Reynolds and Navier-Stokes equations it is necessary to add the turbulent viscosity models to model the Reynolds tensions as well as the scalar transport terms and close the mean flow system. One of the most used and validated methods today are the k-models, they are based on the assumption that there is an analogy between the viscous tensions and the Reynolds tensions in the average flow.

According to Barros (2003), the modeling of chemical kinetics for a mixture of n chemical species ($A_i$), where occur simultaneously m reactions, the general equation that represents all the reactions of the kinetic mechanism is represented in Equation (1).

$$\sum_{i=1}^{n} v'_{ij} A_i \underset{k_{bj}}{\overset{k_{fj}}{\rightleftarrows}} \sum_{i=1}^{n} v''_{ij} A_i \tag{1}$$

The coefficients $v'_{ij}$ and $v''_{ij}$ are the stoichiometric coefficients of the reactants and the products, respectively, and $k_{fj}$ e $k_{bj}$ represent the reaction constants of the reaction j in the forward and reverse directions, respectively.

The fuel used was C gasoline, a combination of 27% anhydrous ethanol with the remainder of gasoline A. Thus it was necessary to make a combination between the mechanism of chemical kinetics of gasoline and that of ethanol. The reaction mechanisms were combined using the software Converge-CFD that allowed to combine and simplify the model. The final model contains 63 species and 380 reactions. The model was used from -60 up to 150 ° of crank angle, to save time during the simulation.

## 2.3. Model Comparison

We will present the results of the three-dimensional model for the simulation and then illustrate comparisons between the three-dimensional simulation and the results of the zerodimensional model.

In order to make comparisons, the initial and contour conditions used were the same for both models. The engine speed is 3500 rpm, Lambda 1.0, compression ratio 11: 1 and fuel E27.

The selected parameter for the comparison between the models will be the pressure curve in the cylinder. This parameter is shown in function of the crankshaft angle.

For the three-dimensional model it is possible to illustrate the flow and distribution of properties in the computational domain, so images of temperature and fuel concentration will be illustrated.

## 3. RESULTS AND DISCUSSION

This chapter will present the main results and the discussion of them. Firstly will be shown results of the three-dimensional model and thus the comparison between the two models.

Figure 2 shows the amount of fuel inside each chamber. It is important to note that the maximum amount of fuel inside the main chamber occurred just before closing the inlet valve, which closes at about -120$^o$ of crank, so the drop observed in the chart occurred due to the reverse flow from the combustion chamber to the admission. At about -90$^o$ of crank it has been observed that the pre-chamber filling with the fuel that was previously in the main chamber begins, so there is a small drop in fuel in the main chamber. The maximum amount of fuel in the pre-chamber takes approximately -10$^o$ of crank angle and represents 6.33% of the trapped fuel. The sudden drop of fuel in the pre-chamber is due to the fact that the combustion has started at -16$^o$, notice that even after the spark the chamber is still 6$^o$ with the pressure lower than the main chamber and thus continues receiving mixture with fuel. The small increase in fuel in the combustion chamber observed at approximately 0° of crank was due to the fact that the pressure increase in the pre-chamber pushed the air / fuel mixture into the main chamber. Soon after this fact the burning in the main chamber begins, a fact observed in the sudden drop in fuel quantity.

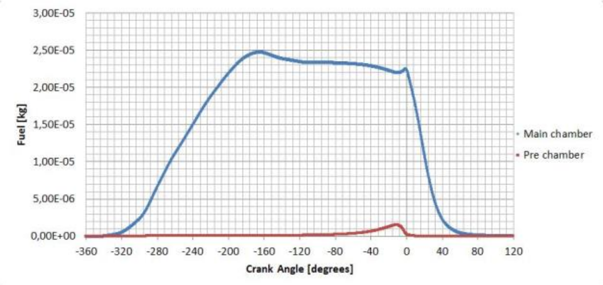


Figura 2. Fuel mass

Figure 3 illustrates the mass fuel concentration in the computational domain at the time of the spark. It is interesting to note that the concentration in the region of the head of the pre chamber is smaller than in the rest of the pre-chamber and combustion chamber, this is one of the design problems that can be observed only with three-dimensional simulation in future projects must be corrected so that the region of the spark plug is richer in fuel and facilitates the ignition of the mixture and decreases the cyclical variability of the combustion.

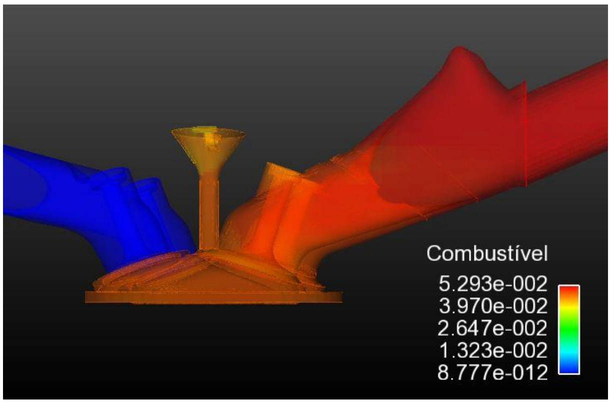


Figure 3. Fuel concentration

Figures 4, 5, 6 and 7 illustrate flame propagation in the domain. The white part represents the flame front, and the temperature in the domain is represented by the colors. Note that the flame front propagates through the pre-chamber and directly into the piston head. This flame front can cause pre-mature wear of the piston, so it must be corrected by redirecting the jet, or splitting it into several holes so that no part of the engine is compromised.

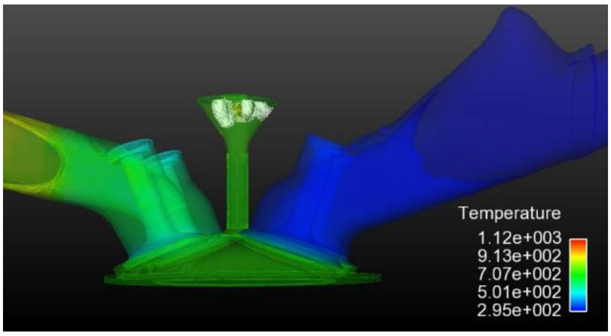


Figure 4. Flame front

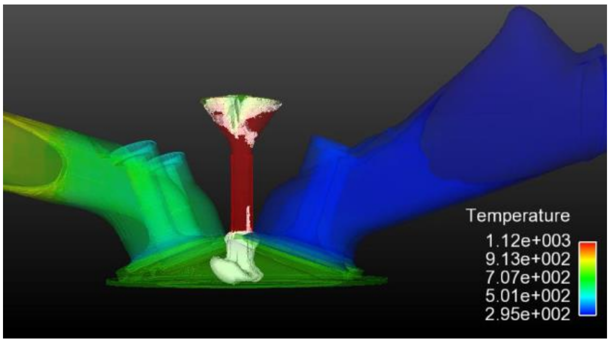


Figure 5. Flame front

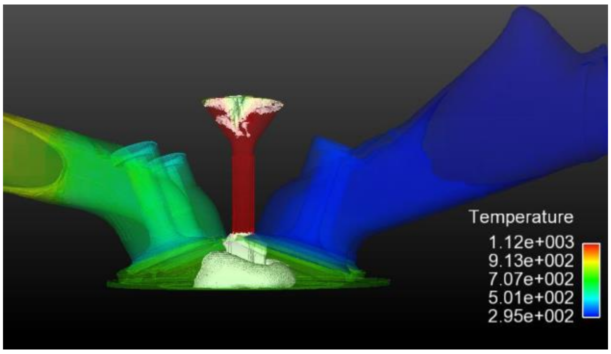


Figure 6. Flame front

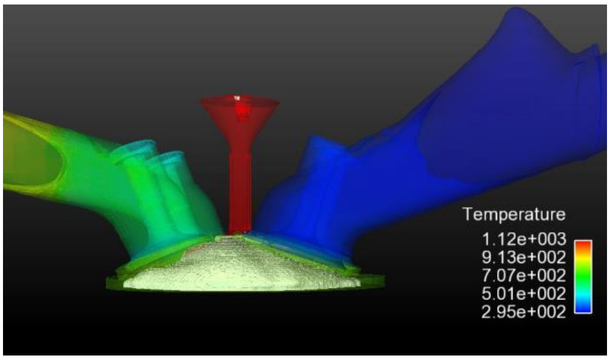


Figure 7. Flame front

A comparison is then made between the pressure calculated by the zero-dimensional and three-dimensional model, this comparison is illustrated in Figures 8 and 9, Figure 9 being a zoom in the region of the pressure peak. It is observed that the zerodimensional model was able to capture the phenomenon with high precision and with a computational cost much lower than the three-dimensional model. The zerodimensional model captured a pressure 11.25% higher than the pressure of the three-dimensional model, this fact was attributed to the heat transfer model of each model, the zerodimensional considers the system as adiabatic while the three-dimensional model has equations for calculating the transfer of heat.

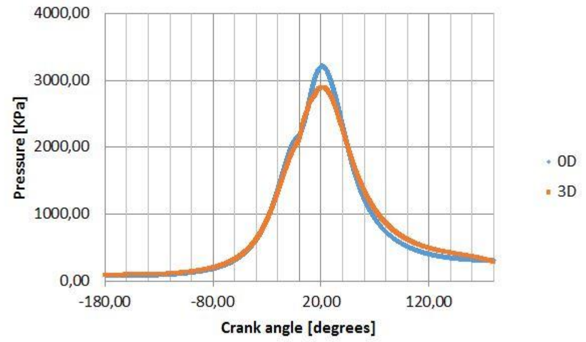


Figura 8. Pressure in main chamber for both models

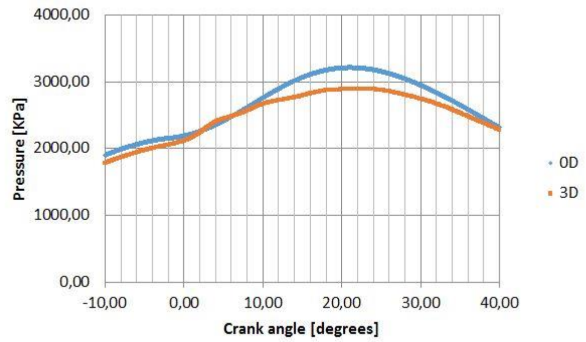


Figura 9. Pressure in pre chamber for both models

## 4. CONCLUSION

By analyzing the images generated by the three-dimensional model it was possible to draw some conclusions to assist in the next design of the pre-combustion chamber.

The fuel concentration image showed a deficiency in generating a higher concentration of fuel in the sail region, showing that a direct injection of fuel in the pre-chamber would be an advantage.

Another procedure that should be adopted in a future project is the direction of the jet. The pre-chamber with only one hole directed the jet to the center of the piston, this jet may be able to damage the same due to its high energy, so for a future project it must rethink the direction of the same and until using more of a hole.

The comparison of the models showed that the zerodimensional model, despite having captured a pressure greater than that of the three-dimensional model, was effective for this calculation because the difference between the models was small.

Thus, the zerodimensional model must be used for the calculation of general parameters in the initial phases of design, since it has a lower computational cost, of the order of minutes for each simulation, and the three-dimensional model when the objective is to optimize the project and its geometry in more advanced phases, because its computational cost is higher and is in the order of weeks for each simulation, considering the computers used in the laboratory.

## 5. REFERENCES

A. Jamrozik et al., "Numerical simulation of two-stage combustion in SI engine with prechamber," Applied Mathematical Modelling, vol. 37, no. 5, pp. 2961-2982, 2013.

Alvarez, E. Castilla, et al. 2015. Metodologia para o cálculo da pré-câmara de combustão de um motor de ignição por lança-chamas multicombustíve. Guayaquil : CIBIM, 2015.

BARROS, J. E. M. 2003. Estudo de Motores de Combustão Interna Aplicando Análise Orientada a Objetos. Belo Horizonte : Universidade Federal de Minas Gerais, Programa de Pós-Graduação em Engenharia Mecânica, 2003.

BRAGA, R.M. 2015. Análise numérica do escoamento de ar em um motor de combustão interna. Belo Horizonte : Departamento de Engenharia Mecânica da Universidade Federal de Minas Gerais, 2015.

C. F. Taylor, The Internal-combustion Engine in Theory and Practice: Combustion, fuels, materials, design: MIT press, 1985.

CONVERGENT SCIENCE. Converge Manual Converge CFD 2.3. [S.l.], 2016.

D. Dunn-Rankin, Lean combustion: technology and control: Academic Press, 2011.

F. A. Ayala, M. D. Gerty, and J. B. Heywood, Effects of combustion phasing, relative air-fuel ratio, compression ratio, and load on SI engine efficiency, SAE Technical Paper, 2006.

F. A. Rodrigues Filho, "Projeto, construção e caracterização do desempenho de um motor de combustão interna provido de um sistema de ignição por lança chamas de carga estratificada," 2014.

F. Ma et al., "Experimental study on thermal efficiency and emission characteristics of a lean burn hydrogen enriched natural gas engine," International Journal of Hydrogen Energy, vol. 32, no. 18, pp. 5067-5075, 2007.

H. Ryu, A. Chtsu, and T. Asanuma, Effect of torch jet direction on combustion and performance of a prechamber spark-ignition engine, SAE Technical Paper, 1987.

J. B. Heywood, Internal combustion engine fundamentals: Mcgraw-hill New York, 1988.

J. C. Wall, and J. B. Heywood, The Influence of Operating Variables and Prechamber Size on Combustion in a Prechamber Stratified-Charge Engine, 0148-7191, SAE Technical Paper, 1978.

J. Getzlaff et al., Investigations on Pre-Chamber Spark Plug with Pilot Injection, 0148-7191, SAE Technical Paper, 2007.

J. Getzlaff, "Erdgas als Kraftstoff für Kfz-Verbrennungsmotoren," 2014.

LIMA, B. S. de. Simulação Tridimensional do Escoamento a Frio em um Motor com um Modelo Unidimensional. 2015. Monografia (Engenharia Mecânica) — Universidade Federal de Minas Gerais.

N. A. Kerimov, and R. I. Mektiev, Engines with Stratified Charge, 0148-7191, SAE Technical Paper, 1978.

O. A. Uyehara, Prechamber for Lean Burn for Low NOx, 0148-7191, SAE Technical Paper, 1995.

R. Raine, C. Stone, and J. Gould, "Modeling of nitric oxide formation in spark ignition engines with a multizone burned gas," Combustion and flame, vol. 102, no. 3, pp. 241-255, 1995.

S. B. Shrestha, and G. Karim, "Hydrogen as an additive to methane for spark ignition engine applications," International Journal of Hydrogen Energy, vol. 24, no. 6, pp. 577-586, 1999.

T. Adams, Theory and evaluation of auxiliary combustion (torch) chambers, SAE Technical Paper, 1978.

T. MOREIRA, "ANÁLISE E CARACTERIZAÇÃO DE UM SISTEMA DE IGNIÇÃO POR LANÇA CHAMAS OPERANDO COM CARGA HOMOGÊNEA," Pós-Graduação em Engenharia Mecânica, Universidade Federal de Minas Gerais, Belo Horizonte, 2014.

T. Wang et al., "A study of ignition by flame torches." pp. 1729-1736.

VERSTEEG, H. W.; MALALASEKERA, W. An Introduction of Computational Fluid Dynamics. [S.l.]: Pearson, 2007.

Y. Sakai et al., Combustion characteristics of the torch ignited engine, 0148-7191, SAE Technical Paper, 1974.

## 6. Responsibility notice

The authors are the only responsible for the printed material included in this paper.