

UNIVERSITÀ DEGLI STUDI DI MODENA E REGGIO EMILIA

PhD school

International Doctorate “Enzo Ferrari” in Industrial
and Environmental Engineering

DEVELOPMENT OF METHODOLOGIES FOR HIGHLY COMPLEX FLUID DYNAMICS ANALYSES OF INTEREST IN MOTOR SPORT APPLICATIONS

Tutor:

Prof. Stefano Fontanesi

Candidate:

Mattia Olcuire

Co-Tutor:

Prof. Alessandro d’Adamo

PhD school director:

Prof. Alberto Muscio

Thesis index

Chapter 1. Abstract.....	3
Chapter 2. Multiphase flows theory	5
2.1. Introduction.....	5
2.2. The sloshing problem.....	9
2.3. Numerical approach	11
2.4. Volume Of Fluid model.....	12
2.5. Eulerian MultiPhase model.....	15
Chapter 3. CFD analysis of lubricant oil tanks sloshing of a high-performance car under racetrack maneuvers	18
3.1. Introduction.....	18
3.2. CFD methodology and setup	19
3.3. Simplified test case simulation	19
3.4. Current production oil tank for high-performance sports car: static simulation 26	
3.4.1. Experimental conditions	26
3.4.2. Sloshing simulation.....	28
3.5. Current production oil tank for high-performance sports car: dynamic simulation 36	
3.5.1. Experimental conditions	36
3.5.2. Sloshing simulation.....	38
Chapter 4. Reactive flows theory.....	44
4.1. Introduction.....	44
4.2. Turbulent Jet Ignition.....	45
4.3. The complexity behind combustion modeling in TJI systems.....	49
4.4. Combustion modelling.....	50
4.5. ECFM-3Z model.....	50
4.6. G-Equation model.....	51
4.7. Detailed Chemistry model	52

Chapter 5. CFD analysis of Turbulent Jet Ignition in a divided chamber bomb.....	53
5.1. Introduction.....	53
5.2. Experimental conditions	53
5.3. Numerical approach	55
5.4. Orifice Mesh sensitivity	58
5.5. Time-step sensitivity	60
5.6. Orifice chocking occurrence	62
5.7. Orifice diameter analysis	68
5.8. Chambers volume ratio analysis	77
Chapter 6. Conclusions and Further Work.....	82
Chapter 7. Bibliography	86
Chapter 8. Abbreviations	98

Chapter 1. Abstract

The aim of this thesis is to describe the workflow and the procedures to be followed in order to perform highly complex analyses in motor sport through the use of computational fluid dynamics (CFD). The described applications concern combustion propagation in modern Turbulent Jet Ignition (TJI) combustion engines and liquid sloshing in lubricant oil tank designed for high-performance sports cars. The activity has been developed at the “Enzo Ferrari” Engineering Department of Modena and Reggio Emilia University.

Focusing on the automotive field, and particularly on high performance car applications, Motor sport world demands reliable and robust responses in a very limited timeframe which is not always pursued by experimental investigations.

In order to be competitive, it's mandatory to look after every single detail of the performance. However, the term “performance” covers a wide range of situations and applications which make the achievement of competitiveness a challenging task. CFD is therefore a fundamental tool which comes in handy in this perspective in order to overcome the limitations of experimental studies in terms of response times and costs and to gain a far broader in-depth insight into the studied processes.

From the above considerations, in this work two main areas are analyzed in this perspective, both making use of CFD and compared to the experimental outcomes. The first one proposes a methodology to perform sloshing analyses of lubricant oil tank for high-performance sports cars under racetrack maneuvers. The latter is a detailed investigation on the turbulent combustion which takes place in turbulent jet ignition (TJI) systems widely adopted in motor sport applications. Although the described chapters are not intended to cover the multitude of applications available in motor sport, it is purposely reported in this thesis to show a typical *modus operandi* which can be worthwhile in this perspective.

Lubricant tanks designed for high-performance sports cars are usually fed by a mixture of oil and air which makes Volume of Fluid (VoF) models unsuitable for this kind of simulations. Hence, a different approach based on a Eulerian Multi-Phase (EMP) model is investigated and adopted. EMP considers interaction between gaseous and liquid phases,

accounting for mixing and separation which largely affect tank and lubricating circuit performance in high-performance sports cars applications, also allowing a massive reduction of computational time compared to VoF by reducing numerical constraints on mesh size and time-step. The methodology is initially validated against experimental data in a simplified tank with internal baffles and under pitch oscillation, whose data and geometry are available from literature. The effectiveness of the methodology is also validated against test-rig measurements on current production lubricating tanks for high-performance sports cars. A static application of an oil tank is initially performed, where the numerical results are compared against measurements of foam formation at the suction side of the feed pump. Finally, a dynamic application of an oil tank under racetrack accelerations is carried out, considering two full laps with the aim to reduce the dependance on the initial position of the free surface. Experimental data of the oil pump delivery pressure signal throughout the lap are compared to the numerical air percentage with reaches the feed pump suction side.

Turbulent Jet Ignition is seen as one of the most promising strategies to achieve stable lean-burn operation in modern spark-ignition engines. A nearly stoichiometric mixture is ignited in a small-volume pre-chamber, following which multiple highly reactive hot jets are discharged promoting rapid combustion in the main chamber. In the present work, a detailed computational investigation on the combustion regime of premixed rich propane/air mixture in a quiescent pre-chamber is performed to study the characteristics of the jet flame without the uncertainties in turbulent and mixing conditions typical of real-engine operations. Detailed chemical kinetics is integrated in the model setup and a kinetics-driven combustion modelling framework is used as well. The results are validated against the available experimental data, showing a remarkable alignment in terms of both combustion duration and jet outflow varying the chamber volume ratio and the nozzle diameter. Based on the above, the conducted investigation aims to offer a renewed procedure for combustion simulations under low Damköhler numbers, which typically concern pre-chamber combustors.

Chapter 2. Multiphase flows theory

2.1. Introduction

Multiphase flows are common practice in many industries, including process, power, aerospace, and transport. They are encountered in a large variety of equipment such as transportation lines, engine injection, stirred vessels, distillation, and bubble columns, lubricating and coolant systems. It is often the case that the performance of the equipment crucially depends on the existence of multiphase flow. Therefore, a deep understanding of multiphase flows is of primary importance to design and optimize many industrial processes and applications. The term “multiphase flow” covers a wide range of flows since they depend on the system through which they are flowing, the fluid properties, the flow speed and the relative volume fractions of different phases present at any point in the system, multiphase flows may occur in many different flow regimes. The Baker Chart [1], also described in [2] and [3], is shown in Figure 1 and shows a classification of flow pattern which might fall within the stratified, wavy, plug, slug, annular, bubbly, or dispersed flow.

- Stratified flow: a complete separation of the two phases can be observed, i.e. the velocities of both liquid and gas are low. The phases are clearly separated by a liquid-gas interface.
- Wavy flow: when the gas velocity increases in such a way that waves are formed on the free surface, the flow is called “wavy”. The higher the relative velocity of the phases, the higher is the amplitude of the waves.
- Plug flow: it is an intermittent flow that occurs at low gas flow rates and moderate liquid rates in such a way that liquid plugs are separated by zones of elongated gas bubbles.
- Slug flow: it occurs whenever the gas velocity increases under plug flow conditions, leading to small gas bubbles formation within the liquid slugs.
- Annular flow: the free surface is disturbed by small amplitude waves due to high gas flow rates. Liquid droplets can also be dispersed in the gas core.
- Dispersed flow occur whenever liquid droplets disperse within the gas phase.

- Bubble flow: it occurs when gas bubbles are dispersed in the liquid phase.

The Baker Chart axes might be represented based on the flow rates of the two phases, the total mass flux or the mass fraction of the phases. The dimensionless parameters λ and ψ make the chart usable for any liquid-vapor combination different from the standard combination where both equal unity and represent water and air flow under room temperature conditions. G [kg/m²/s] and L [kg/m²/s] represent the superficial mass velocity of the gas and of the liquid, respectively.

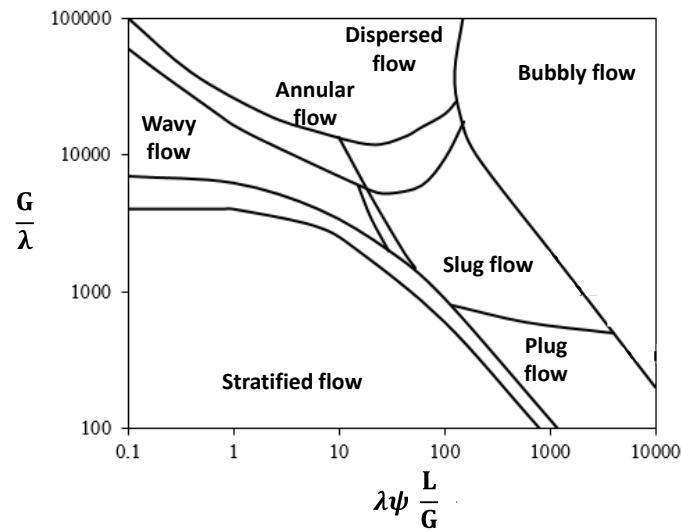


Figure 1: Classification of flow patterns using an example of Baker Chart

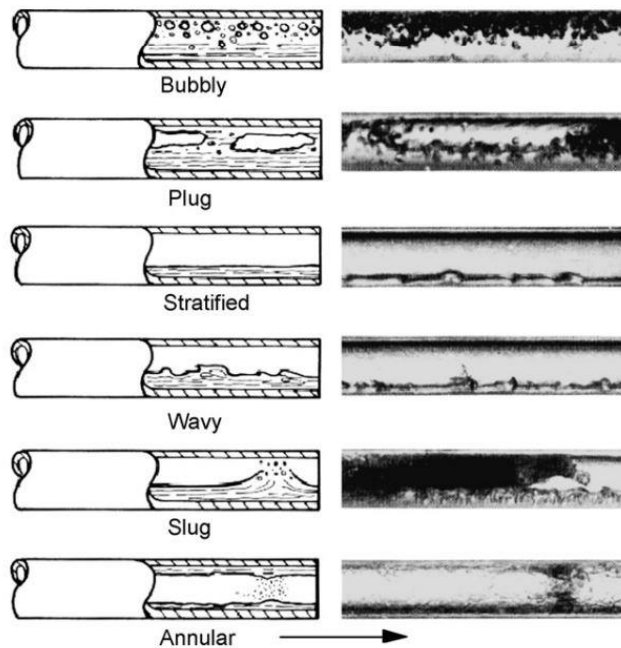


Figure 2 Two phase flow regimes [4]

Therefore, from a more macroscopic standpoint, multiphase flows can be topologically classified as separated (e.g., free surface flows or annular film flows in pipes), dispersed (e.g., droplet, bubbly and particle flows) or transitional [5, 6]. Such wide range of possibilities makes the design of industrial equipment for multiphase flow applications a challenging task, which is made even harder by the technical difficulty (and related cost) of obtaining experimental data under actual operating conditions.

Many applications of CFD in the process and associated industries are found in literature [7, 8, 9, 10]. CFD and related numerical methods are still under development in many ways and will continue to be so for the foreseeable future, benefiting from the complimentary use of experimental validation. Nevertheless, they are at present able to meet many of the requirements posed upon them. While the development of CFD methodologies for dispersed multiphase flows has been long focused on relatively low volume fractions ($< 10\%$) of the dispersed phase, many important industrial processes are carried out at high volume fractions. The prediction of these processes poses additional problems: firstly, the inter-phase momentum, heat and mass transfer as well as turbulence effects are not well understood; and secondly, the flow regime might change when the volume fraction of the dispersed phase is very high. This is particularly clear when addressing phase inversion and separation, which may occur either by design or by accident in many industrial processes. Phase separation is often employed to isolate products from unwanted by-products, e. g. in gravity settlers, distillation and bubble columns, lubricating oil tanks. Secondary effects such as phase inversion and the prediction of the entrainment of drops into the gas streams and vice versa are to be correctly represented since they are often unwanted. In both phase inversion and separating flows, regions where the originally dispersed phase becomes the continuous phase and vice versa exist simultaneously. Thin transition regions exist whenever phase inversion and separation occur, which are referred to as free-surfaces hereafter. At free-surfaces, rapid changes in the flow properties occur, which tend to cause singularities in numerical methods and are often troublesome.

In this perspective, among the many industrial problems where multiphase flows are found, liquid sloshing in storage tanks has been largely studied over the past decades for the analysis of a multitude of applications ranging from naval to aeronautics, from industrial processes to automotive [11, 12, 13]. It is a challenging task to study since it involves both fluid dynamics and the interaction of the fluid with the containing body. FSI (Fluid-Structure Interaction) is often used in the design of many engineering systems in the field

of transportation or structures with the aim of studying the mutual interplay of movable or deformable structures with an internal or surrounding fluid flow. Shelke and co-authors [14] compared different simulation techniques to analyze nonlinear random sloshing motions. In particular, they used VoF to map pressure distributions due to liquid sloshing on structural analyses of liquid tanks. Negrut and co-authors [15] analyzed the impact on vehicle dynamics of a fluid that sloshes in a tank attached to the vehicle via two-way coupled simulations. In the field of marine applications, studies performed by Brizzolara et al. [16] highlighted that the accurate prediction of running trim and sinkage, the calculation of lift force components and its longitudinal moment is fundamental in the simulation of planing hulls. However, even where the liquid mass is not relevant enough to influence the displacement of the body, the correct tracing of the interface motion is of primary importance, most of all when immiscible (or quasi-immiscible) fluids are involved. Jafari and co-authors [17] suggested that multi-phase applications may be approached using CFD in the field of blood transportation, using VoF model to assess the effect of the arteriole pressure drop on the capillary flow rate.

Krata and Jachowski suggested in [18] that CFD can be adopted in a regulatory framework to address bunker oil overflow in the design stage of onboard tanks, thus improving the environmental safety of maritime transportation. Xue and co-authors used CFD to address pressure distribution in membrane liquefied natural gas (LNG) tanks subject to sloshing dynamics [19]. In the field of wave energy converters, Chaplin et al. [20, 21] ran experiments to correlate rate of volume change of the free surface to the conversion efficiency. In the same field, the study of Zabihi et al. [22] provided an insight of wave reflection in energy conversion chain in offshore Oscillating Water Column (OWC). They also addressed free surface fluctuation inside the OWC chamber emphasizing on sloshing phenomenon through irregular wave impingement. Studies by Wei et al. [23] highlighted the importance of recognizing the free surface in ocean and coastal hydrodynamic phenomena such as breaking wave free surface and local bubbles formation, to mention a few. They also applied a deep learning method to segment the free surface according to the kinematic characteristics of free surfaces. With reference to the reactor design field, Lu H. et al. [24] carried out experimental and numerical investigations of sloshing behavior in fast reactor design, pointing out the importance of CFD simulation to analyze resonance phenomena under sinusoidal excitation. Again, in the field of fast reactors, studies by Cheng et al. [25] highlighted the importance of sloshing analyses in

capturing cooling sodium and molten fuel interaction to reduce severe accidents for sodium-cooled fast reactors. Liu Z. et al. [26] found out that sloshing excitations have great influence also on the thermal physical process in cryogenic storage tanks, promoting the mixing of vapor and liquid and enhancing heat exchange. As for aircraft wing design, studies by Martinez-Carrascal et al. [27] for a European airplane manufacturer highlighted the importance of aircraft fuel sloshing in damping wing vibrations. They also quantified separately the sloshing force and the damping force concluding that the fluid action can be modelled as a dual force with inertial and damping components. Saltari et al. [28] studied sloshing dynamics in flexible tanks to be embedded in larger aerospace structure that have substructures characterized by internal liquid movement. In the field of railroad transportation, Grossi et al. [29] analyzed both free and constrained oil sloshing oscillations inside railroad tank cars, highlighting the importance of using internal bulkheads and baffles for a more efficient oil height wave damping. With reference to the automotive field, Fontanesi et al. [30] investigated the use of automated mesh adaptation to limit the numerical diffusion issues in CFD simulations of a lubricant tank for high performance engine applications.

2.2. The sloshing problem

Focusing on the automotive field, and particularly on high performance car applications, the design of oil and fuel tanks must face several challenging issues, from the clear lubricating purpose up to emission-related challenges. Firstly, the correct estimate of the instantaneous position and shape of the free surface in both oil and fuel tanks is crucial to prevent partial or total uncovering of the suction section of feed pumps which could lead to sudden pressure drops in the fuel / oil circuits. Secondly, in modern lubricant oil tanks it is increasingly important to properly account for oil vapor formation due to high thermal loads and severe inertial accelerations concerning high performance engines. Oil vapor should be properly managed since it should be sent back up to the intake system without being released into the atmosphere according to current legislation. However, once the oil enters the intake line, it is well known its effect on HC emission release due to the oil burning within the combustion chamber. Such condition is detrimental for modern high-performance engines, mostly because of the increasingly strict emission regulations they

are going to deal with in the near future. Thirdly, the mixture entering the tank is usually a well dispersed air/oil mist or even a foam-like mixture. Therefore, it is of primary importance, in the design process of the tank, to seek solutions promoting phase separation upstream of the feeding pump, and by doing so avoiding pressure drops due to low-density mist/foam in the oil pump delivery pressure.

Considering that during the standard vehicle development plan, the engine, the oil tank and the vehicle itself meet usually close to the production phase, it is crucial to develop a robust numerical methodology in order to:

- Minimize oil tank overall dimension for a better integration within the vehicle.
- Be compliant with performance and emission targets, thus reducing the number of experimental campaigns to solve potential system anomalies.
- Minimize time-to-market of the oil tank.

As reported previously, tank sloshing dynamics is often a very difficult topic to investigate. The experimental investigation of this kind of applications as sloshing problems is extremely challenging and complex since it usually relies both on the visualization of the motion of the free surface by means of transparent models and on local pressure signals measured by transducers placed in different locations of the tank. In addition, experiments are useful as long as they are performed imposing accelerations and motions of the tank as close as possible to real ones. CFD is therefore widely adopted for the analyses of sloshing problems, since it allows a deeper understanding of the thermo-physical phenomena affecting sloshing dynamics with lower response times and costs than experimental practices. Thanks to CFD, it is possible to simulate the aforementioned phenomena such as interface tracing and detection, phase change, mixing and separation as well as fluid-structure interaction.

The thesis aims to describe an accurate 3D-CFD simulation methodology which allows to mimic the sloshing dynamics of a lubricant oil tank for high performance car applications under actual vehicle accelerations on a racetrack.

2.3. Numerical approach

A wide variety of alternative approaches exist to model complex multiphase flows within the academic and industrial community. Among them, methods are available which consider the interface as a sharp discontinuity and are referred to as Sharp Interface Methods (SIM). Most of them use a Lagrangian approach [31, 32, 33], in which the computational mesh is distorted and moved to follow the material interface. Eulerian methods also exist which rely on the level-set equation [34, 35, 36] to locate the interface. Relevant examples can be found in [37] with the Ghost Fluid Method, [38] with a simplified version of this method and [39]. Another class of SIM corresponds to Front Tracking methods, in which explicit tracking of the interface is performed over a fixed Eulerian mesh. Noticeable efforts have been paid to develop computational tools employing this approach [40]. Conversely from SIM, Diffuse Interface Methods (DIM) treat the interface as a numerically diffused zone. Pioneering examples of this approach can be found in [41]. Determination of thermodynamic flow variables in the diffused zones is achieved on the basis of multiphase flow theory. Saurel and co-authors applied such method to model cavitating flows and shocks in multiphase mixtures [42] as well as to model solid-fluid interactions in case of extreme deformations [43]. Jacqmin [44] applied DIM to a large-deformation sloshing flow in a two-dimensional rectangular domain. More recently, Dumbser [45] demonstrated the potentials of DIM through a thorough comparison of experimental reference data with the computational analyses of a large set of three-dimensional test cases. Tiwari [46] used a simple interface capturing scheme for the simulation of compressible multiphase flows, where the interfaces between different fluids are represented using field variables.

Radically different strategies based on Lagrangian particle schemes recently emerged for the analyses of multi-phase flows. Among them, the well-known Smooth Particle Hydrodynamics (SPH) method [47], [48], or the Particle Finite Element Method (PFEM). SPH is a so-called meshless method, i.e. it does not require any a-priori known connectivity between the moving points which are used to discretize the fluid. It is therefore particularly attractive in case of large and/or complex computational domains and whenever the physical time to be simulated is long. Examples of SPH applications to complex free surface flows can be found in [49, 50, 51, 52]. PFEM has made its first appearance in [53, 54, 55, 56]. Conversely from SPH, PFEM is a mesh-based Lagrangian scheme in which an

unstructured computational mesh is used by the underlying finite element method and it is updated at each computational time step using either an Arbitrary Lagrangian-Eulerian (ALE) approach, or, in case of large deformations, an Extended Delaunay Tessellation (EDT). To conclude this brief overview of existing approaches, and without the presumption to provide a complete survey of the existing techniques, methods exist which rely on linear dynamic analytical solutions [57] and models which attempt to provide simplified 0D/1D solutions to complex multiphase problems. Recent examples can be found in [54], where Tiwari and co-authors proposed a fast and computationally efficient approach to model a tank draining problem using the commercial tool GT-SUITE, allowing for the representation of a complete fuel line within a system-level framework. More recently, a similar approach was proposed for the simulation of a complete aircraft fuel system [59]. Mixed 1D/3D approaches can also be found, among which a recent application was proposed by Nichani and co-authors [60] to analyze the impact of engine oil pan sloshing on the performance of a lubrication pump of an automotive engine subject to typical vehicle longitudinal and latitudinal accelerations.

2.4. Volume Of Fluid model

The majority of CFD applications concerning oil tank sloshing start from the assumption that air and liquid are immiscible fluids; therefore, a sharp interface separating the fluids can always be assumed. Such applications are traditionally investigated using the Volume Of Fluid (VoF) approach [61, 62, 63, 64].

Such model assumes that the two phases are immiscible and is capable to resolve the interface between the phases of a mixture. Therefore, there is no need for extra modeling of inter-phase interaction and the assumption that all phases share the same velocity, pressure and temperature field all over the free surface becomes a discretization error. On the basis of such assumption, the Navier-Stokes equations are solved in the same way as for single-phase fluids.

Both the distribution of each phase inside the computational domain and the motion of the interface are described by means of a transport equation for the phase volume fraction α_i defined as follows:

$$\alpha_i = \frac{V_i}{V} \quad \text{Equation 1}$$

where, V_i is the volume of phase i in the cell and V is the volume of the cell.

The sum of volume fractions of all phases in each computational cell must sum up to one:

$$\sum_i \alpha_i = 1 \quad \text{Equation 2}$$

The fluids present in the same interface-containing cell are treated as a mixture and each property is calculated from the volume fraction weighted average:

$$\rho = \sum_i \rho_i \alpha_i \quad \text{Equation 3}$$

$$\mu = \sum_i \mu_i \alpha_i \quad \text{Equation 4}$$

$$c_p = \sum_i \frac{c_{pi} \rho_i}{\rho} \alpha_i \quad \text{Equation 5}$$

Where ρ_i , μ_i and c_{pi} are respectively the density, the dynamic viscosity, and the specific heat of phase i .

The volume fraction transport equation is driven by the phase mass conservation equation:

$$\frac{\partial}{\partial t} \int_V \alpha_i dV + \oint_A \alpha_i \bar{v} \cdot d\bar{a} = \int_V \left(S_{\alpha_i} - \frac{\alpha_i}{\rho_i} \frac{D\rho_i}{Dt} \right) dV - \int_V \frac{1}{\rho_i} \nabla \cdot (\alpha_i \rho_i \bar{v}_{d,i}) dV \quad \text{Equation 6}$$

where $\bar{a}$ is the surface area vector, $\bar{v}$ is the mixture (mass-averaged) velocity, $\bar{v}_{d,i}$ is the diffusion velocity, S_{α_i} is a source term of phase i , and $\frac{D\rho_i}{Dt}$ is the Lagrangian derivative of the phase densities ρ_i .

The continuity equation takes into account of a mass source term related to the phase source term and is given by:

$$\frac{\partial}{\partial t} \int_V \rho dV + \oint_A \rho \bar{v} \cdot d\bar{a} = \int_V (\sum_i S_{\alpha_i} \rho_i) dV \quad \text{Equation 7}$$

The momentum equation is given by:

$$\begin{aligned} \frac{\partial}{\partial t} (\int_V \rho \bar{v} dV) + \oint_A \alpha \rho \bar{v}_i * \bar{v}_i \cdot d\bar{a} = & - \oint_A p \bar{I} \cdot d\bar{a} + \oint_A \bar{T} \cdot d\bar{a} + \int_V \rho \bar{g} dV + \\ & \int_V \bar{f}_b dV - \sum_i \int_A \alpha_i \rho_i \bar{v}_{d,i} * \bar{v}_{d,i} \cdot d\bar{a} \end{aligned} \quad \text{Equation 8}$$

Where p is the pressure, $\bar{I}$ and $\bar{T}$ are respectively the unity and stress tensors and $\bar{f}_b$ the body forces vector.

Thanks to its numerical efficiency, the model is suited for simulations of flows where each phase represents a larger structure with a relatively small contact area between phases. Hence the need to use a mesh resolution sufficient to resolve the position and the shape of the interface between phases.

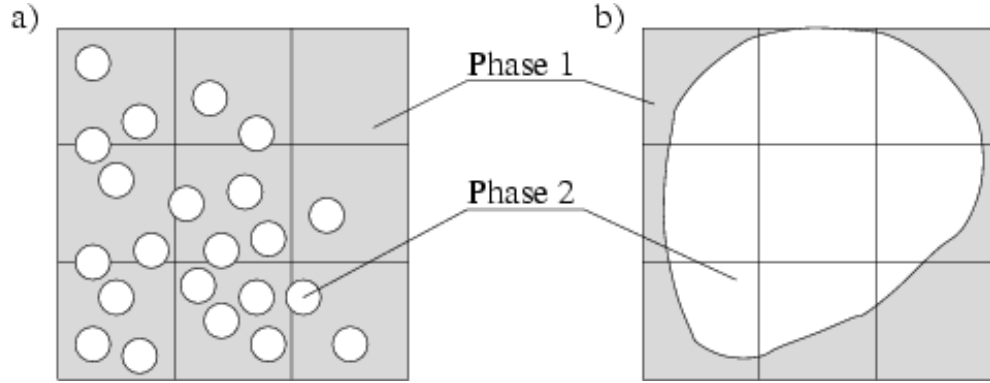


Figure 3: Unsuitable grid on the left; suitable grid on the right when using VoF approach

Figure 3 provides an example of the grid finesse required when using VoF approach. As it can be seen, a sufficient number of cells must be used to correctly represent the interface and all important flow features, also resolving flow separation and shed vortices. Whenever this requirement is not fulfilled, it would lead to loss of information of the position and shape of the free surface and, therefore, numerical diffusion. Likewise, it is also crucial to use appropriately low time steps for a robust representation of the interface motion. The rule-of-thumb is to maintain the local Courant-Friedrichs-Lewy number [65] below 0.5 over the free surface. Whenever physical time to be covered is long, such as for the simulation of multiple track laps for the presented application, computational cost grows dramatically despite the adoption of devoted strategies such as adaptive time-step and localized mesh refinements. Furthermore, the VoF approach could be intrinsically unsuitable in this specific investigation considering that a fully mixed air/oil mixture enters the tank. It is therefore crucial to adopt a model able to handle the separation between the phases, as well as possible foam formation. For these reasons, the Eulerian MultiPhase (EMP) model is proposed as an alternative candidate.

2.5. Eulerian MultiPhase model

EMP treats each phase as inter-penetrating continua since it solves the governing flow equations everywhere in the simulation domain, with volume fraction also being solved to account for changes in the amount of each phase. This approach is well-suited to a broad range of multiphase flows and is commonly used for modeling dispersed flows. Phases are considered to be mixed on length scales smaller than the length scales to resolve and they coexist everywhere in the flow domain. As in the VoF framework, a volume fraction of each phase is defined. Conservation of mass and momentum are reported in the following equations:

$$\frac{\partial}{\partial t} \int_V \alpha_i \rho_i dV + \oint_A \alpha_i \rho_i \bar{v}_i \cdot d\bar{a} = \int_V \sum_{j \neq i} (m_{ij} - m_{ji}) dV + \int_V S_i^\alpha dV \quad \text{Equation 9}$$

$$\begin{aligned} \frac{\partial}{\partial t} \int_V \alpha_i \rho_i v_i dV + \oint_A \alpha_i \rho_i \bar{v}_i \cdot d\bar{a} = & - \int_V \alpha_i \nabla p dV + \int_V [\alpha_i \rho_i \bar{g} + \bar{M}_i + \bar{F}_i^{int} + \\ & S_i^\alpha + \sum_j (m_{ij} - m_{ji})] dV + \oint_A [\alpha_i (\bar{T}_i + \bar{T}_i^t) \cdot d\bar{a} \end{aligned} \quad \text{Equation 10}$$

Where α_i , ρ_i , v_i are the volume fraction, the density and the velocity of a phase i , m_{ij} is the mass transfer rate to phase i , S_i^α is a user-defined source term, p is the pressure, $\bar{T}_i$ and $\bar{T}_i^t$ are the molecular and turbulent stresses, $\bar{M}_i$ is the interphase momentum transfer per unit volume, $\bar{F}_i^{int}$ represents internal forces. Transport equations for mass, momentum and energy are solved for each phase while a common pressure field is shared. The conservation equations for each phase variable require closure by the definition of phase interactions at each phase interface. Consequently, this method does not strictly require the use of very low time steps or very fine computational grids. Moreover, the model can simulate phase separation and foam formation according to local inertial forces. Phase interaction is controlled by local interaction area density and local drag force. The former is computed as the interfacial area available for drag and mass transfer between each pair of interacting phases. The latter is calculated through a linear multiplier of the relative velocity between the two phases. The governing equations are given below:

$$a = \frac{6\alpha_d}{l} \quad \text{Equation 11}$$

$$F^D = A_D v_r = \frac{1}{2} C_D \rho |v_r| \frac{a}{4} \quad \text{Equation 12}$$

In the first equation, which is based on the surface area of a spherical particle, a is the interaction area density, α_d the volume fraction for the dispersed phase in the interacting phase pair, l the interaction length scale. As for the second equation, F^D represents the drag force, A_D the linearized drag coefficient, v_r the relative velocity between the phases and C_D the drag coefficient, which in turn is calculated by the models of Tomiyama [66] and Schiller-Naumann [67]. The interaction length scale should be representative of the mean particle size, i.e. the size of the air bubbles dispersed in oil and the oil droplets dispersed in air. Since the evaluation of these quantities is a challenging estimation and it is strongly related to the application concerned, in the present investigation an original formulation is adopted. Interaction length scales for both phases are handled with a volume fraction-based approach, as shown in Figure 4. As it can be seen, constant values are considered for volume fractions higher than 10% while a linear decrease down to zero is applied for lower volume fractions. This approach is adopted to limit the relative velocity between the phases in quasi-single-phase regions, which promote numerical instabilities and may lead to numerical divergence. The adopted strategy considerably improves the stability of the simulations without affecting the results. Following sensitivity analyses (under confidentiality agreement with a collaborating entity), it is also negligibly affected by the slope of the initial ramp, i.e. by the volume fraction threshold below which a linearly decreasing behavior is set.

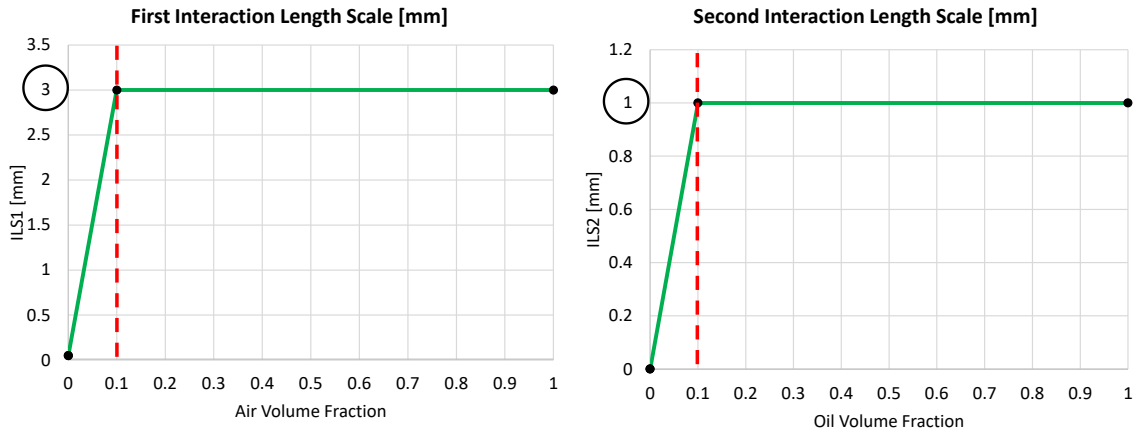


Figure 4 First and secondary interaction length scales, referred to air and oil respectively

The turbulence of a continuous phase could be modified by the presence of a dispersed phase in several ways. To provide an example, the wake of the dispersed phase particles is a source of turbulence and could generate velocity fluctuations in the continuous phase. In addition, the mutual interaction of velocity fluctuations and volume fraction fluctuations can generate further turbulence enhancements or damping terms. Therefore, shear production is properly modified considering the effect of drag and buoyancy on velocity profiles.

The multiphase turbulence transport equation is reported below:

$$\begin{aligned} \frac{\partial}{\partial t} \int_V \alpha_i \rho_i \Phi_i dV + \oint_A \alpha_i \rho_i \Phi_i \bar{v}_i \cdot d\bar{a} = \oint_A \alpha_i \left(\mu_i + \frac{\mu_{t,i}}{\sigma_\Phi} \right) \nabla \Phi_i \cdot d\bar{a} + \int_V \alpha_i (P_{\Phi,i} - \\ D_{\Phi,i} + S_{\Phi,i,u}) dV + \int_V S_{\Phi,c,PIT} dV + \sum_{i \neq j} \int_V \alpha_i S_{\Phi,ij} dV + \int_V \sum_{i \neq j} (m_{ij} \Phi_j - m_{ji} \Phi_i) dV \end{aligned}$$

Equation 13

Where Φ represents the transported scalar variable; depending on the turbulence model Φ can be replaced by the dissipation rate ε , the turbulent kinetic energy κ , the specific dissipation rate ω or the components of the specific Reynolds stress tensor $\bar{\bar{R}}$. μ_i and $\mu_{t,i}$ are respectively the dynamic and the turbulent viscosities, σ_Φ is the turbulent Prandtl number, $P_{\Phi,i}$ is the production term, $D_{\Phi,i}$ is the dissipation term, $S_{\Phi,i,u}$ is the user-defined source term, $S_{\Phi,c,PIT}$ is the source term due to particle induced turbulence and enters only the continuous phase transport equation, $S_{\Phi,ij}$ is the source term due to turbulence transfer between phases, m_{ij} and m_{ji} are the mass transfer rates between phases.

Chapter 3. CFD analysis of lubricant oil tanks sloshing of a high-performance car under racetrack maneuvers

3.1. Introduction

Under rapidly changing and severe inertial accelerations the assumption of immiscible fluids becomes critical and VoF-based models may provide misleading results. This, in combination with the high computational costs of transient VoF simulation, inevitably leads to waste of both time and resources during the design phase of the tank. The procedure described in the present work aims to overcome the limitations of traditional VoF-based models, extending the use of CFD to applications involving complex phase interactions.

The effectiveness and responsiveness of the proposed EMP approach in managing the phases interaction is tested with different applications, characterized by increasing complexity. Firstly, a relatively simple test case, studied in detail by Akyildiz et al. [68, 69] is simulated; Secondly, the approach is tested in static application of a real oil tank for high-performance sports cars with the aim of reproducing the pressure drops affecting the experimental pressure signal of the oil circuit, attributable either to partial/total uncovering of the pump feeding section or to air entrainment during the oil motion. The effectiveness of the proposed methodology is also assessed through the analysis of air/oil foam percentage measured downstream of the pump feeding section. Thirdly, a current production lubricating oil tank for high-performance sports car is analyzed but under actual driving inertial accelerations, that is a dynamic simulation of two consecutive track-laps. In this case the numerical air percentage which reaches the feed pump suction side is compared to the experimental oil pump delivery signal along the track-lap.

3.2. CFD methodology and setup

The following setup applies for all the sloshing simulations: the simplified test case and the static and dynamic simulations of current production oil tanks for high-performance sports cars. Simulations are carried out using the software STAR-CCM+ 15.02.009, licensed by Siemens DISW. The SIMPLE algorithm is used as implicit time integration method. A second order discretization scheme is used for velocity and turbulence and a first order discretization scheme is used for volume fraction. Since all cases are at room temperature, a constant density assumption is applied. A $k-\varepsilon$ two-Layer turbulence model is adopted for all cases. As for the wall treatment, an All- Y^+ approach is used because of the wide range of y^+ experienced throughout the simulation, in particular as for the lubricant oil tanks simulations. Such approach is able to simultaneously handle high- Y^+ and low- Y^+ situations by blending the corresponding wall function-based and low-Reynolds near wall formulations. A variable time-step is also adopted in order to comply with Courant number requirements in transient simulations. The time-step aims to maintain maximum Courant numbers of 0.5 and 1 over the free surface and within the whole domain, respectively.

3.3. Simplified test case simulation

In the studies of Akyildiz et al. [68, 69], the sloshing affecting a rectangular tank partially filled with water was analyzed under pitch oscillations. Several tests were performed varying the roll amplitude, the liquid fill depth, the rotation frequency and the disposition of the internal baffles. In each case a sensitivity of sloshing loads and the observation of the sloshing waves were carried out. They highlighted that both vertical and horizontal baffles were able to significantly reduce fluid motion. The tank was purposely designed with interchangeable internal baffle arrangements. Nine pressure transducers were mounted to monitor pressure distribution at different locations. The tank overall dimensions, the internal baffles disposition and the pressure transducers are shown in Figure 5. The tank can rotate about a transverse axis, z-axis in Figure 5, passing through the tank geometrical center. Despite this application could be effectively simulated using a traditional VoF framework, the application of EMP modelling is preferred in order to test the capability of

the approach to properly reproduce the entrapment and releasing of air into water as well as the formation of water droplets with a cheaper computational cost than that of VoF. With the specific purpose to evaluate the effectiveness of the model in challenging scenarios, the case with the most complicated dynamics is considered, i.e., with the maximum roll amplitude (8 deg), the maximum liquid fill depth (75%) and with a rotation frequency of 3 rad/s; both horizontal and vertical baffles are considered.

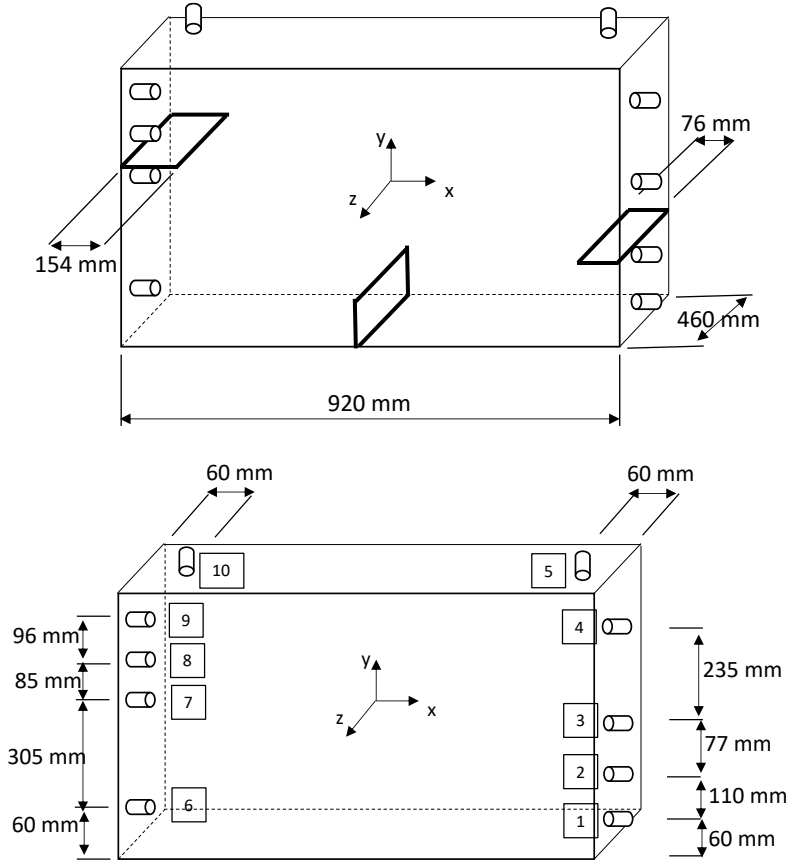


Figure 5 Baffle arrangement and location of pressure transducers elaborated from [68, 69]

Unlike VoF-based simulations, which require the use of a very fine grid to avoid numerical diffusion and to maintain a sharp interface, the EMP ones can handle phase interaction regardless of the adopted grid, because phase interaction models act at a sub-grid level. Likewise, it is still recommended to use a sufficiently fine grid to capture the sloshing dynamics and fit even the thinnest passages, typical of lubricant tanks of high-performance cars. Hence a trade-off between conflicting requirements has to be found. A polyhedral grid is considered a good solution to well reproduce the complex geometry of

the oil tanks which will be described in the next sections. It is also worthwhile to highlight that in sloshing applications a preferential orientation for the free surface is seldom detected, this leading to furtherly favor the use of randomly oriented polyhedral cells. To confirm grid-independence of the EMP model, a mesh sensitivity is preliminarily carried out and both hexahedral and polyhedral grids are compared. In addition, due to the symmetry of the real geometry, only half of the domain is modelled to reduce the computational effort, thus resulting in a computational grid made up of approximately 1'000'000 cells. A uniform cell size of 5 mm is adopted for both grids. As shown in Figure 6 and Figure 7, a qualitatively comparable water distribution is found with the two grid topologies. Plenty of cells with intermediate volume fraction, i.e. between 0.05 and 0.95 are found after 3.4 seconds, whereas a much sharper interface is recreated after 6.4 seconds. Such evidence, although qualitative, confirms the ability of the EMP model to reproduce phase mixing and subsequent separation according to the instantaneous local inertial accelerations applied to the tank. As shown in Figure 8, the EMP model grid-independency is further confirmed by the numerical pressure signals measured at Probe2, Probe7 and Probe8, which are almost overlapped. A good agreement comes out comparing the numerical pressure signal and the experimental one, regardless the adopted grid topology.

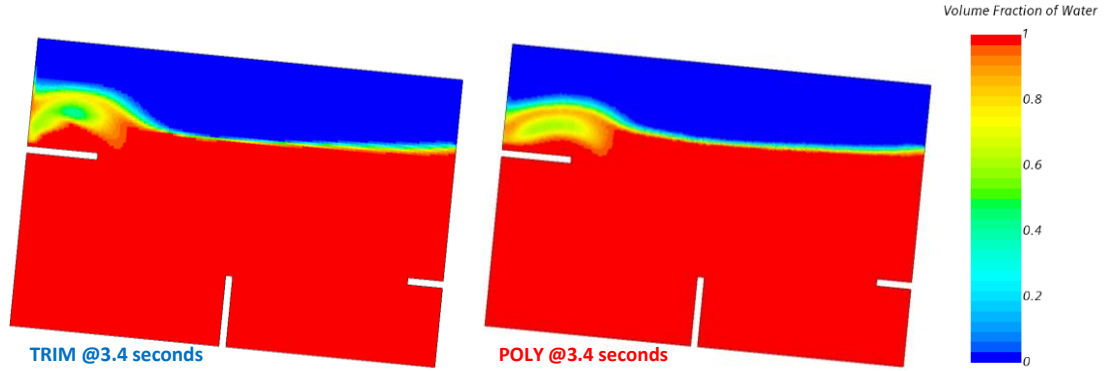


Figure 6 Free surface detection after 3.4 seconds with trim and polyhedral grid, respectively on the left and on the right. Vertical and horizontal baffled tank, fill depth = 75%, pitch angle = 8° , roll frequency = 3 rad/s

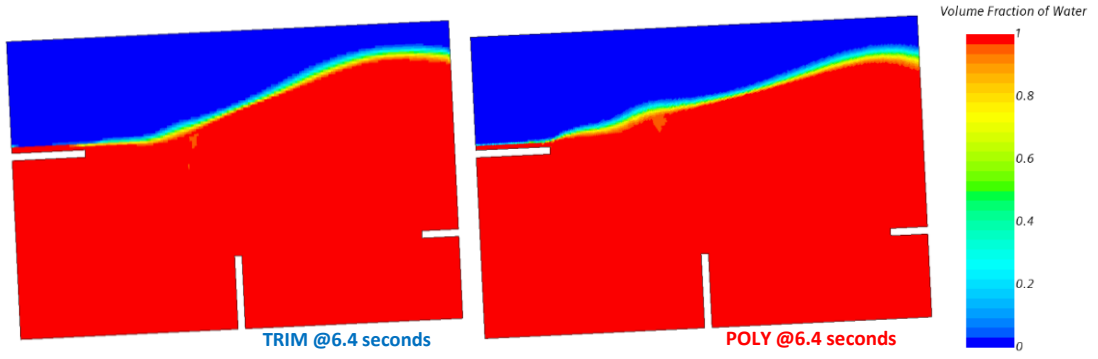


Figure 7 Free surface detection after 6.4 seconds with trim and polyhedral grid, respectively on the left and on the right. Vertical and horizontal baffled tank, fill depth = 75%, pitch angle = 8° , roll frequency = 3 rad/s

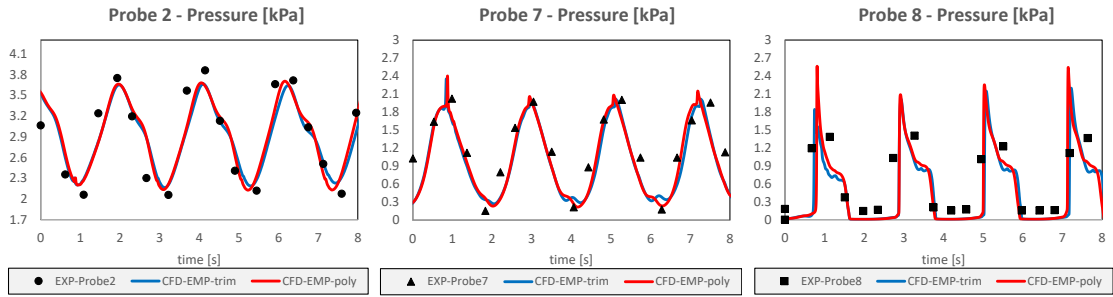


Figure 8 Comparison between experimental data and CFD results with EMP model, vertical and horizontal baffled tank, fill depth = 75%, pitch angle = 8° , roll frequency = 3 rad/s

Figure 9 shows the first interaction length scale in the fluid domain after 6.4 seconds. Such value directly affects the drag force which concerns liquid droplets dispersed in air. A constant value of 3mm is considered for volume fractions of air higher than 10%, promoting oil separation in those cells where air is predominant. In addition, progressively lower values are applied for lower volume fractions to promote numerical stability in quasi-single-phase regions. Figure 10 shows the second interaction length scale at the same instant. Likewise, this value directly affects the drag force which concerns air bubbles dispersed within the liquid phase. Here too a constant value of 1 mm is used for volume fractions of oil higher than 10%, thus promoting air separation where oil is predominant. Its value becomes rapidly lower for lower volume fractions to promote numerical stability.

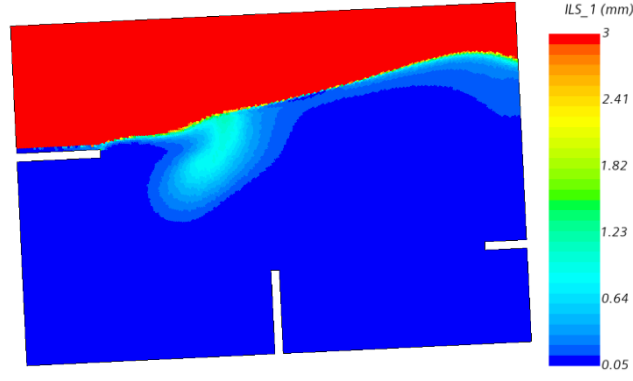


Figure 9: First Interaction Length scale after 6.4 seconds with polyhedral grid. Vertical and horizontal baffled tank, fill depth = 75%, pitch angle = 8° , roll frequency = 3 rad/s

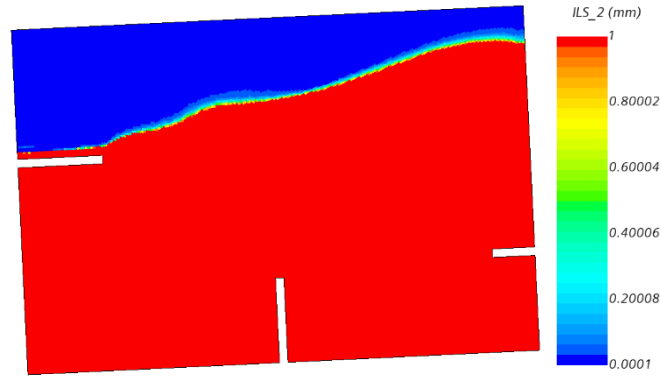


Figure 10: Secondary Interaction Length scale after 6.4 seconds with polyhedral grid. Vertical and horizontal baffled tank, fill depth = 75%, pitch angle = 8° , roll frequency = 3 rad/s

In order to comply with the convective Courant number but saving computational time as well as possible, adaptive time-step is used based on maximum Courant numbers on both fluid domain and free surface. Particularly, the time-step aims to maintain maximum Courant numbers of 0.5 and 1 over the free surface and within the whole domain, respectively. The average CFL number is around 0.2 whereas the maximum CFL number is below unity throughout the simulation, with the time-step ranging from $1e-4$ and $3e-3$ seconds, according to the instantaneous average and maximum Courant numbers of the previous time-step. Figure 11 shows the Courant number over the fluid domain respectively at 0.8 and 4.0 seconds.

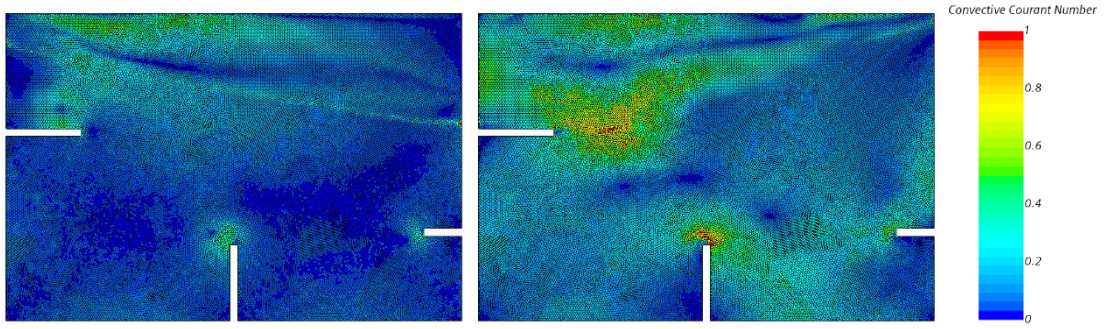


Figure 11: Convective Courant number after 0.8 seconds on the left and after 4.0 seconds on the right

Finally, the EMP model is compared with its VoF counterpart on equal grid spacing and time-step. As depicted in Figure 12, phases quickly start to “mix” together. Despite VoF assumes immiscible fluids and the two phases are completely separated at the beginning of the simulation; partially mixed air/water spots appear throughout the domain. Once formed, such spots cannot be recovered by the model. This is a confirmation of the well-known sensitivity of VoF to numerical diffusion unless very fine grids and time steps are adopted.

The numerical pressure signals measured at Probe2, Probe7 and Probe8 using both EMP and VoF simulations on equal grid are compared with the corresponding experimental pressure signals and they are reported in Figure 13. Again, the EMP model shows a better agreement with experimental data besides its superior capability to predict shape and position of the free surface, local phase separation and mixing.

Before assessing the performance of EMP in the simulation of the lubricant tank subject to multiple consecutive racetrack laps, a further preliminary sensitivity analysis is carried out on the oil tank subject to a static load, i.e., neither displacements nor accelerations are applied.

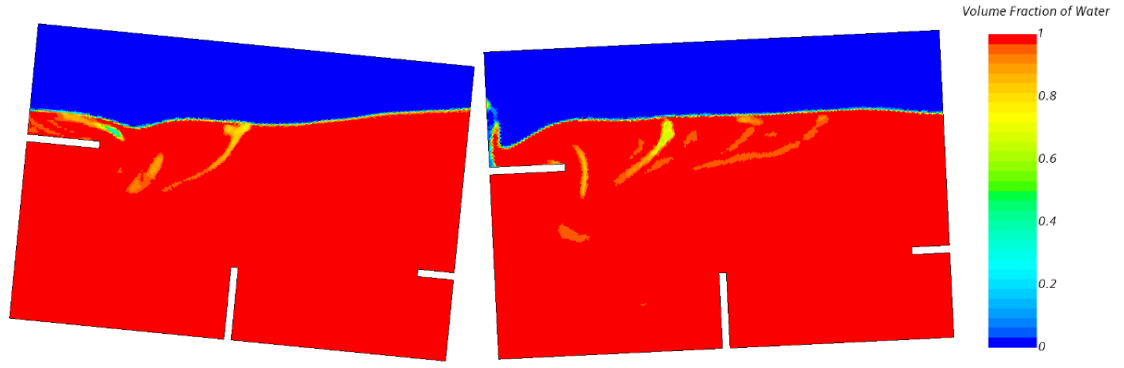


Figure 12 Free surface detection after 3.4 and 6.4 seconds, respectively on the left and on the right, with VoF model, poly mesh. Vertical and horizontal baffled tank, fill depth = 75%, pitch angle = 8° , roll frequency = 3 rad/s

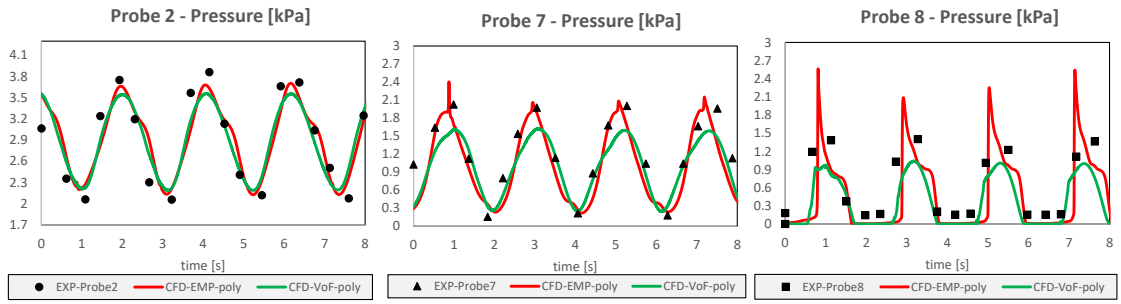


Figure 13 Comparison between experimental data and CFD results with EMP and VOF model, poly mesh, vertical and horizontal baffled tank, fill depth = 75%, pitch angle = 8° , roll frequency = 3 rad/s

3.4. Current production oil tank for high-performance sports car: static simulation

Based on the encouraging results of the EMP model in terms of ability to simulate the separation and mixing of phases without the need of using either numerically expensive high-resolution grids or very low time steps, the methodology is further tested in a real oil tank for high-performance sports cars. Particularly, attention is paid to the capability of the EMP approach to predict the formation of lubricant oil foam.

3.4.1. Experimental conditions

In this application no car inertial accelerations are taken into account and the tank is placed in a fixed position. Experimental data used in this work for model development consisted of test rig data at engine peak power operation @9250rpm and Wide-Open Throttle (WOT) condition. The experimental dataset, which have been provided by a collaborating entity under confidentiality agreement, consists of the volume percentage of foam quantity measured shortly downstream the suction side of the feeding pump as a function of the initial oil level inside the tank. As previously described, it is crucial to guarantee a continuous oil exchange from the inlet pipe up to the feeding pump, thus preventing partial or total uncovering of the feed pumps which could lead to engine failures. A network of chambers and ducts is used to allow the incoming air/oil mixture, which is perfectly mixed with 61% air in volume, to separate so that the suction side of the feeding pump is supplied with a continuous flow of oil. The entering air percentage is calculated considering the air volumetric flow rate, equal to 9975 l/h, and the oil mass flow rate, equal to 83 kg/min. The oil entering temperature is 90°C.

As aforementioned, specific vent ducts are adopted to allow the oil to be reused without being released into the atmosphere. A sketch of a typical tank layout is visible in Figure 14 below. As it can be noted, the primary circuit is made up of the main chamber and the inlet/outlet pipes. Its function is to manage the incoming mixture with the purpose of guaranteeing a continuous oil flow supply at the feeding pump. The secondary circuit is

made up of a network of chambers with the purpose of retaining oil vapors before they reach the vent ducts. Up to 5 chambers can be used in this perspective.

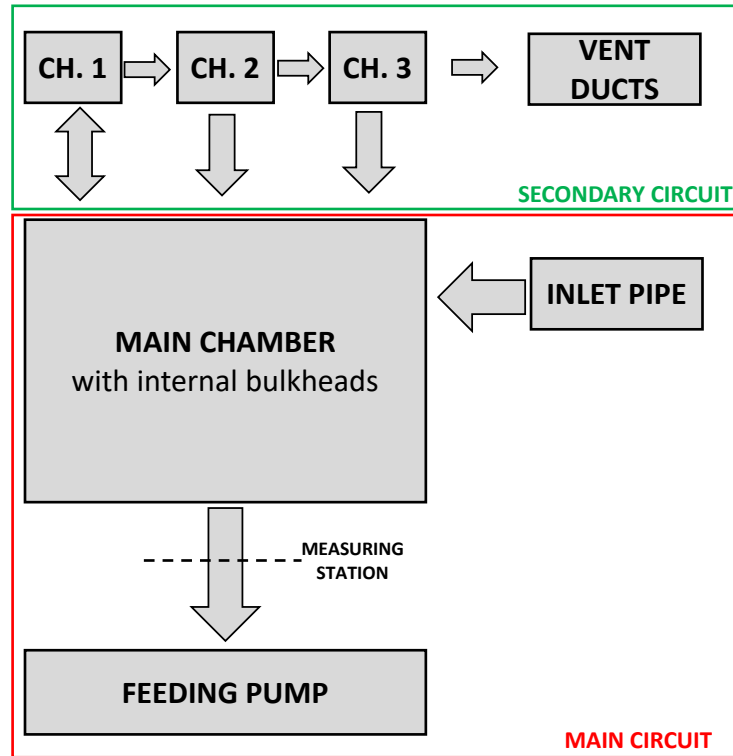


Figure 14 Typical tank layout and foam measuring station

Under static conditions and with a constant feeding flow rate, the oil tank should be perfectly able to separate the incoming phases. Therefore, equal oil mass flow rates should be found at the inlet pipe and at the suction side of the feed pump. The experimental analyses performed on this tank, on the contrary, revealed the formation of foam at the suction side of the feed pump. The percentage of foam can be directly measured from the difference between the mass flow of the oil entering the inlet pipe of the tank and the oil exiting towards the pump, both deriving from experimental measurements and which should be identical in a well-designed oil tank. As shown in Figure 15, in the present work two oil levels are simulated, for the sake of simplicity named as “high” and “low” in the following. An experimental foam percentage of 0.5% and 8% is measured for the high and low oil levels, respectively. As it can be noted, the measured foam percentage is expected to decrease for increasing oil quantity within the tank. The absolute uncertainty affecting the experimental data is $\pm 6\%$.

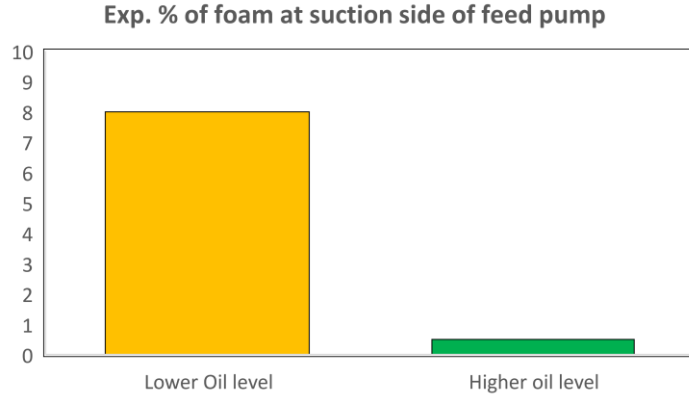


Figure 15 Experimental foam percentage measured at suction side of feed pump varying the oil level within the tank

3.4.2. Sloshing simulation

The simultaneous presence of multiphase fluid-dynamics and the complex geometry of a real lubricant oil tank requires the adoption of a model able to properly handle the interaction between phases, while preserving rapidity and efficiency.

For the sake of consistency with the experimental measurements, the computational domain covers the whole lubricant tank from the mixture inlet duct to the suction side of the feeding pump, resulting in a grid made up of nearly 800'000 cells with cell sizes ranging between 1 and 4 mm. These values are small enough to properly represents the complex internal connections of the tank, without excessively affecting the computational cost of the simulation. Unlike the simplified test case described in the previous section, this oil tank is rich in cavities, edges, fittings, and bulkheads. This further underlines the advantage of discretizing the domain using a polyhedral grid in order to better preserve consistency with the original CAD geometry. Since a constant oil mass flow rate is applied, the instantaneous and local formation of foam leads to instantaneous filling or draining phenomena of the tank itself. Therefore, even though constant boundary conditions are considered, the sloshing dynamics is always unsteady and the adoption of a time-based simulation is mandatory. Since these phenomena and their effects take tenths of a second or even whole seconds to evolve, 15 seconds are simulated for each oil level. Once again, the time-step aims to maintain maximum Courant numbers of 0.5 and 1 over the free surface and within the whole domain, respectively. The time-step ranges from $1e-4$ and $1e-3$

seconds, according to the instantaneous average and maximum Courant numbers of the previous time-step.

Constant density is again assumed for both phases and an All- y^+ approach is used to accommodate the wide range of y^+ experienced during the simulation.

Figure 16 shows a section cutting the inlet duct within the tank and, specifically, a comparison of VoF and EMP models in managing the entering air/oil mixture under the same numerical setup and grid. A fully homogeneous mixture, representative of the real functioning of the tank, is applied for both cases. As it can be noted, a huge difference is found in terms of oil distribution inside the tank since the early stages of the simulations. Focusing on the EMP approach, the centrifugal force to which the mixture is subjected along the 90-degree curvature of the inlet duct tends to separate the phases, pushing the heavier oil towards the outer radius. The mixture is then pushed upwards vertically against a hat-like surface with the aim of slowing down its speed rapidly. Inertial forces tend to compact the oil against the internal wall of the hat. As for the VoF model, phase separation is not found either along the 90-degree curvature or in the subsequent hat-like surface. On the contrary, a remarkable non-physical transport of the oil in the upper chambers of the tank is found. This phenomenon is related to numerical diffusion and highlights the limits of VoF model in representing phase separation and mixing. In this type of application, the limits of VoF cannot be solved by brute-force reduction of time-step size and grid size, as the problem lies in the fact that a fully homogeneous mixture enters the domain, and its separation and mixing must be correctly handled by the model.

To further emphasize the combination of improved robustness and higher computational efficiency of the EMP model, the same application is approached using a numerical setup which better suits VoF requirements in terms of grid finesse and convective Courant number. Automatic mesh adaptation is used for the best trade-off between computational time and reduction of numerical diffusion over the iso-surface. A cell size of 0.5 mm is always enforced in close proximity of the free surface. Also, a stratified mixture enters the domain to fulfill VoF constraint of immiscible fluids, even though in reality the incoming fluid is a fully homogeneous mixture of air and lubricating oil. The adopted stratification considers oil below and air above, coherent with gravity force. However, this limitation would already be enough to consider VoF unsuitable for this application. As depicted in Figure 17, where a different section of the tank main chamber in order to show the suction side of the feed pump, phase interaction is much better predicted and air bubbles dragging

is well appreciable. However, numerical diffusion is still visible in the domain, i.e. cells with volume fraction ranging from 0.05 to 0.95, which is not considerable foam since this model is only able to properly manage fully-separated phases. On top of that, the total computational time increases by about 20 times due to the combination of finer grid and resulting lower timestep needed to ensure low CFL numbers over the free surface.

So far, VoF and EMP models have been firstly compared under same grid and time-step, considering a fully-mixed entering air/oil mixture for both of them, which is representative of the real functioning of the tank. Therefore, VoF have been applied considering finer grid size and lower time-steps in order to be compliant with the model requirements. Both applications highlight VoF unsuitability for tank sloshing modelling: the first in terms of phase mixing and separation, not captured by VoF model; the latter because of the excessively longer computational time required and the non-physical representation of the intake mixture concerning VoF application.

From now on, the description of the investigation is therefore focus strictly on EMP results. Figure 18 shows a comparison between the initialized mixture and the solution found after 5.6 seconds, arbitrarily chosen but sufficiently later in time to appreciate the foam formation up to the suction side of the feed pump. The mixture, not perfectly separated, tends to reach the feed pump. It is also notable the stagnation areas where buoyancy phenomena bring the air back to the surface.

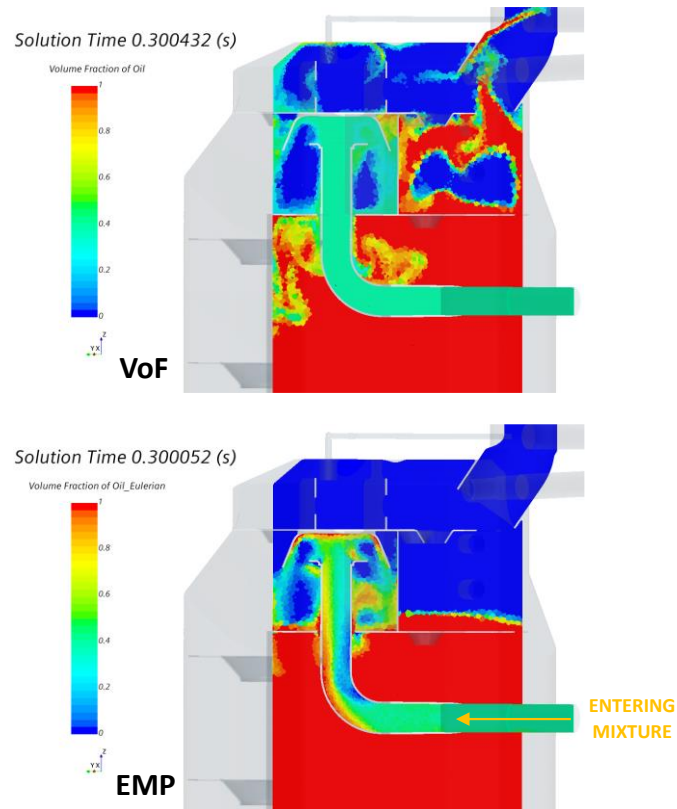


Figure 16 Entering air/oil mixture, inlet duct and hat internal surface phases separation after 0.3 seconds. Both approaches are compared, VoF on the left and EMP on the right. Higher oil level.

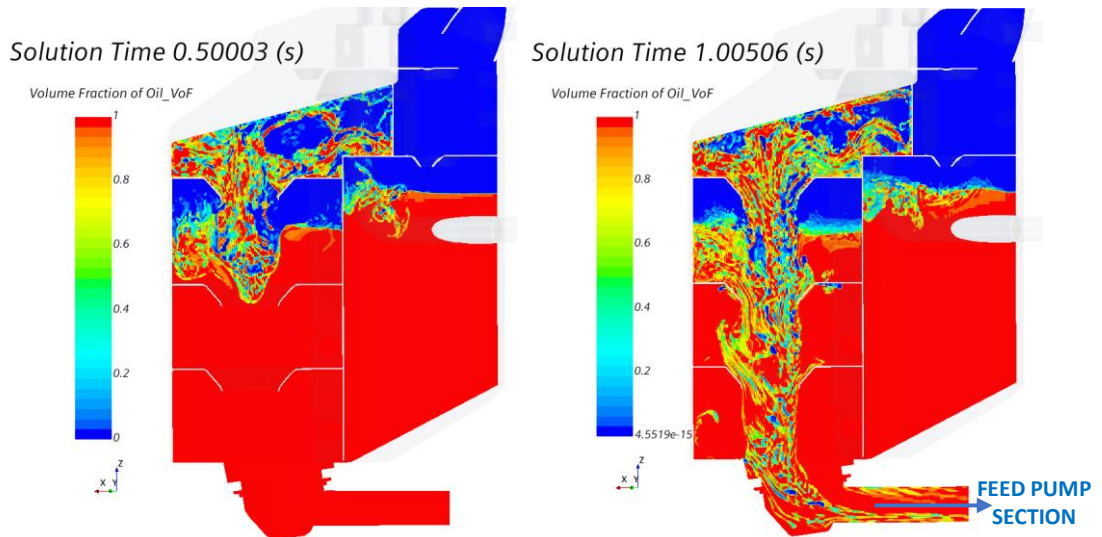


Figure 17 Oil volume fraction at feed pump after 0.5 and 1 second, VoF model with adaptive mesh adaptation, lower oil level.

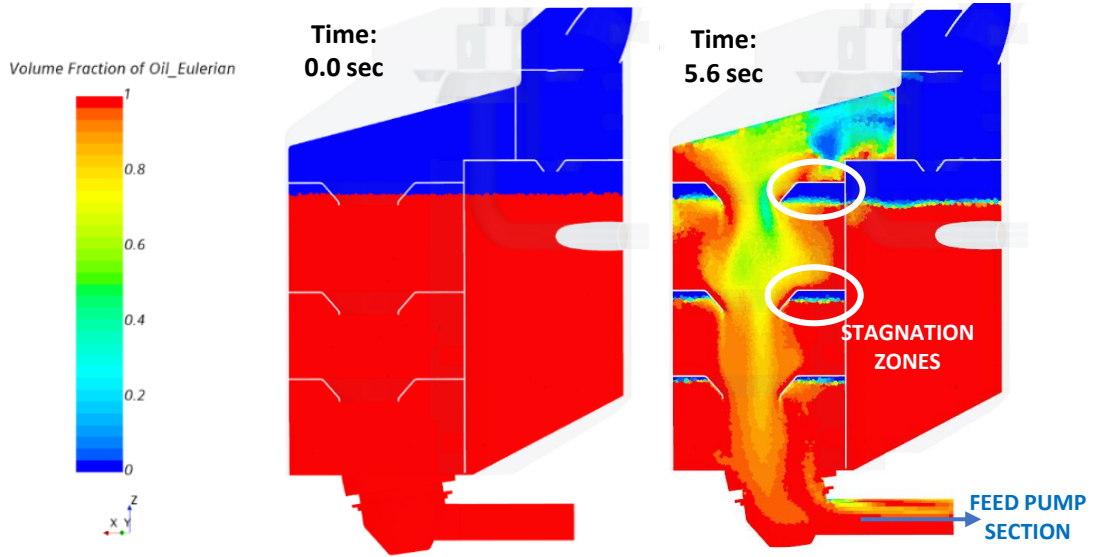


Figure 18 Oil volume fraction at feed pump @5.6sec. compared to the initial one, EMP model, lower oil level.

For a broader in-depth insight on interaction length scales working principle, Figure 19 shows the liquid volume fraction and the interaction length scales after 5.6 seconds. Focusing on the first interaction length scale, the higher value of 3 mm is adopted whenever the air volume fractions exceed 0.1. As it can be noted, phase separation is therefore promoted where air bubbles are dragged downwards. Conversely, the first interaction length scale tends to zero where the cell contains mostly oil, i.e. phase separation can't take place. Similarly, the second interaction length scale assumes the highest value of 1 mm within the cells with high oil volume fraction ($>10\%$) and it becomes close to zero where phase separation it's not supposed to take place, i.e. where close-to-zero oil volume fractions are present. The convective Courant number is monitored throughout the simulation and, as depicted in Figure 20, it never exceeds 0.5 in the majority of the domain and most importantly within the zones of major interest.

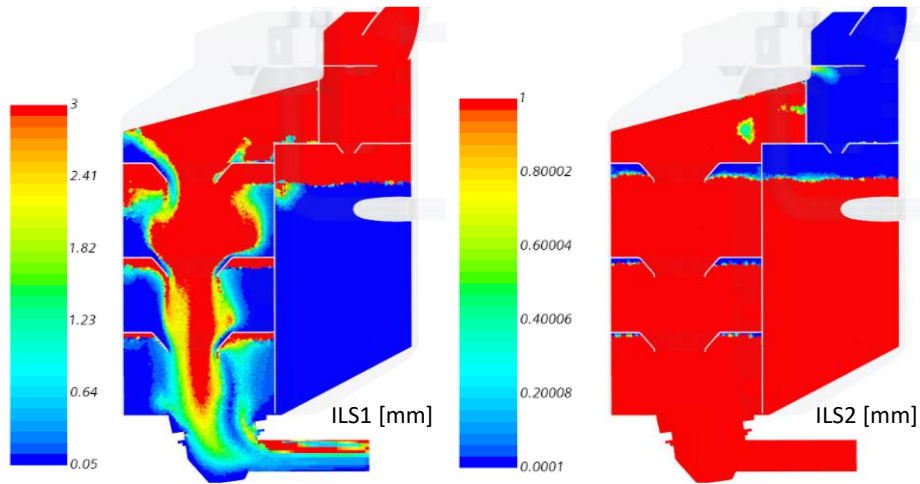


Figure 19: First interaction length scale in the left, second interaction length scale on the right, after 5.6 seconds.

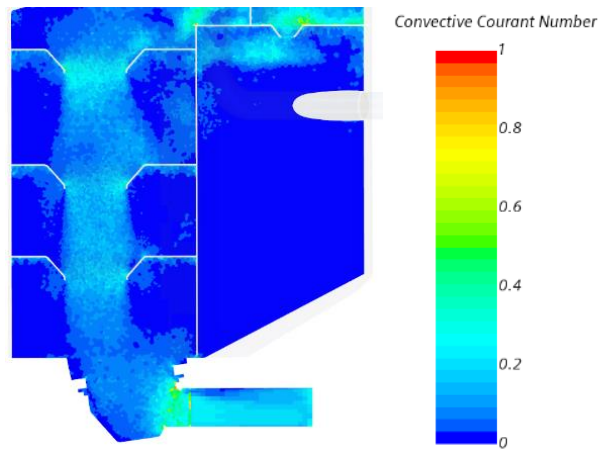


Figure 20: Convective Courant number after 5.6 seconds

Finally, Figure 21 shows iso-surfaces for different oil volume fraction (25, 50 and 75%) and, in light blue, where air volume fraction is at least 25%. The iso-surfaces aim to indicate the zones of major interest for phase interaction. For instance, the iso-surface @25% of oil points out the flaws of this tank design, since it allows the air to descend the tank main chamber that much leading to unwanted foam formation. Likewise, the air light blue dots show the effectiveness of the leakages, through which the air is allowed to get back to surface. These dots also indicate where the air stagnates and struggles to go up. The image points out the effectiveness of the model in capturing the complex phenomena related to tank dynamics such as air bubbles buoyancy, phase separation and foam formation.

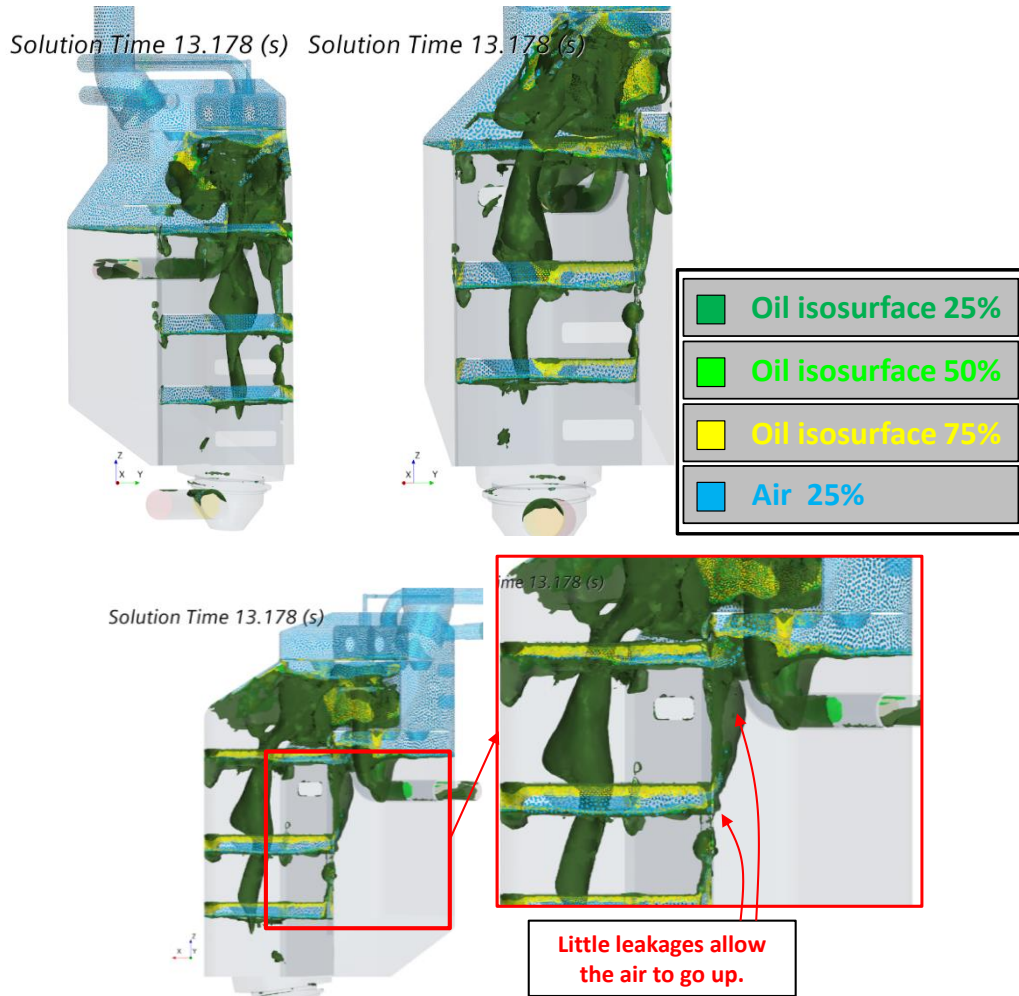


Figure 21: Phase distribution shortly after 13 seconds.

The foam percentage varying the oil level is finally shown in Figure 22 for both VoF and EMP. The figure suggests that the EMP model is able to capture the expected qualitative nature of the foaming phenomenon, i.e. a higher foam percentage is found with lower oil initial volumes. Focusing on EMP results, from a qualitative standpoint the progressive increase of foam formation reducing the oil level inside the tank is well reproduced by CFD. The EMP model can therefore be considered highly effective to simulate both phase separation and phase mixing.

Conversely, VoF approach appears not to be able to capture the expected trend of the phenomenon, even though a finer mesh size and lower time-steps are used.

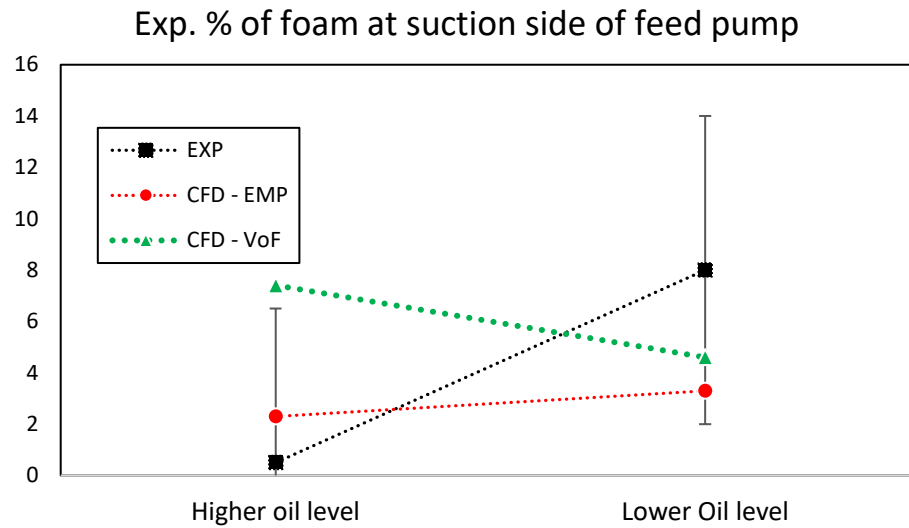


Figure 22 Comparison between EXP data and numerical results varying the oil volume inside the tank

Given the promising results found not only in a simplified rectangular tank with internal baffles, but also in a real oil tank geometry designed for high-performance sports cars, the model is finally applied to simulate the lubricating oil tank under actual accelerations in typical racetrack maneuvers.

3.5. Current production oil tank for high-performance sports car: dynamic simulation

As stated above, to simulate the sloshing dynamics affecting real lubricating tanks under actual racetrack accelerations, it is crucial to adopt a 3D-CFD modelling framework able not only to properly handle interaction between the phases (separation and mixing), but also to do it in a responsive and computationally efficient manner.

3.5.1. Experimental conditions

The experimental dataset, which have been provided by a collaborating entity under confidentiality agreement, consists of car longitudinal and lateral accelerations measured throughout a full racetrack lap, as it is shown in Figure 23. The lap time is around 92 seconds.

The mixture which feeds the tank is once again made up of air and oil fully mixed. Experimental mass flow rates of oil and air entering the tank are measured throughout the lap, resulting in an air volume fraction which it is now varying in time ranging from a minimum of 68% to a maximum of 83, as depicted in Figure 24. Moreover, the experimental dataset includes the pressure signal (Figure 25), measured shortly before the suction side of the feed pump, which unfortunately only covers the first 60 seconds. A schematic of the tank and the measurement station is shown in Figure 26.

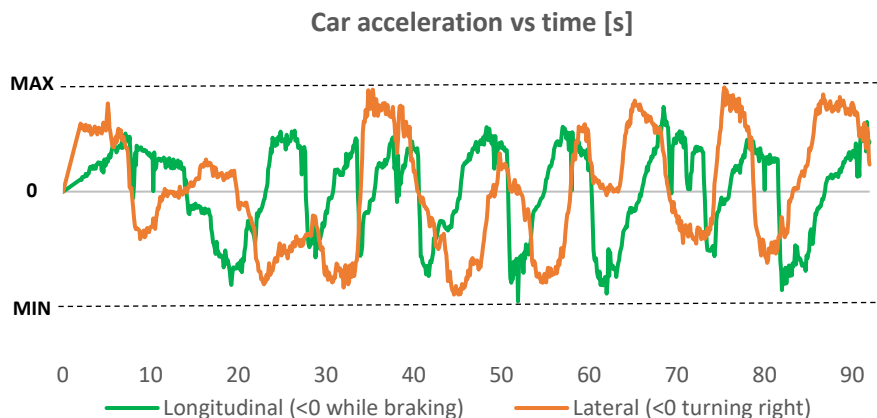


Figure 23 Racetrack accelerations vs time (1 full lap)

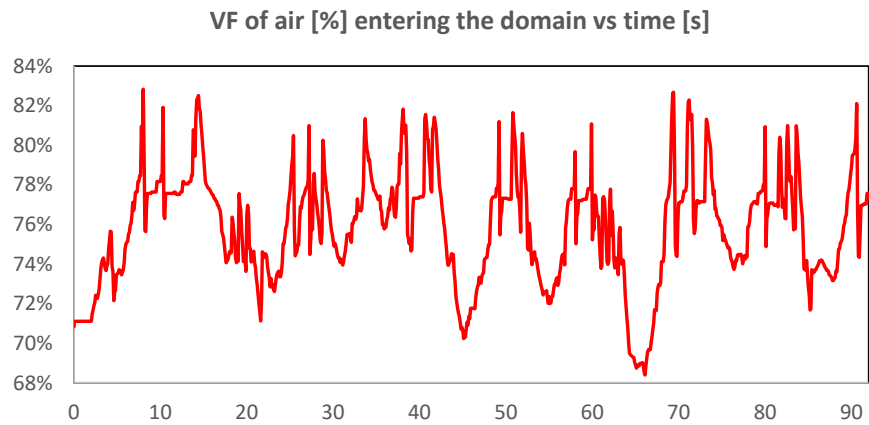


Figure 24 Experimental volume fraction of air entering the inlet domain vs time (1 full lap)

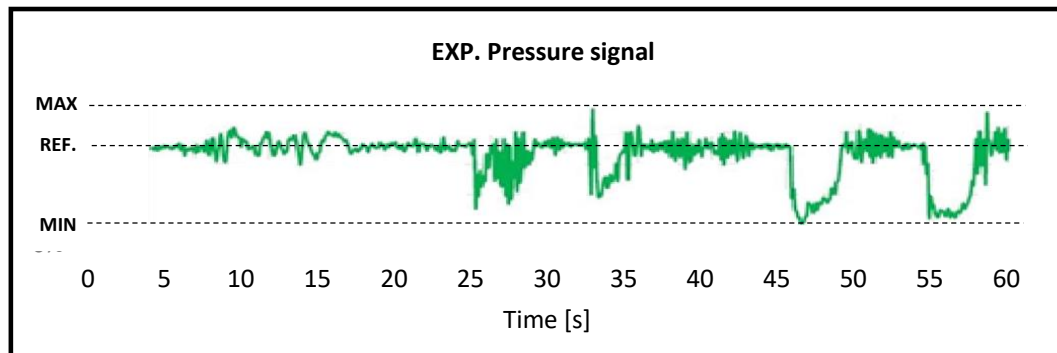


Figure 25 Experimental pressure trace measured at suction side of feed pump

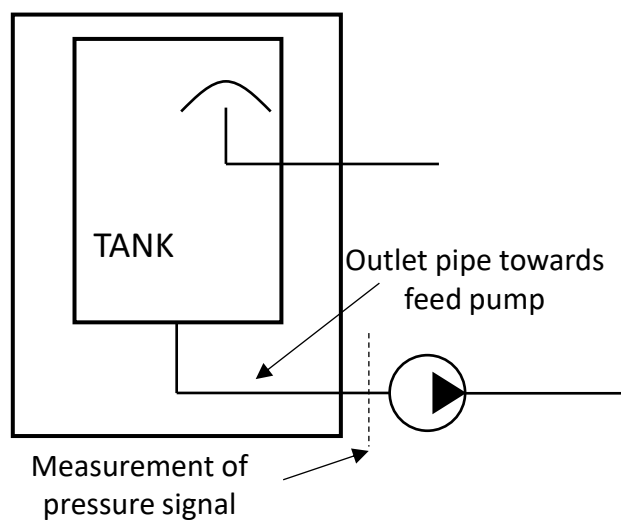


Figure 26 Tank layout and pressure measuring station

3.5.2. Sloshing simulation

As aforementioned, both longitudinal and lateral track lap accelerations onto the vehicle, and therefore onto the lubricant tank, vary quickly in both magnitude and sign.

The first consequence of such steep variations on the CFD analyses is that it is difficult to identify and isolate specific maneuvers, since arbitrary imposition of prescribed initial conditions (mainly, location of the free surface and local mixture state) would tremendously impact the numerical outcomes. Therefore, it is recommended to perform CFD analyses covering multiple (at least two) consecutive laps to reduce the dependency from the imposed initial conditions. The second consequence is that the adopted modelling framework should be able to manage phase interactions during and after the most invasive maneuvers in a sound manner, since they are expected to lead to foam formation and/or separation of oil droplets, whose dynamics must be properly managed. In fact, the tank design phase is strongly influenced by these phenomena, which therefore deserve maximum care and attention by the CFD simulations.

Consistently with the preceding static case, a time-based simulation is here too applied. The adaptive time-step ranges from $1e-4$ and $1e-3$ seconds with the aim of never exceeding maximum Courant numbers of 0.5 and 1 respectively over the free surface and within the whole domain. The computational domain covers the whole oil tank from the mixture inlet duct up to the lubricating pump inlet section. A polyhedral grid is once again adopted because of the complexity of the tank geometry. The computational grid is made up of approximately 600'000 cells. A lower number of cells is related to the lower number of connections and bulkheads which concern this tank with respect to the previous one. Starting from an initial oil position at rest, i.e. with a horizontal free surface inside the tank, two consecutive track-laps are simulated to get rid of the influence of arbitrarily imposed initial conditions. The total time spent for the simulation, considering both laps and therefore 184 seconds, is around 20 days using 128 cores.

Figure 27 shows some of the internal features of the main chamber of this tank. The others chambers and their connections are shown on the top while the vent ducts used to manage the oil vapors are shown on the sides. Gravity applies upon z-axis. Going into detail, the figure shows the distribution of the oil phase after 1 second of simulation, to stress the ability of the EMP model to manage separation of the two phases along the inlet duct. Here too, the inlet duct is purposely designed to slow down the incoming air/oil

mixture to increase the time available for the two phases to separate, due to gravity or inertial forces to which mixture is subjected inside the duct. Consistently with the previous case, the inlet duct faces an internal “hat-like” surface, which is designed to stop the mixture with the aim to emphasize the separation of the phases.

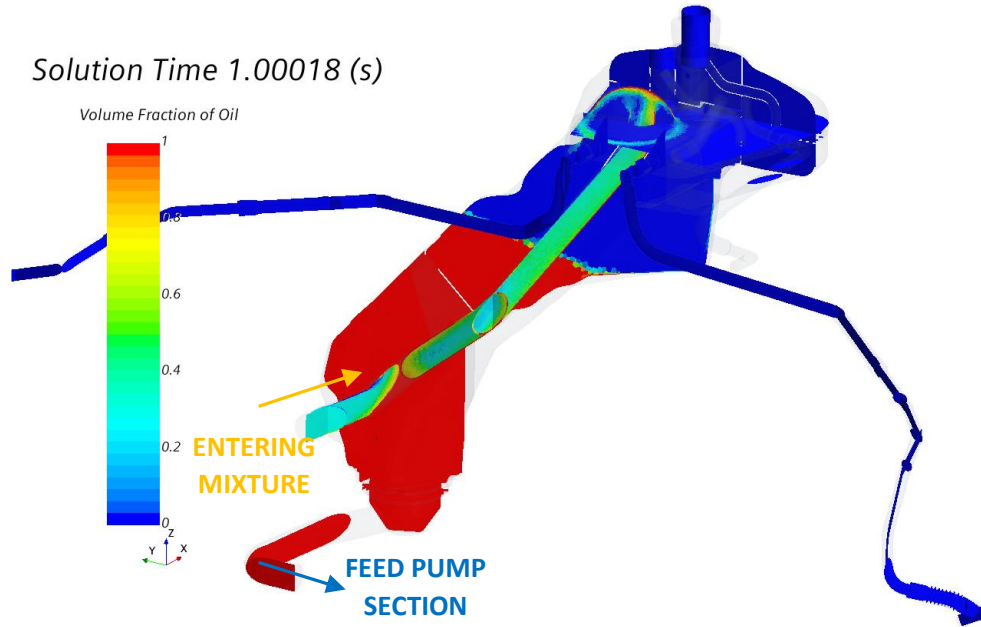


Figure 27 Oil volume fraction after 1 second

For the sake of completeness, Figure 28 shows 9 consecutive snapshots of the oil phase distribution. Snapshots are taken every 10 seconds during the second (final) track lap. As it can be seen, according to the instantaneous acceleration experienced during the track lap, the oil free surface tends to vary strongly in both shape and position. It is also noticeable the sloshing dynamics within the network of chambers in the upper part of the tank. During the most invasive maneuvers the oil tends to fill the chambers. Therefore, it is fundamental, during the design phase of the tank, to properly design piping to allow the oil to return into the tank main chamber. Ultimately, shortly after the beginning of the second track lap, i.e. after 100 seconds, the oil distribution is strongly different from the horizontal free surface initialized at the beginning of the first lap, pointing out the importance of simulating more than one lap.

A section cutting the internal hat surface is depicted in Figure 29 to show how inertial forces tend to compact the oil against the hat surface. Inside the tank main chamber, it is also noticeable the free surface slope due to the inertial accelerations experienced during a

right-turn, which for this tank is the most worrisome maneuver in terms of foam formation as it will be later described.

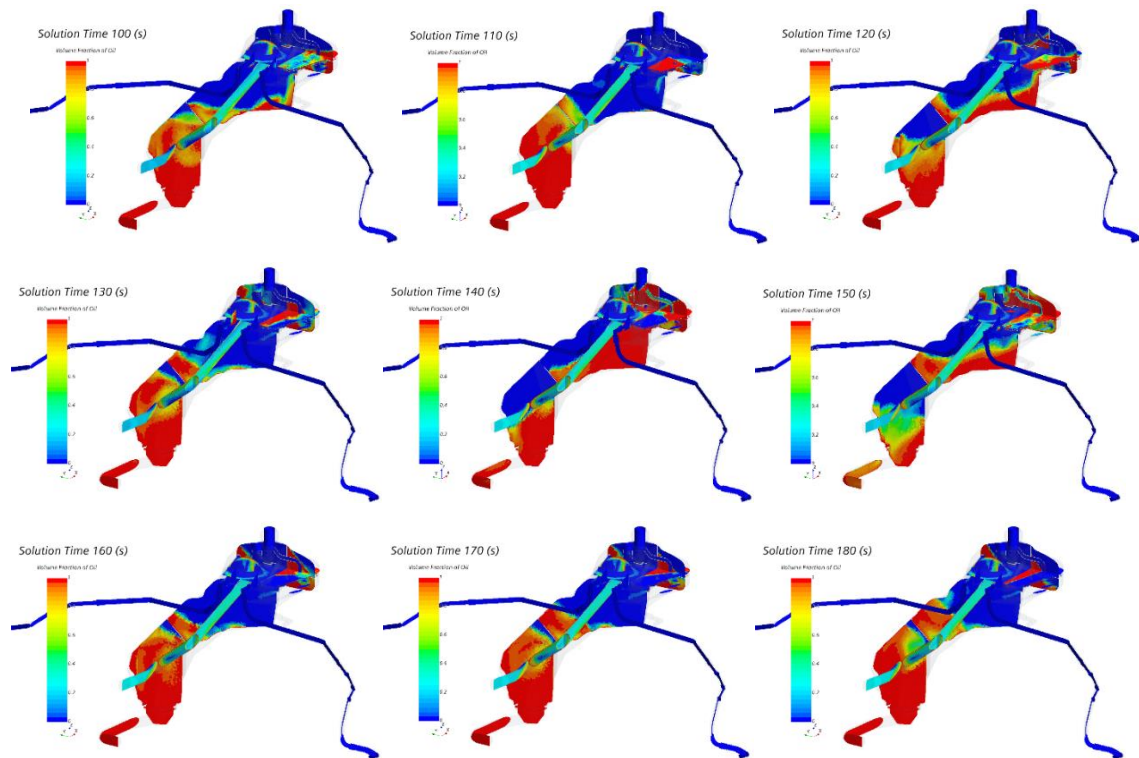


Figure 28 Oil phase distribution during second track lap simulation

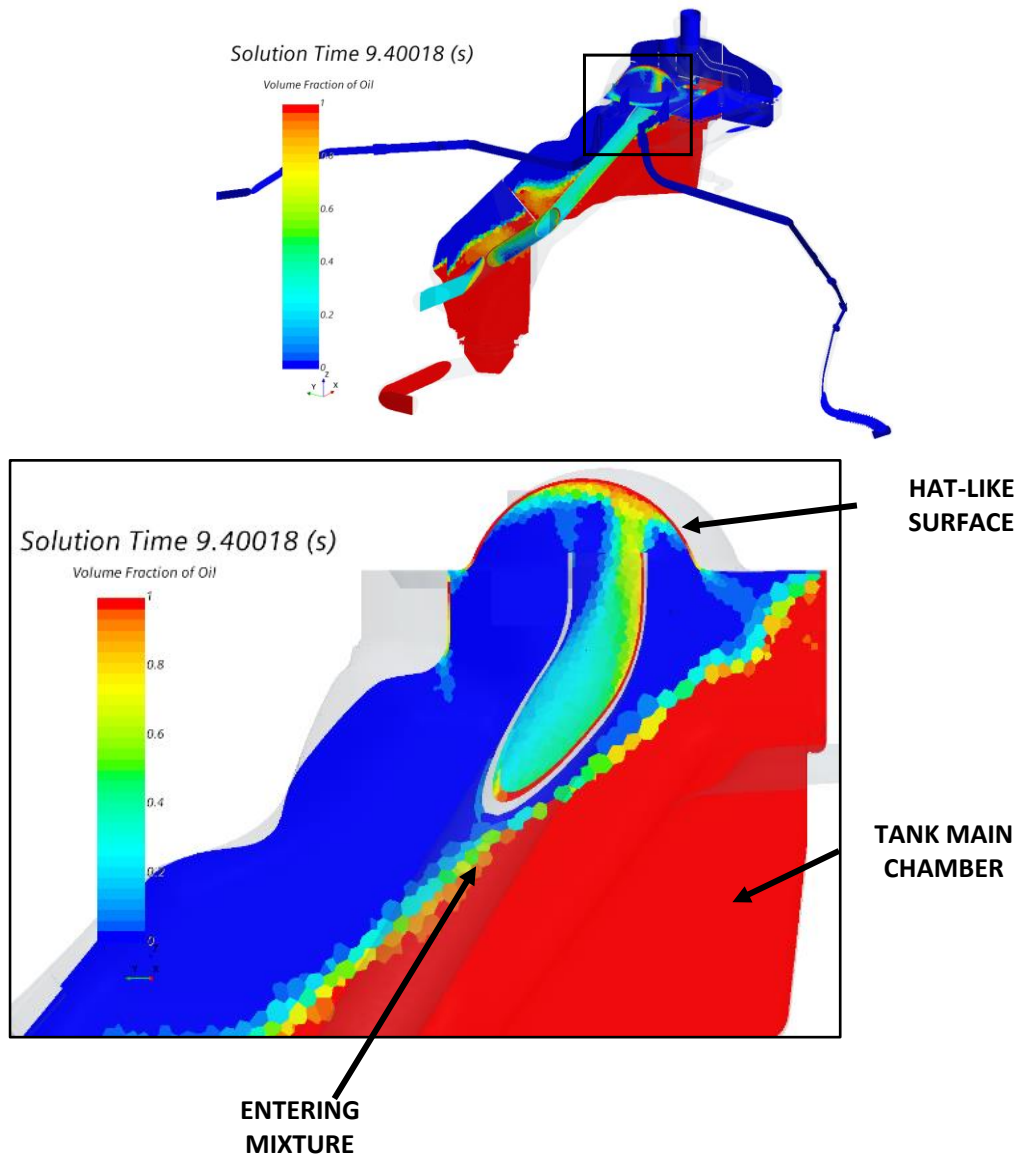


Figure 29 Detailed of hat internal surface: the oil tends to separate from air

The same interaction length scales of the aforementioned cases are here too adopted as shown in Figure 30. Since actual driving inertial accelerations are applied to the tank and since the interaction length scales are strongly related to phase interaction phenomena, it's fundamental to use them properly and to monitor their application throughout the whole simulation, with particular attention to the most invasive maneuvers where phase mixing and foam formation may easily occur.

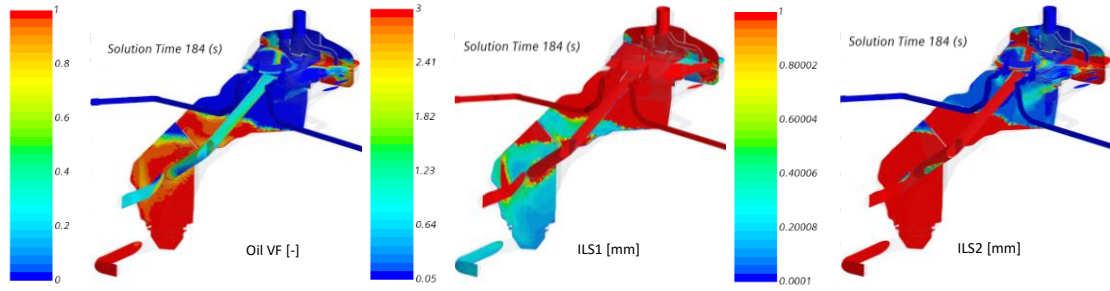


Figure 30: Liquid volume fraction on the left, first interaction length scale in the middle, second interaction length scale on the right, at the end of the second lap-track.

The oil pump delivery pressure signal is measured during the entire lap time. Figure 31 shows the signal measured during the first 60 seconds of a track lap, which takes into account about two third of the total lap time. Any oscillation or deviation with respect to the reference value is related to a non-uniform and non-continuous functioning of the feed pump; it can therefore be associated with the formation of foam inside the tank. As abovementioned, this unwanted behavior could lead to mechanical failures even after very short interruptions of the lubrication circuit. The investigated application, where geometrical and physical complexities are combined to very long physical times to be covered, emphasizes the necessity to define a proper modelling approach able to robustly and efficiently simulate sloshing dynamics and phase interactions at reasonable computational costs and times.

Consistently with the static application described above, the air percentage is measured at the suction side of the feeding pump throughout the simulation, and it is shown in Figure 31. As it can be noted, a good agreement is found between the experimental pressure signal measured shortly before the feeding pump and the numerical air percentage measured accordingly. Whenever a perturbation in the experimental pressure signal is measured, air is detected in the corresponding CFD simulation. In addition, the peaks of air concentration seen by CFD at the pump inlet section are well phased with the sudden drops in the oil pump delivery pressure.

In the analyzed tank, the right-turns tend to tilt the oil free surface uncovering the suction side of the feed pump. The first severe right-turn is reached only after 22 seconds. This is the reason why almost zero foam formation is found up to that moment, in both laps.

Ultimately, non-negligible differences between the simulated laps can be observed. Even though peaks are phased in time in the two consecutive simulated laps, higher air

percentages are computed along most of the second lap. This is due to the effect of initial conditions: while the first lap starts with an unrealistic horizontal free surface inside the main chamber of the tank, the second lap starts with the oil well spread inside the tank and part of it falls inside the ducts and chambers in the upper part of the tank. Consequently, a lower level of oil is found inside the main chamber and, based on the results of the static application described in the previous chapter, a higher tendency towards foam formation is found.

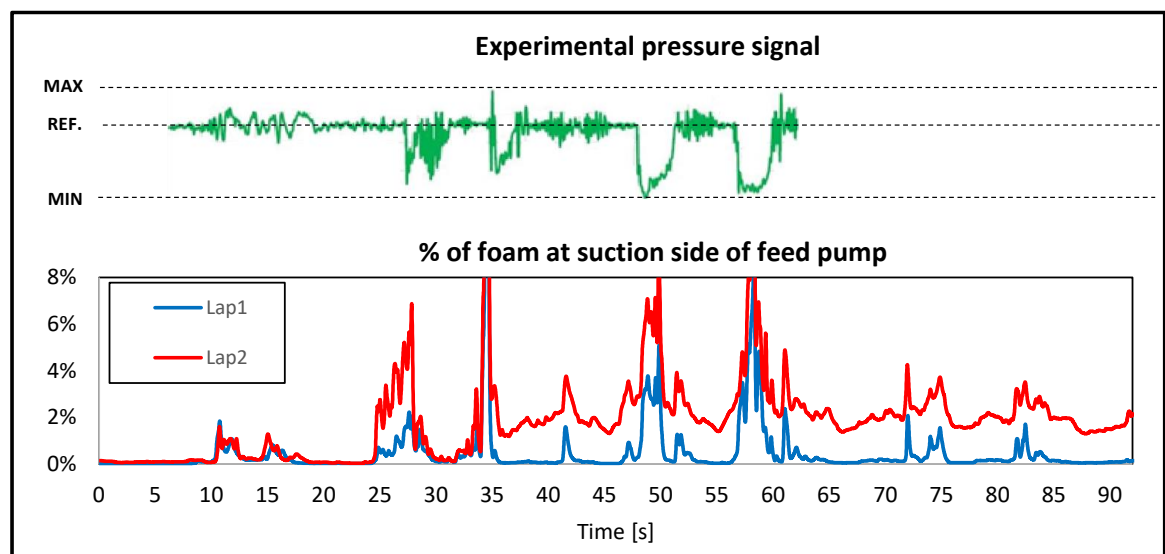


Figure 31 Comparison between experimental pressure trace (about 60 seconds are available) measured at suction side of feed pump vs. numerical percentage of foam

Chapter 4. Reactive flows theory

4.1. Introduction

The evergreen purposes to reduce fuel consumption and pollutants emissions improving engine efficiency are still among the main goals for the automotive industry. This decade sees as a priority the claim for a complete electrification of the passenger car fleet, CO₂ emissions reduction target, pushed by legislations, can realistically be obtained only by a proper mix of more efficient combustion engines and of hybrid/full-electric powertrains. Problems related to electric energy production, storage and distribution, battery raw materials and lifecycle, recharging time and driving autonomy are still far from being fully solved, hence the most viable immediate option for a cleaner mobility is hybridization [70, 71]. A hybrid vehicle is a complex system where the thermal and electric energy sources are proficiently combined, taking advantage of both. Issues related to electric energy storage and range can be partially overcome thanks to the internal combustion engine (ICE), while tailpipe emissions can be cut down thanks to the development of ad-hoc combustion technologies and strategies. To save costs and weight the thermal unit of a hybrid vehicle should be as simple and cheap as possible, especially regarding the aftertreatment system, while retaining high performance in terms of fuel efficiency. From a tailpipe emissions standpoint, use of alternative or CO₂-neutral-fuels combined with ultra-lean combustion seem particularly effective in reducing soot formation [72] CO and NO_x emissions in Spark Ignited (SI) engines.

Cho et al. [73] called attention to how the NO_x emissions can be strongly reduced by low-temperature combustion strategies and lean hydrogen burn, largely increasing the engine efficiency. The use of lean or ultra-lean mixture can prevent the use of three-way catalytic converters, whose high efficiency is constrained by stoichiometric engine conditions. If combustion temperatures are low enough to avoid NO and NO₂ production, which becomes negligible below 1800 K, it could be possible to strongly reduce or even completely remove any NO_x after-treatment system. In addition, under lean conditions cylinder intake pressure needs to be higher, allowing the engine to reduce throttling and therefore pumping losses. On top of that, since SI engines are particularly sensitive to air-

fuel ratio (AFR), with air excess combustion can be more complete, reducing the CO production typical of incomplete combustion. Temperature reduction obtained using lean mixtures increases the knock limit [74, 75, 76] and, consequently, allows the compression ratio to be raised increasing combustion efficiency. Nevertheless, ultra-lean operation gives rise to significant challenges. Quader [77] and Germane et al. [78] highlighted how poor-quality ignition or failure to ignite ultra-lean mixtures can cause misfires and undesired cycle-to-cycle variability [79], therefore reducing combustion efficiency. Therefore, the reliable and efficient combustion of homogenous ultra-lean mixtures remains an ambitious target which requires innovative technologies to increase combustion stability and velocity and to overcome potential issues. To this aim, TJI (Turbulent Jet Ignition) is being given a renewed interest [80].

4.2. Turbulent Jet Ignition

TJI allows to reach stable combustion of ultra-lean mixtures with low emissions and high efficiency thanks to the adoption of two divided chambers, connected through one or more orifices. TJI systems use a small quantity of near-stoichiometric or slightly rich mixture, ignited inside the smallest chamber (usually referred to as “pre-chamber”, PC), which is then forced out by thermal expansion through small orifices into the largest chamber (usually referred to as “main-chamber” MC). The combustion within the PC induces a pressure gradient between the two chambers, pushing the reacting flow and/or the hot products into the MC through the orifice. The hot turbulent jets ignite the main chamber lean mixture thanks to the complex coupling of fluid mechanics, thermal effect and chemical kinetics. The radical species inside the jets can result from complete or incomplete combustion, according to the PC mixture thermo-chemical composition and orifice diameter. They are highly reactive and, since they contain partially or fully-burnt combustion products, they are characterized by very high temperature, allowing the main ignition onset. Furthermore, the high velocity of the jets exiting from the orifices promotes the production of turbulent kinetic energy and allows an efficient distribution of reactants in the MC, forming a number of simultaneous highly-reacting hot spots equal to the number of nozzles.

In modern high-performance engines, TJI technology is widely investigated and already adopted as main combustion initiator in several current production ICEs. Indeed, TJI technology usually allows a reduction of the combustion duration, thus leading to a more efficient and rapid combustion of the end-gases, reducing the engine knock-tendency. Consequently, if compared under same knock-tendency with Spark Ignited (SI) engines, TJI technology allows the adoption of earlier combustion phasing, i.e. earlier 50% of mass fuel burnt (MFB50). If properly designed, TJI technology can therefore be used to increase the engine Indicated Mean Effective Pressure (IMEP) or even to exploit the more efficient combustion using leaner mixtures without compromising the overall performance with respect to SI engines.

Focusing on Formula1, this environment still remains a field where horsepower race can evolve without any “external” stringent legislation regarding tailpipe emissions. However, F1 technical regulation imposes the maximum fuel mass flow, which must not exceed 100 kg/h. Since horsepower can be directly related to the amount of air trapped within the combustion chamber, in order to be competitive is mandatory to look for efficient combustion systems able to burn lean or highly lean mixtures. This is the main reason why TJI combustors rapidly spread in F1 field.

Going into detail about the TJI working principle, turbulent structures generated by these jets strongly enhance mixing and ignition of the MC mixture; they are deeply influenced by the main chamber temperature because of the mixture viscosity [81].

The benefit deriving from the use of pre-chamber combustion principle has been widely investigated in the last two decades. Distaso et al. [82] found a strong reduction of fuel consumption, up to 20%, when using TJI technology. Attard et al. [83] found a strong reduction of NO_x emission due to the near elimination of N₂ dissociation, thanks to the low combustion temperatures and a 18% improvement in fuel consumption when compared to conventional stoichiometric SI. In [84] they found that a PC-initiated combustion allows to reduce the ignition energy demand and consequently, the spark plug size, being the ignition driven by the chemical, thermal, and turbulent effects of the jets exiting the pre-chamber rather than by the flame kernel growth of conventional SI combustion. They also found very good performance and combustion stability using a single fuel or different fuels for the chambers [85, 86].

The physics behind Turbulent Jet Ignition technology is very complex and hardly attributable to a unique governing aspect, as both chemistry and turbulence have a

paramount role in the jet propagation through the nozzles. The ignition process [87] is strongly related to the local turbulence and mixing characteristics which make the PC a difficult device to study through numerical simulations. Ignition is also strongly related to flame quenching, deeply studied by Kyrtatos et al. [88] who classified thermal and hydrodynamic quenching mechanisms. The former occurs when the heat losses towards the nozzle wall exceed the heat release of the flame, whereas the latter is caused by the mixing of the combustion products exiting from the PC with the cold unburnt mixture in the MC.

Several studies investigated the impact of the geometrical details of the combustion chamber(s) on the characteristics of TJI combustion. As aforementioned, orifice design is also an important parameter for combustion propagation. Being the only link between the chambers, nozzles (both shape and global cross-section) play an important role in jet propagation and PC mixture. Bunce et al. showed in [89] that a high number of orifices with low overall cross area results in the highest net thermal efficiency due to the flame front exiting from each of the ignition sites having less chamber volume to cover. Gentz et al. [90, 91] found that jets that are evenly distributed to cover the largest possible MC volume will consume the main-chamber charge more rapidly; faster and more vigorous jets with longer initial penetration are required as the mixture becomes leaner. Biswas et al. [92] investigated the ignition characteristics of a homogeneous mixture inside a pre-chamber combustor, finding two different ignition mechanisms, namely “jet ignition” and “flame ignition”. As the orifice diameter increases, the ignition mechanism tends to switch from jet to flame ignition. The orifice diameter therefore plays a fundamental role in the amount of mass flowing into the PC through the nozzle before the ignition event. It also determines the velocity of the jets entering the MC, and, ultimately, the turbulence promoted by the flame when crossing the nozzle; ultimately, it strongly affects the combustion regime once in the MC. Muller et al. in [93] isolated the thermal effect from that of chemical kinetics injecting a mixture of CO_2 , H_2O and N_2 into the main chamber. The MC ignition is faster in the reactive species case than in the inert one because of the increase of mean temperature, decrease in ignition delay time and radical proliferation. The authors concluded that the chemical kinetics effects on the turbulent jet are very important and should be considered for TJI simulation and prediction. In addition, they compared LES and RANS approaches to evaluate the impact of turbulent structures on TJI performance. Strong correlation between mixing, temperature and vorticity were observed. This last aspect is confirmed in [81] in which Benekos et al. analyzed the effect of wall temperature,

main chamber mixture composition and unburned mixture temperature through 2D-DNS studies.

Several investigations have already been performed applying TJI to real engines [94, 95, 96] or rapid compression machines (RCMs) with no uncertainties in terms of fuel/air mixing. In the latter case, an investigation on the performance of the PC combustor varying its geometrical parameters was carried out in [90], while Gholamisheeri et al. in [97] studied the effect of mixture stoichiometry on jet penetration speed and the same authors in [98] discussed the relationship between the reduction of hot jet penetration speed and equivalence ratio, explained in terms of density increases for richer mixtures. Further studies using Rapid Compression and Expansion Machines (RCEMs) were performed by Bolla et al. [99], who found that during the jet mixing the mixture in the MC is mainly ignited by autoignition followed by a progressive transition to a deflagrative premixed flame propagation mode. They also investigated the effects of tangential nozzle angle and volume variation, concluding that the best performance can be obtained with straight nozzles and larger volume, delivering better mixing and faster combustion [100]. Finally, Xu et al. [101] studied RCEM under engine-like conditions, further highlighting the importance of initial hot jet speed to drive the MC early combustion.

4.3. The complexity behind combustion modeling in TJI systems

The combustion regime affecting the flame inside pre-chamber, main-chamber and nozzles has not received analogous attention since it is strongly correlated not only with the geometrical characteristics of the combustion chamber but also with the local turbulent and chemical scales. In this scenario, CFD simulations can be very useful to give a comprehensive insight into the wide-ranging physics of the engine, i.e. a deeper understanding of the complex physics behind the observed phenomena. A reliable CFD simulation of combustion can be performed only choosing the most suitable modelling framework for the analyzed problem, which in turn is strongly dependent on the combustion regime itself. Under laminar conditions a purely chemical kinetics-based approach is preferable, while a model for turbulence-flame interaction is mandatory when flame propagation is mainly driven by turbulence and flow field (this is the case of the “flamelet” regime experienced in SI internal combustion engines).

In TJI systems it is not possible to a-priori decide which choice is to be made, considering that both chemistry and turbulence related phenomena are both strongly relevant, mostly within the pre-chamber. The study of the combustion regime during PC, orifices and MC combustion can help in understanding what is the best CFD framework to simulate TJI cases in both laboratory test cases and real engine geometries. This is the major aim of the present study. Even though the use of rapid compression machine with homogeneous mixture would allow the investigation of the TJI physics without the uncertainties related to PC mixture composition, very few studies have been conducted with no uncertainties also in terms of turbulence and flow field. For this last reason, in the present study numerical simulations are carried out on a fully premixed and quiescent combustion vessel tested by Yamaguchi et al. [102]. The availability of experimental data and the lack of uncertainties in terms of turbulence, mixing and wall temperatures are the reason behind the choice of this test case. Laboratory experiments are simulated using CFD with the aim to understand the reasons behind the experimental evidence. A critical discussion of the combustion regime is carried out to understand what the most suitable modelling framework for this kind of problem is.

4.4. Combustion modelling

CFD modelling of internal combustion engines is playing an increasingly important role in improving engine design, leading to a reduction in pollutant emissions and fuel consumption. In the present work, simulations are carried out using STAR-CD 2020.1, licensed by SIEMENS DISW. Below are briefly described the most common and widespread combustion models, usually adopted for “standard” 3D-CFD simulations.

4.5. ECFM-3Z model

The Extended Coherent Flame Model-3Z (ECFM-3Z) model is suitable for non-homogeneous premixed combustion and it splits the combustion phenomenon into two parts, i.e. the premixed and the post-flame part. In the first one the flame surface density equation is solved while in the latter the Magnussen eddy break-up model is adopted. The ECFM-3Z model is a combustion model capable of simulating the complex phenomena behind turbulent mixing, flame propagation, diffusion combustion and pollutant emissions which concern modern internal combustion engines. A schematic of the model is shown in Figure 32.

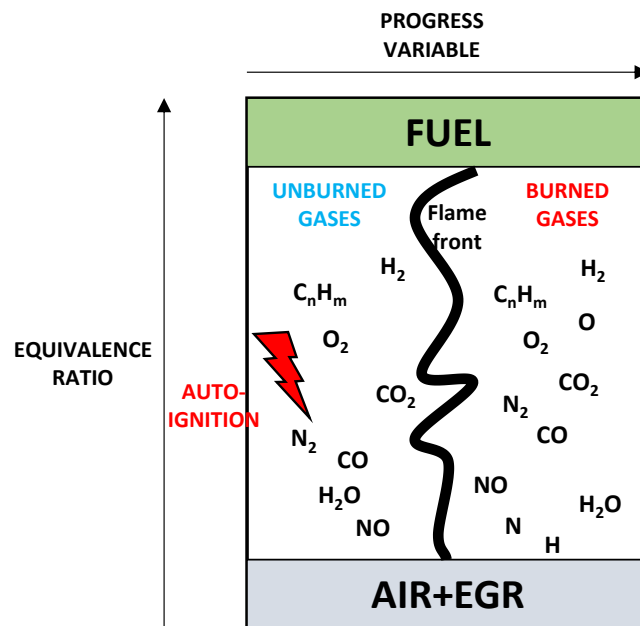


Figure 32: Schematic of ECFM-3Z combustion model

The model is so called since it considers the three zones of mixing, namely the unmixed fuel zone, unmixed air plus EGR zone and the mixed gases zone. These zones are modelled at a sub-grid level, being too small to be resolved by the grid.

4.6. G-Equation model

The G-Equation combustion model separated the unburnt from the burnt region through a scalar named G , with the isosurface $G = 0$ demarcating the border between the two regions. Therefore, this isosurface represents the flame front position. A schematic of the working principle of this model is shown in Figure 33. Nonetheless, the variance of G , G' , is needed to obtain the flame brush thickness from which a value for the progress variable is computed.

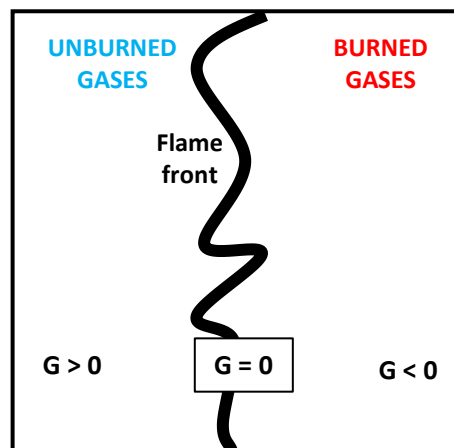


Figure 33: Schematic of G-Equation and isosurface $G=0$

The model may be used for premixed or partially premixed combustion.

4.7. Detailed Chemistry model

The Detailed Chemistry model is designed to meet the requirements of incorporating detailed chemical kinetics in engine simulations since it efficiently solves transport equations for all species using detailed reaction mechanisms.

Solving the set of equations is very time consuming so the dynamic multi-zone (DMZ) method is used to create clusters of cells with a similar thermo-chemical state. The working principle of the DMZ method starts with constructing groups, then it solves the chemistry equations and ultimately a mapping of the group solution to each cell is performed. More specifically, groups are constructed for cells that share similar thermo-chemical states. The quantities which are usually adopted in this perspective are the mixture fraction (mixing process), temperature (combustion phenomenon), enthalpy (energy) and the hydrogen mass fraction to capture the auto-ignition phenomenon (other species such as H_2O_2 can be used in this perspective).

Therefore, group mean quantities such as density, temperature, pressure and mass fraction are calculated by averaging the corresponding quantities for all cells:

$$\rho_g = \frac{\sum_i \rho_i v_i}{\sum_i v_i}, \quad T_g = \frac{\sum_i T_i v_i}{\sum_i v_i}, \quad P_g = \frac{\sum_i P_i v_i}{\sum_i v_i}, \quad Y_{ig} = \frac{\sum_i Y_i \rho_i v_i}{\sum_i \rho_i v_i} \quad \text{Equation 14}$$

Where ρ_g , T_g , P_g and Y_{ig} are the density, temperature, pressure and mass fraction of group g.

Therefore, the reaction rate of each species in group g is calculated according to the following equation:

$$\omega_{ig} = \rho \frac{\partial y_{ig}}{\partial t} \quad \text{Equation 15}$$

Finally, the mass fraction of all species in a cell can be calculated from:

$$y_i^{n+1} = y_i^n + \delta t * \frac{y_{ig}^{n+1} - y_{ig}^n}{\delta t} \quad \text{Equation 16}$$

As for ignition, this is defined in terms of the spark period, its position, the size of the spark kernel and the ignition temperature. The latter is applied to the reaction rate calculation inside the spark kernel.

Chapter 5. CFD analysis of Turbulent Jet Ignition in a divided chamber bomb

5.1. Introduction

The combustion propagation within the pre-chamber, the main-chamber and the orifices is strongly affected not only by the local turbulence but also by the chemical characteristics of the mixture. Consequently, detailed chemical kinetics use is widespread for TJI systems. In addition, due to the quiescent initial state and the ambient conditions which concern the analyzed case, Detailed Chemistry model is incorporated in the model setup for combustion modelling. Finally, the analysis of the combustion regimes concerning the flame propagation within the pre-chamber and the orifice, which will be described in depth hereafter, make the Detailed Chemistry model the most reliable and robust approach for this type of application.

5.2. Experimental conditions

The divided chamber (also named “bomb”) investigated in the present study is a closed volume combustion vessel described in detail by Yamaguchi et al. [102] and schematized in Figure 34. It consists of a cylindrical MC, with 80 mm of inner diameter and 30 mm of width, and a cylindrical PC, with either 35.7 or 25 mm of inner diameter and same width as the MC. The volume ratio between the tested PC and MC (termed α) is 0.1 or 0.2, which is notably larger than modern type pre-chamber SI engines. The two chambers are connected through a central orifice of 23.7 mm length and different diameters (4, 5, 6, 7, 8, 9, 10, 12 or 14 mm). The divided chamber design and the optically accessible window rendered the study in [102] a noteworthy example of combustion analysis sharing the same principles and rate-determining factors as modern spark-ignited pre-chamber systems. A single pressure probe is located on the end wall of the MC and a spark ignitor is mounted on the opposite side in the PC. In the simulated cases the divided quiescent chamber is

filled by a slightly rich homogeneous propane-air mixture ($\Phi = 1.1$) at room pressure and temperature (1 bar and 300K). The most important differences with respect to modern pre-chamber combustion systems are the large PC/MC volume ratio and the quiescent combustion in the PC at ignition; nevertheless, the quality of the combined pressure- and optically-based investigation in [102] justified the choice of this test case.

From the analysis of the experimental data (Figure 35) [102], a clear connection emerges between the orifice area and the slope of the pressure rise, which is strongly related to combustion duration and tends to increase with larger orifice diameters. When larger nozzles are used, jet outflow appears to be little affected by the orifice diameter. Furthermore, a peculiar behavior of the smallest orifice case is detected: it is in fact the only case showing a very delayed jet exit. Such phenomenon emerges even more clearly when a larger α is adopted. These considerations were made also in [102], where ignition was classified into four different types based on the chemical and physical characteristics of the torch jet. The authors justified the more explosive behavior of the case with the smallest orifice by means of the combined effect of turbulence and fully dispersed active chemical species.

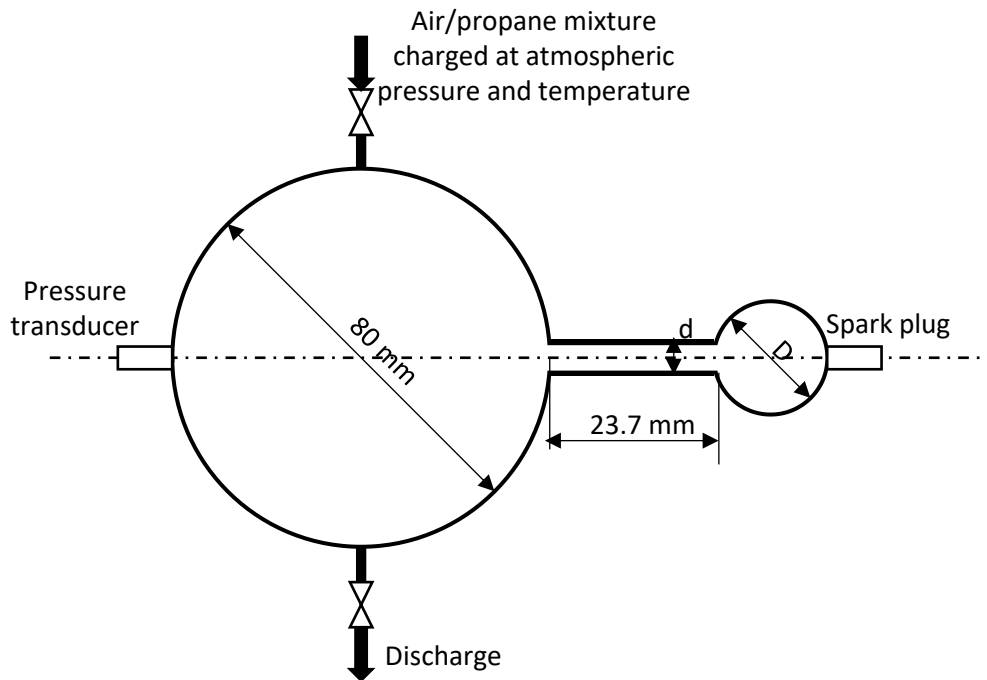


Figure 34 Schematic diagram of the divided chamber bomb elaborated from [102].

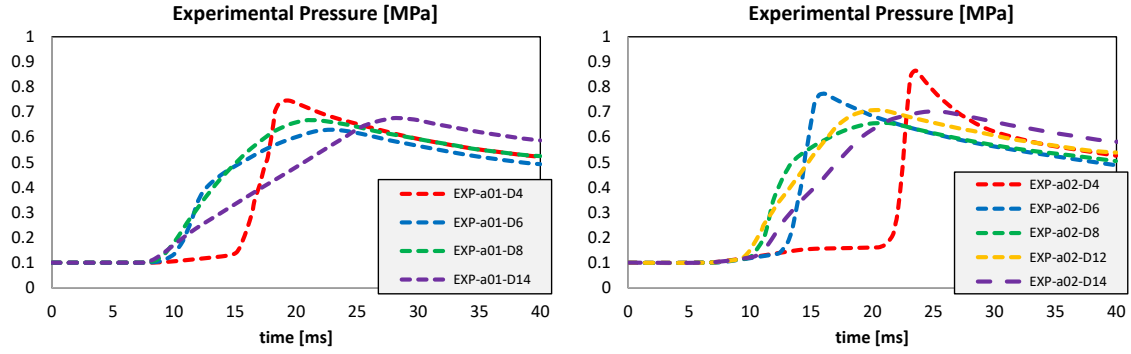


Figure 35 MC pressure trace for $a=10\%$ and 20% in the uniform charge, nozzle diameter ranges from 4mm (D4) up to 14mm (D14), elaborated from [102].

5.3. Numerical approach

Coherently with the experimental data, a physical duration of 40 ms is considered in the simulations. Spark ignition occurs at the beginning of the simulation. Thanks to the two symmetry planes of the real geometry, only a quarter of the domain is modelled to reduce the computational effort. Dictated by experience with similar pre-chamber combustors and RCEMs, a hexahedral computational grid with a base cell size of 1 mm is used. A cylindrical refinement is adopted withing the PC, the orifice and the portion of the MC exposed to the jet outflow, locally reducing the cell size to 0.5 mm. An analysis of the grid resolution within the nozzle is carried out and reported below, highlighting the importance of a double block refinement, reducing the cell size to 0.25 mm. The near-wall grid makes use of a single layer of 1 mm according to the high-Reynolds approach. In the refinement block, the single wall layer reduces to 0.5 mm, while a layer thickness of 0.25 mm is used at orifice wall. The total number of cells is approximately 320'000, with minor variations with the orifice diameter or the chamber volume ratio. From a time-step sensitivity analysis which is reported below, a constant timestep of 8.3×10^{-7} seconds is used in most of the simulations while the timestep is halved when ignition occurs. The PISO algorithm is used as implicit time integration method, while a second order discretization scheme is used for momentum, turbulence, energy, and species.

Consistently with the experimental setup, the flow in both chambers is initially quiescent at room conditions for all cases. The orifice, MC and PC wall temperature is set to 300 K,

with a no-slip boundary condition for velocity. A homogeneous air/propane mixture ($\Phi = 1.1$) is initialized in the whole domain. All 3D simulations are performed within a RANS framework, using the k- ϵ RNG turbulence model, which is one of the most widely adopted models in the in-cylinder community. The GruMo-UniMORE wall heat transfer model formulation [103, 104, 105] is used in all cases. By doing so, the heat transfer model is slightly modified to account for variations in the Prandtl number as it follows instantaneous local conditions.

The C3-mechanism for propane combustion used in the current study is the one proposed by Qin et al. [106], consisting of 70 species and 463 reactions. For this application, the adopted DMZ method makes use of 4 flow variables chosen for clustering, i.e. temperature (1360 groups), enthalpy (200 groups), mixture fraction (1000 groups), and hydrogen mass fraction (100 groups).

Six virtual probes (Figure 36) are used to measure local values of turbulence, pressure, temperature, and velocity. Probe1 coincides with the spark electrodes position, while Probe2, Probe3 and Probe4 are used to measure the flow characteristics inside the orifice. Probe5 is set 5 mm downstream from the nozzle exit on the axis, consistently with the ionization probe used in the experiment. Probe6 position corresponds to the experimental pressure transducer position.

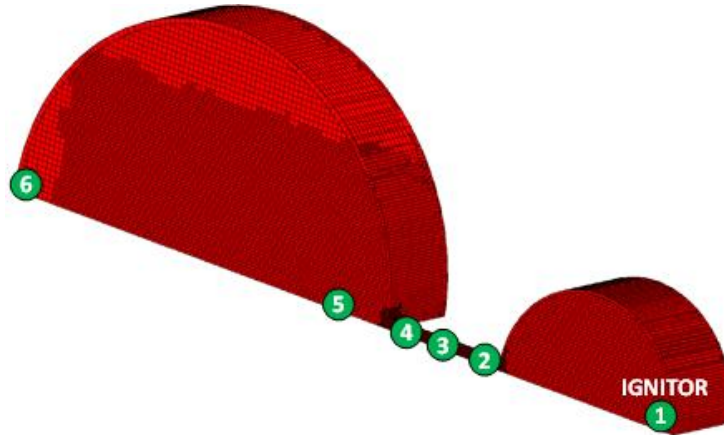


Figure 36 Probes and Computational domain, a quarter of the whole geometry is simulated.

To compare numerical results with experimental data, the pressure trace is processed similarly to the most common real-engine applications. The sampling covers 40 ms in accordance with the experimental pressure trace acquisition duration. The burnt fraction

(X_b) is calculated as in the following equation, which derives from the 1st law of thermodynamics, properly simplified for a constant volume case:

$$X_b = \frac{\frac{\sum_0^{40ms}(V_i * dP_i)}{k-1}}{MAX(\frac{\sum_0^{40ms}(V_i * dP_i)}{k-1})} * 100 \quad \text{Equation 17}$$

where k is the specific heat ratio, here considered constant and equal to 1.3, dP_i is the pressure variation over time, V_i is the total volume of the domain.

To further investigate the flame propagation through the nozzle the progress variable and the chemical reaction zone are re-calculated.

$$\tilde{c} = 1 - \frac{Y_{fuel}}{Y_{fuel_init}} \quad \text{Equation 18}$$

$$ReactionZoneScalar = \tilde{c} * (1 - \tilde{c}) \quad \text{Equation 19}$$

where Y_{fuel} is the fuel concentration and Y_{fuel_init} is the initial fuel concentration. In each cell, the progress variable ranges from 0 to 1, respectively for unburnt and fully-burnt mixture, thus it is representative of the flame propagation itself. As for the *ReactionZoneScalar*, it is equal to 0 when the $\tilde{c}$ is 0 or 1 and it reaches its maximum value 0.25 when $\tilde{c}$ is 0.5, i.e. half of the fuel within the cell is burnt. In this way, the *ReactionZoneScalar* returns a qualitative distribution of the reaction zone.

In addition, an analysis of the Damköhler number is performed, which is used to weight the contribution of turbulent mixing and of chemical reactions. The Damköhler number Da is calculated as follows, using the length scale $\left(\frac{l_t}{\delta}\right)$ and the velocity scale $\left(\frac{s_l}{u'}\right)$ ratio.

$$Da = \frac{\frac{l_t}{\delta}}{\frac{s_l}{u'}} = \frac{l_t}{\delta} \cdot \frac{s_l}{u'} \quad \text{Equation 20}$$

The integral length scale l_t and the turbulence intensity u' equations are reported below:

$$l_t = 0.09^{0.75} \cdot \frac{k^{1.5}}{\varepsilon} \quad \text{Equation 21}$$

$$u' = \sqrt{\frac{2}{3}k} \quad \text{Equation 22}$$

The laminar flame thickness δ is calculated using the Zeldovich correlation [107, 108]:

$$\delta = \frac{2 * \lambda}{\rho_{unb} * c_{p,unb} * s_l} \quad \text{Equation 23}$$

Where λ is the thermal conductivity, ρ_{unb} and $c_{p,\text{unb}}$ the density and heat capacity of the fresh gases. s_l is the laminar flame speed, here calculated with a chemistry-based correlation, using a methodology developed in [109, 110, 111] and applied to air-propane mixtures in [112] and equal to approximately 35 cm/s. Therefore, as it can be seen the thickness of the reaction zone depends on chemistry

5.4. Orifice Mesh sensitivity

Chemistry-based combustion models are usually numerically expensive due to the large number of transported chemical species. In the specific case of TJI systems, computational cost is made even more demanding due to the high velocities reached inside the nozzles, imposing reduced timesteps comply with the Courant-Friedrichs-Lewy number constraint thus obtaining a good representation of the flow field inside the nozzle, it is also fundamental to perform a fine spatial discretization. Three cases are carried out varying the cell size within the orifice, set to 0.5 mm, 0.25 mm, and 0.125 mm respectively, as reported in Figure 37. The mesh sensitivity is performed on the geometry with the smallest orifice (4 mm) and the largest volume ratio, named a02-D4 and for which the highest jet velocity is expected. The total number of cells is around 286k (for a cell size of 0.5 mm), 293k (for the intermediate case), and 333k (for the finest case).

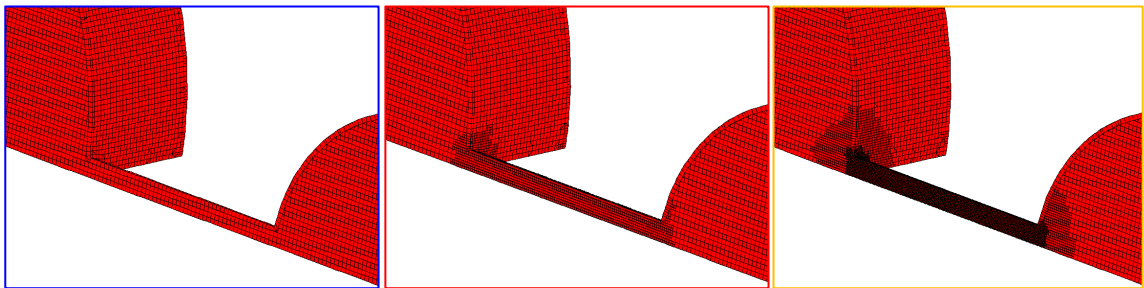


Figure 37 Mesh size inside the nozzle, respectively of 0.5 mm, 0.25 mm, and 0.125 mm.

The numerical pressure trace is computed at the MC probe (n.6), and a high dependency of the orifice flow dynamics on grid density arises. As shown in Figure 38, pressure traces are almost superimposed until the flame develops within the main chamber. Even though each case correctly shows the early pressure rise, only the cases with the lower cell sizes of

0.5mm and 0.25 mm reproduce well the subsequent pressure increase, corresponding to main-chamber ignition and combustion. A deeper comparison between the finer cases is performed considering wall y^+ (Figure 39) at the maximum velocity time. Low values, i.e. wall y^+ around 10, are found over the nozzle wall when the finest mesh is used. Such grid strategy may lead to potential issues since even lower values are expected when the fluid velocity is lower; this in turn leads to potential errors in the modelling of heat transfer and velocity profiles, which can be considerable reliable only once the wall y^+ falls within the log-law region, i.e. wall y^+ higher than 30. Moreover, the 0.125 mm case further increases the computational cost of the simulations because of the higher number of cells and the lower time-step required to comply with the Courant-Friedrichs-Lewy number constraint, especially during the nozzle outflow.

Therefore, the cell size of 0.25 mm represents the best trade-off amongst grid finesse, computational cost and heat transfer and velocity profiles representation within the orifice and such value is finally chosen in the present study.

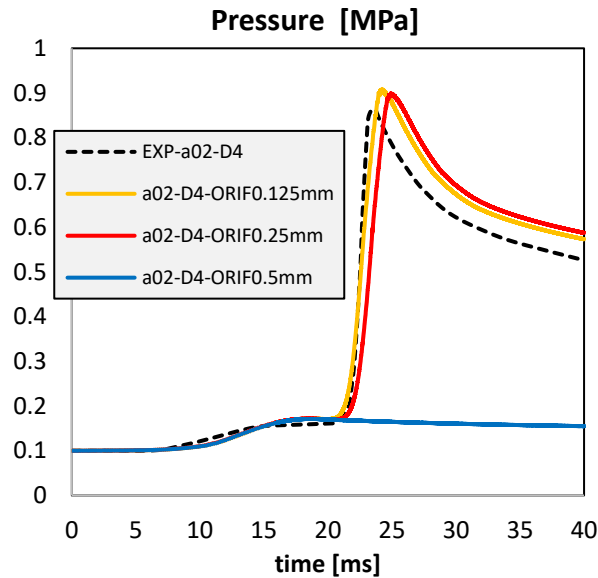


Figure 38 Experimental and numerical pressure traces for $a=20\%$ and $d=4\text{mm}$ varying the nozzle cell size, measured at main chamber probe.

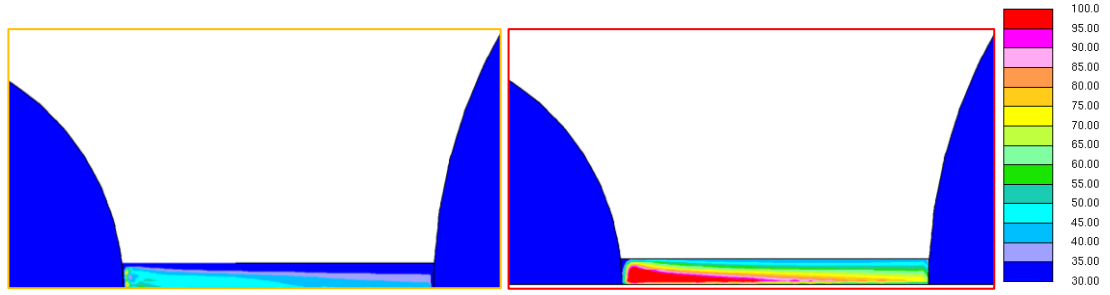


Figure 39 Wall Y^+ when the maximum velocity is reached inside the nozzle, i.e. $t=14$ ms. Cell size of 0.125mm on the left; 0.25mm on the right.

5.5. Time-step sensitivity

As for the time-step, the rule of thumb usually is to adopt a time-step around $1e-6$ s during the most “delicate” phases, that in a real-engine simulation are valves opening/closing windows, spray injection, combustion, or jet exit in TJI applications. In this case, starting from an even lower value, i.e. 8.333×10^{-7} s, a time-step sensitivity analysis is performed progressively increasing the time-step size.

As depicted in Figure 40, using a timestep of 8.333×10^{-7} s the average Courant number in the case with the largest pre-chamber and the smallest nozzle (i.e. the highest velocities within the nozzle itself) remains sufficiently low, with maxima equal to 0.22 during the jet ejection and 0.28 when the MC pressure overcomes the PC one, causing a fluid backflow. However, increasing the time-step by two or three times, the average Courant number increases dramatically. On top of that, as shown in Figure 41, higher time-steps result in a loss of information regarding the two slope variations visible in the experimental pressure trace, which nature will be later described in depth. Based on the above, the time-step of 8.333×10^{-7} s is chosen in the present study.

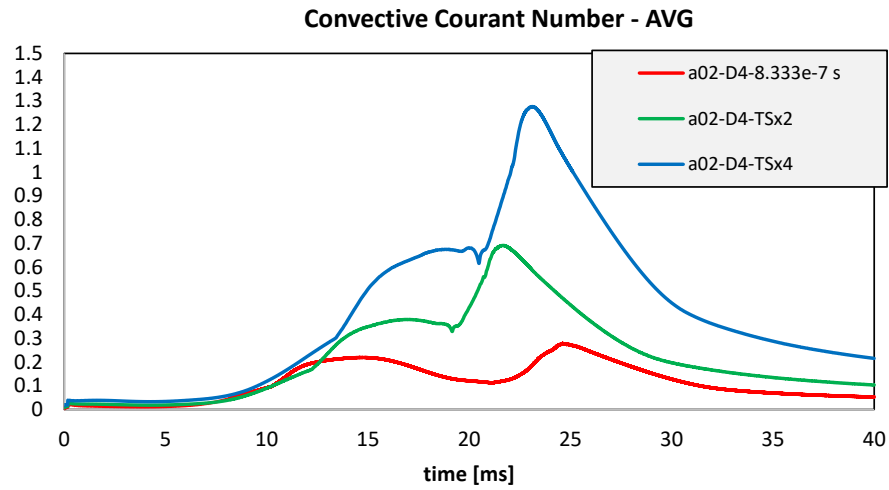


Figure 40 Average Courant number for $a=20\%$ and $d=4\text{mm}$, varying the time-step.

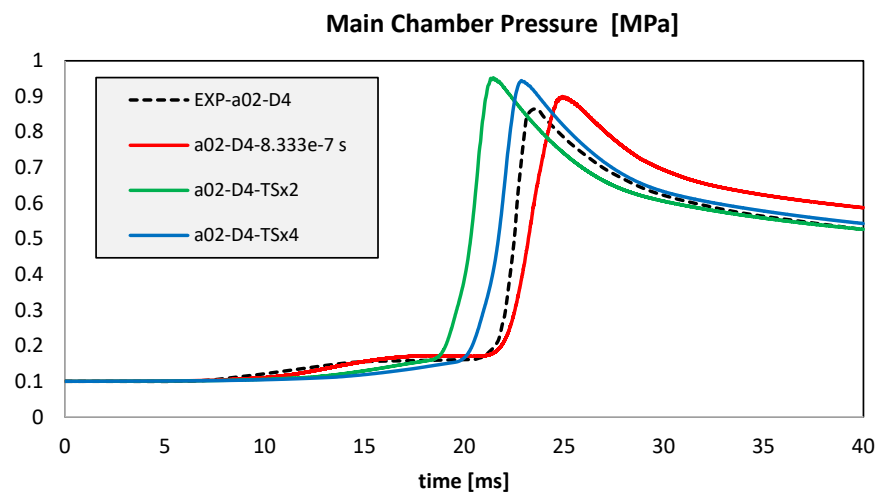


Figure 41 Experimental and numerical pressure traces for $a=20\%$ and $d=4\text{mm}$ varying the time-step, measured at main chamber probe.

5.6. Orifice chocking occurrence

A preliminary analysis is performed based on the highest PC/MC volume ratio and the smallest orifice diameter (a02-D4). Adopting a kernel radius of 1 mm a satisfactory agreement between experimental and numerical ignition delays is found and the pressure rise phasing is well reproduced (Figure 38, orifice grid size 0.25 mm). Once the combustion propagates from the PC to the MC it is hard to determine the nature of the outflowing jet, since a partially quenched flame regime can be expected as a results of intense strain rate and wall interaction during orifice crossing. The experimental pressure trace shows a rapid pressure drop, due to heat exchange, shortly after reaching 90% of fuel burnt, which is well reproduced by the CFD model.

Combustion analysis (Figure 42 and Figure 43) shows a good agreement with the experimental data in terms of both ignition delay and combustion duration, respectively calculated as 0-10% and 10-90% of fuel mass fraction burnt. It is highlighted that the two largest indicators are MFB5 and MFB10 (occurring at 13 and 21 ms, respectively), corresponding to the two slope variations along the pressure trace. This phenomenon affects only this specific case and it will be further investigated hereafter.

As it can be noted, once the maximum heat release rate is reached, the numerical trace appears to be overestimated. This is also visible in the other cases described below and can be attributable to an underestimation of the heat flux to the walls.

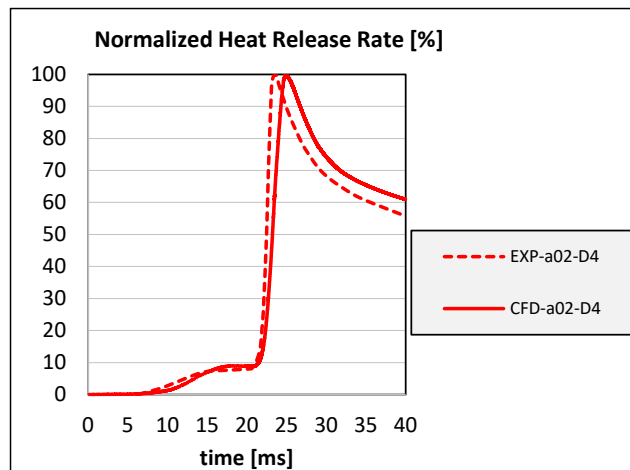


Figure 42 Heat Release Rate normalized @maximum value. Experimental and numerical traces for $\alpha=20\%$ and $d=4\text{mm}$, based on pressure measured at main chamber probe.

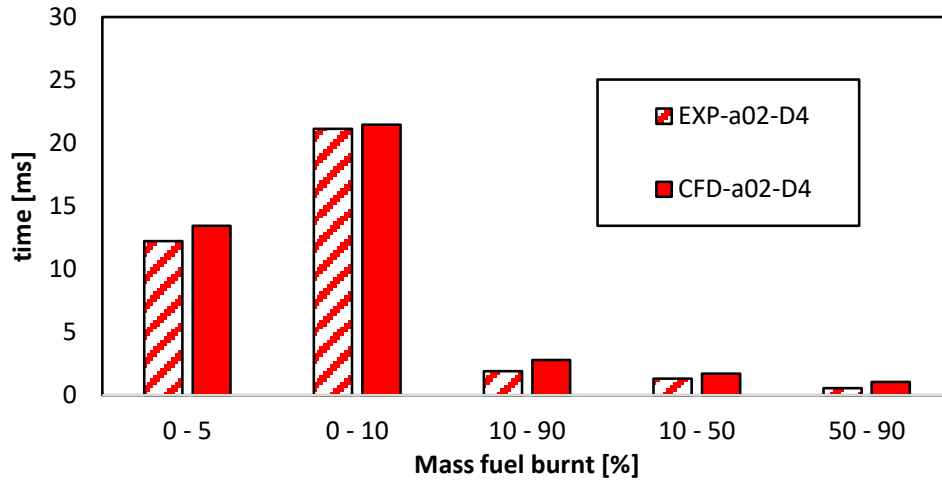


Figure 43 Ignition Delay and Combustion Duration, $a=20\%$ and $d=4\text{mm}$.

The analysis of Mach number in Figure 44 reveals a relatively long phase (approximately 3 ms) in which $Ma > 1$ is found within the orifice. The related flow choking concerns the nozzle portion facing the MC, i.e. where the highest velocities are reached. In this stage the mass flow rate passing through the nozzle is physically blocked, preventing any further increases. This phenomenon is further investigated analyzing the local quantities measured by the Probe n°3 within the nozzle, as reported in Figure 45.

As it is shown in Figure 46, the heat release rate trace, which is re-calculated based on the average pressure within the domain, is strictly related to the choking phenomenon affecting the nozzle. A small amount of heat is released as the flame burns within the pre-chamber and advances towards and inside the orifice. As the choking phase occurs, partially burnt and reacting species flow into the MC and diffuse. In this phase the heat released progressively reduces up to almost zero since the effect of the walls, which are at set at room temperature, prevails over the heat released by the combustion process. As soon as the choking phase ends, the main portion of combustion is very short due to the highly reactive species already spread within the main chamber.

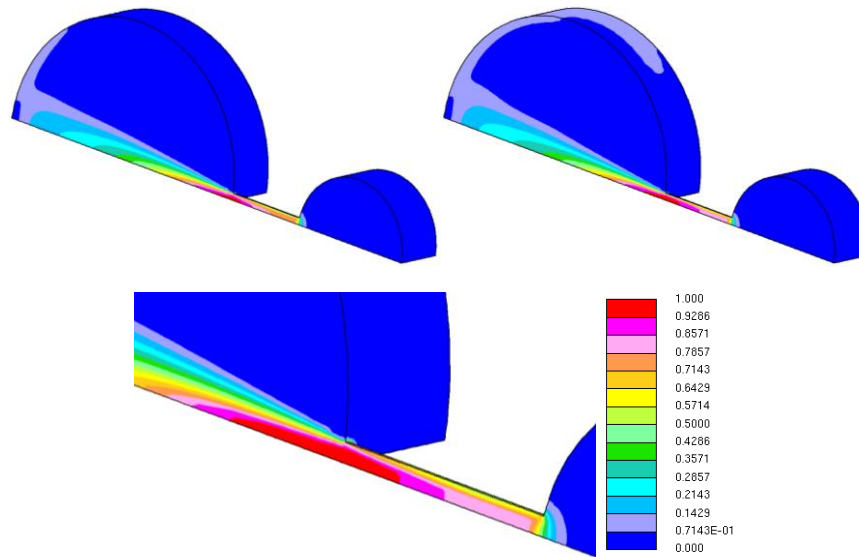


Figure 44 Mach number during chocking onset and end, respectively at 11 and 14ms. Below, a detail of the orifice during maximum chocking intensity at 12,5 ms. $a=20\%$ and $d=4\text{mm}$

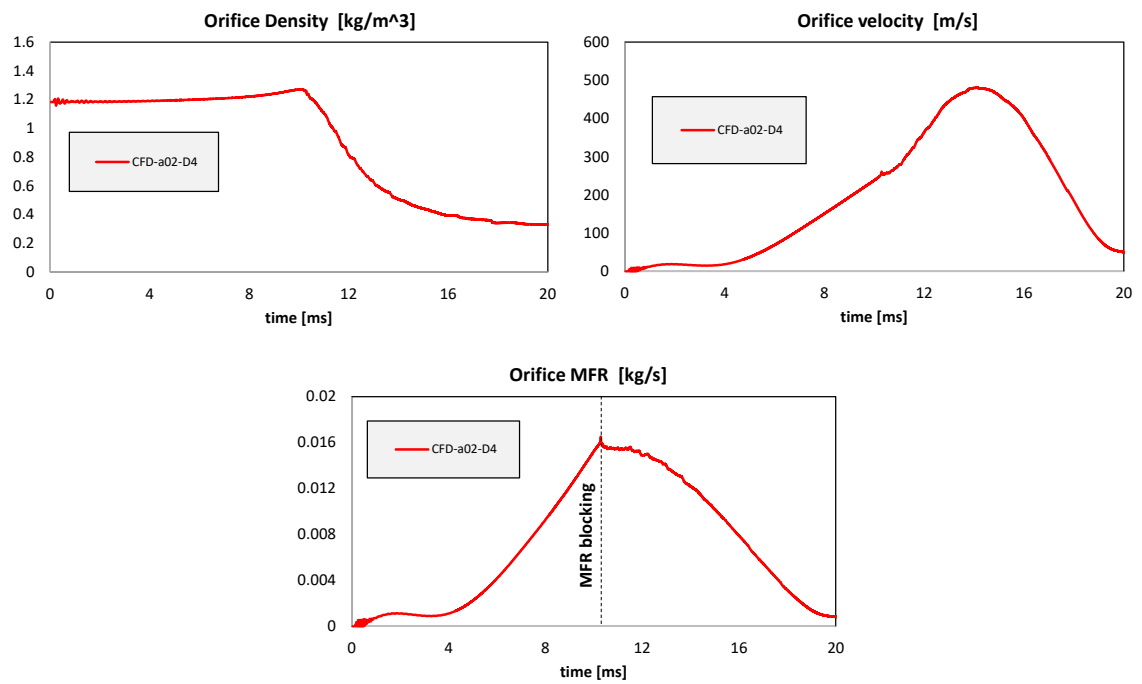


Figure 45 Velocity, density and MFR measured at Probe3 inside the nozzle, $a=20\%$ and $d=4\text{mm}$.

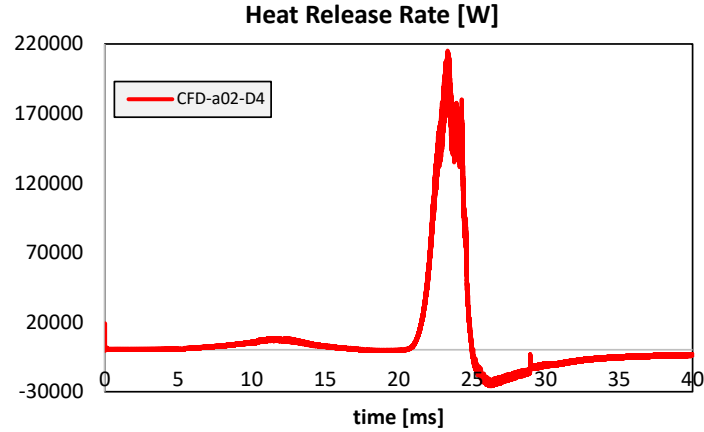


Figure 46 Heat Release Rate calculated based on average pressure trace, $\alpha=20\%$ and $d=4\text{mm}$

As depicted in Figure 47, a strict correspondence between the choking within the orifice and the combustion parameters MFB5 and MFB10 can be found. In particular, the hot outflowing jet from the PC can be correlated to MFB5. From this point on, orifice choking occurs while the partially and fully-burnt combustion products spread within the MC and, being highly reactive, increase the ignitability of the local mixture. The choking phase lasts several milliseconds until, after 21 ms, i.e. after the 10% of fuel burnt is reached, the MC mixture ignites. Due to the simultaneous effect of fully dispersed active species and turbulent jet jets, combustion durations 10-50 and 50-90 are quickly covered (Figure 48). Figure 49 and Figure 50 show the ReactionZoneScalar throughout the simulation for a deeper insight into the choking phase which concerns this case.

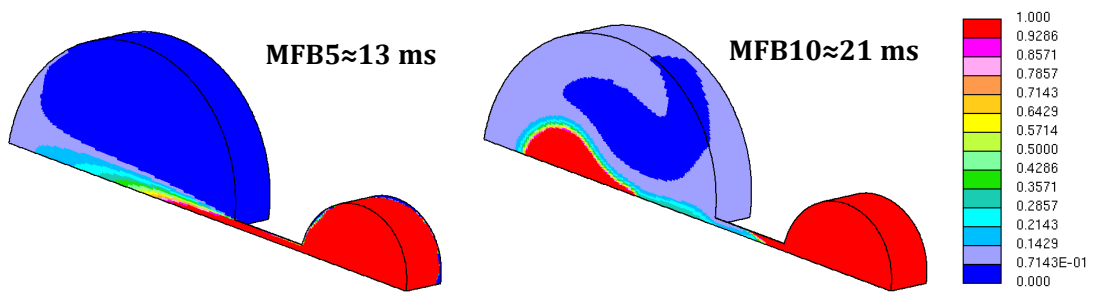


Figure 47 Progress variable at MFB5 and MFB10, coincident with the pressure changes in slope, $\alpha=20\%$, $D=4\text{ mm}$

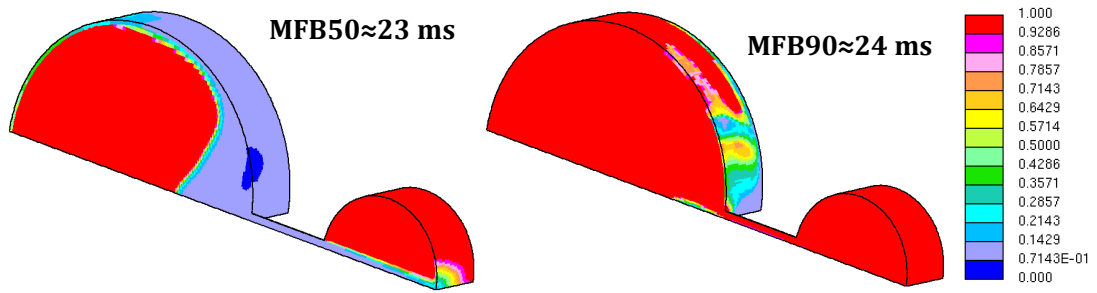


Figure 48 Progress variable at MFB50 and MFB90, $\alpha=20\%$, $D=4$ mm

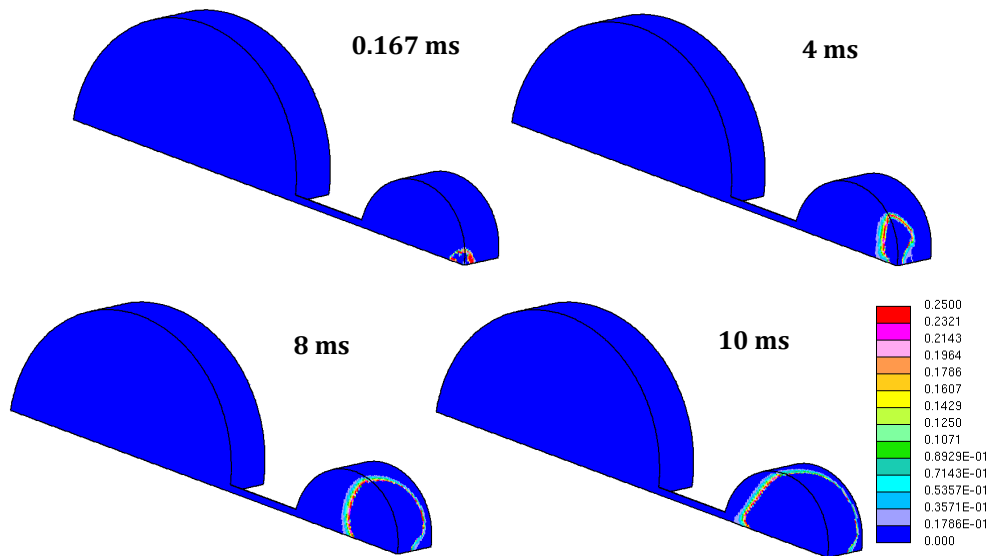


Figure 49 Pre-chamber chemical reaction front, $\alpha=0.2$ and $d=4$ mm case, the color scale also applies to next figure

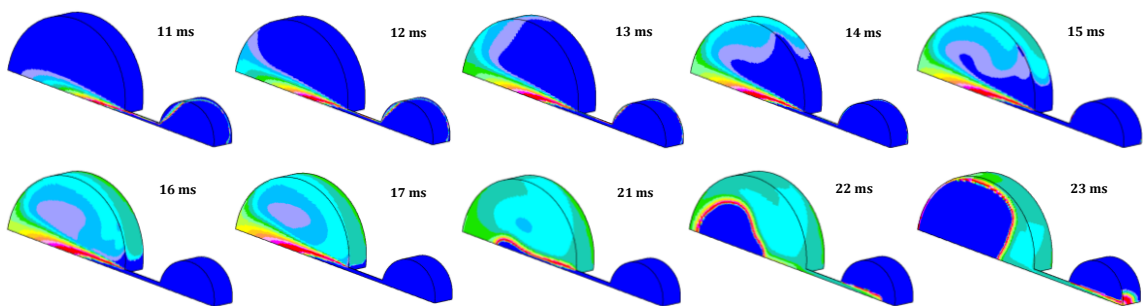


Figure 50 Main chamber chemical reaction front, $\alpha=20\%$ and $d=4$ mm case. Once the choking phase occurs, the chemical species spread into the main chamber, strongly increasing the reaction zone area and facilitating the subsequent flame ignition.

A deeper analysis of the reaction zone features is performed. A subset of the cells with $0.05 < \tilde{c} < 0.95$ is isolated and mass-average quantities on the flame brush are calculated. As shown in Figure 51, due to the ambient quiescent conditions during the early combustion stage, low Da are initially found within the pre-chamber. Later, a linear-type increase of Da is observed, corresponding to the flame development in the PC. Subsequently the flame experiences the orifice flow, strongly increasing the turbulence encountered by the flame inside the nozzle itself and causing a $Da < 1$ condition shortly before the jet ejection. In the final stage, the reactants diffuse in the MC and a $Da > 1$ regime shift.

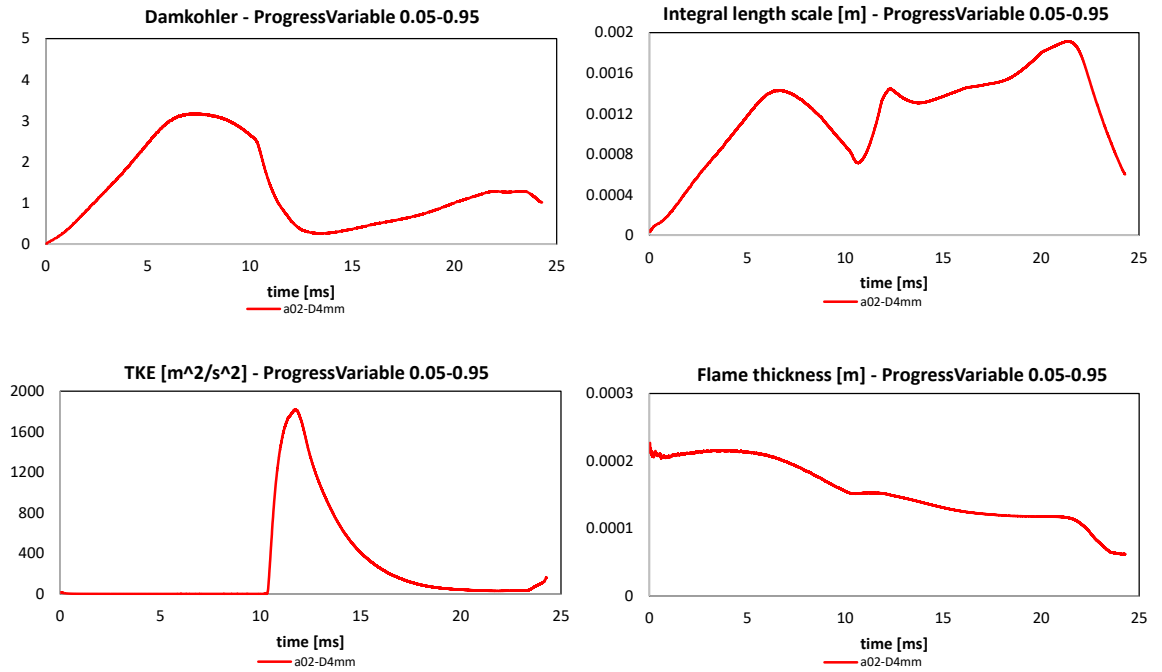


Figure 51 Average quantities seen by the flame, calculated based on the progress variable between 5% and 95%, $a=20\%$ and $D=4mm$.

The Borghi-Peters diagram [113, 114, 115, 116] is an important tool to evaluate the flame regime evolution during the combustion process (Figure 52). An investigation of the flame regime is performed via post-processing the length and velocity scales ratio on the flame brush. This analysis reveals that most of the PC combustion develops in the well-stirred reaction regime ($Da < 1$). During the early flame growth, the reaction kinetics accelerates as temperature increases. This leads to a Da increase, moving the turbulence-chemistry interaction towards a thin reaction zone condition [117]. In the following phase, the flame regime reverts to a well stirred reactor type, due to the high turbulence generated

during the jet outflowing. From here on, the combustion completion lies in proximity of the $Da = 1$ line. Motivated by the above analysis, a chemistry-based combustion model seems to be the most suitable choice when simulating combustion originating from quiescent PC, suggesting that the chemistry rate has a governing role in this case.

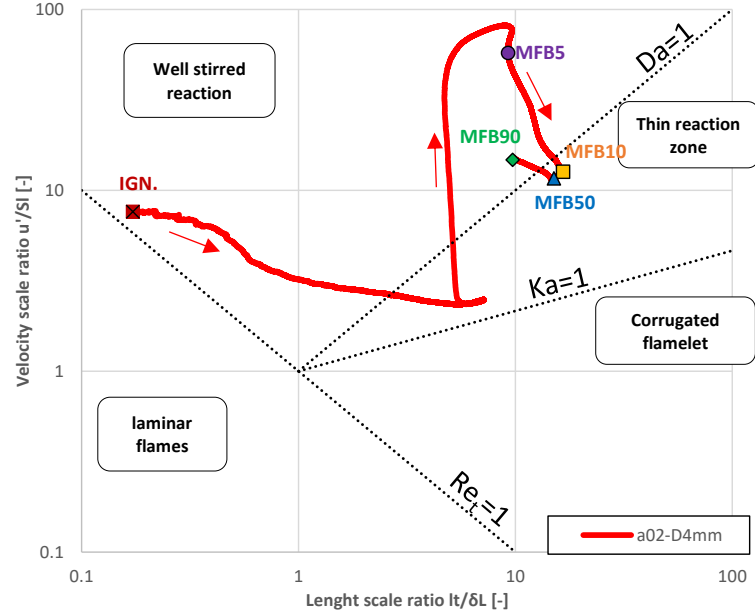


Figure 52 The combustion outcome of the a02-D4 case is presented in the Borghi-Peters diagram.

5.7. Orifice diameter analysis

Equal ignition/combustion chemistry-based setup was applied progressively increasing the orifice diameter until 14 mm, while preserving the same PC/MC volume ratio, i.e. 20%. As mentioned, in the experimental data the two slope changes are observed only for the D4 case, and the jet outflowing phasing is generally advanced compared to the previously analyzed case, pointing out the absence of flow choking. In addition, faster combustion development is observed for the narrower nozzle cases. As depicted in Figure 53, from the analysis of the pressure traces it is notable that the anticipated jet outflowing from the smallest orifices is well predicted also by the adopted model setup. As soon as the orifice diameter is large enough to avoid choking, the jet outflowing seems to be less affected by the nozzle area variation: all the cases with $D \geq 6 \text{ mm}$ show only one slope change of the pressure trace and all of them are similarly phased.

From the burnt fraction analysis reported in Figure 54 and Figure 55, several considerations can be made on the sensitivity to the orifice diameter. The time delay between 5% and 10% of fuel burnt is very short for all other cases (except for the nozzle-choked a02-D4 case), and both 5% and 10% duration indicators are quantitatively aligned with experimental data. The late-combustion indicators (10-50% and 10-90%) increase for larger orifice diameter, again in agreement with the experiments. It is to be noted that the predictive capability of the model is not optimal for large D cases.

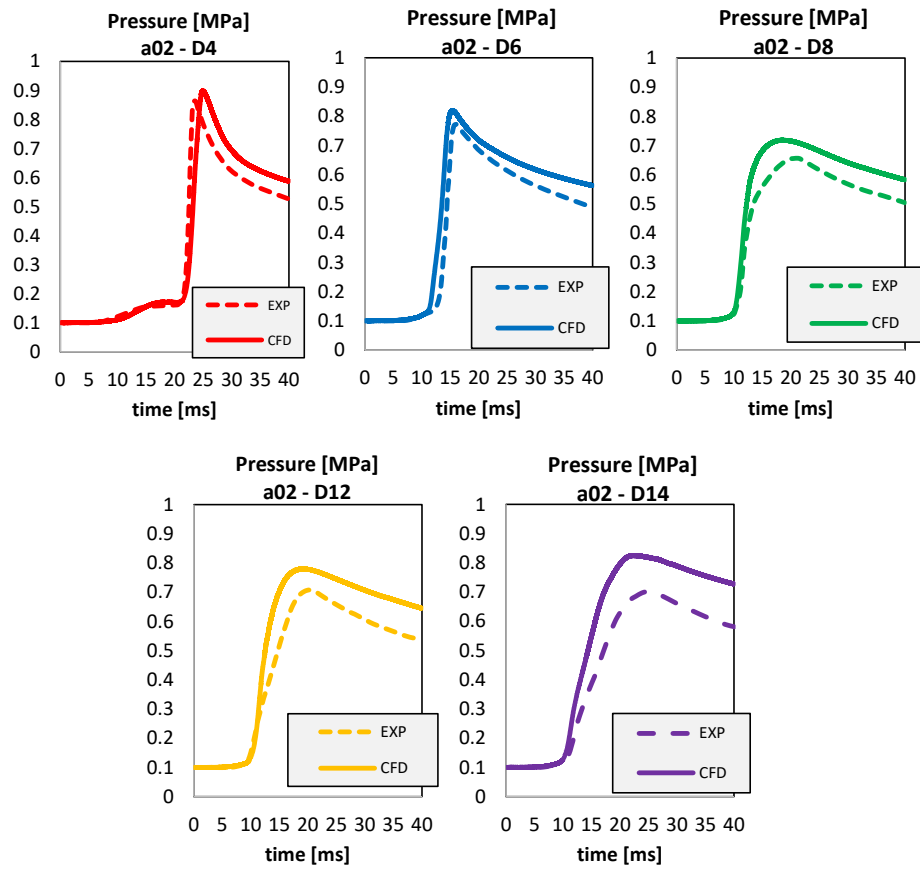


Figure 53 Experimental and numerical pressure traces for $a=20\%$, measured at main chamber probe.

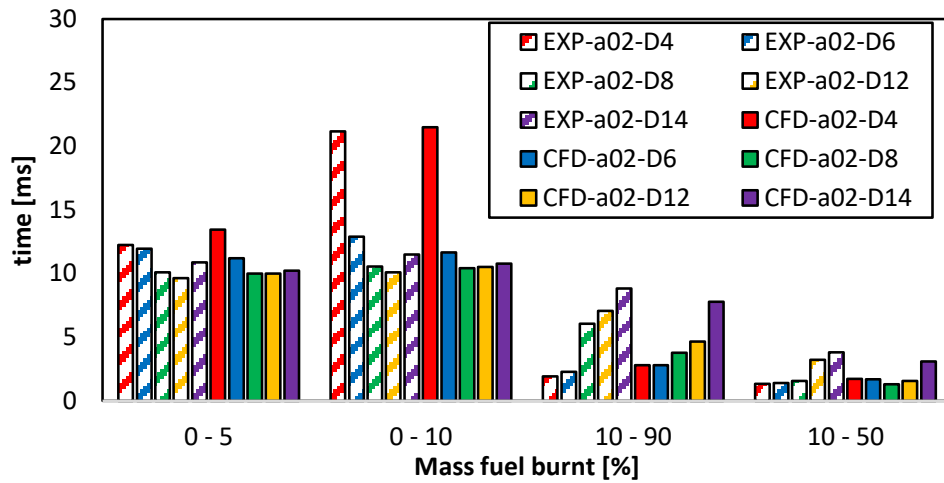


Figure 54 Ignition Delay and Combustion Durations for different orifice diameters, $a=20\%$.

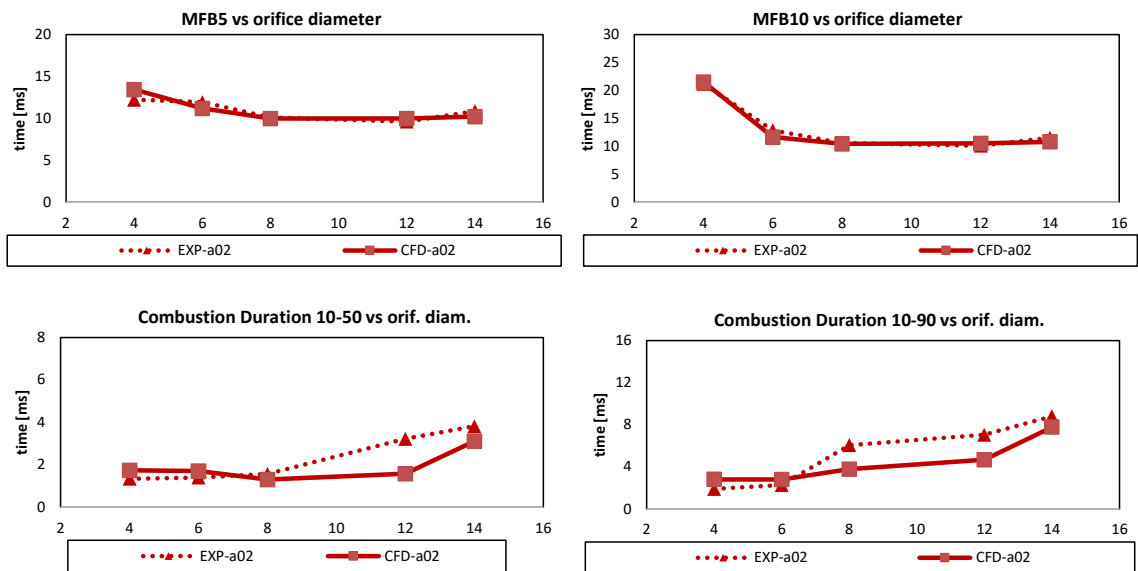


Figure 55 Ignition delay and Combustion durations varying the orifice diameter, $a=20\%$.

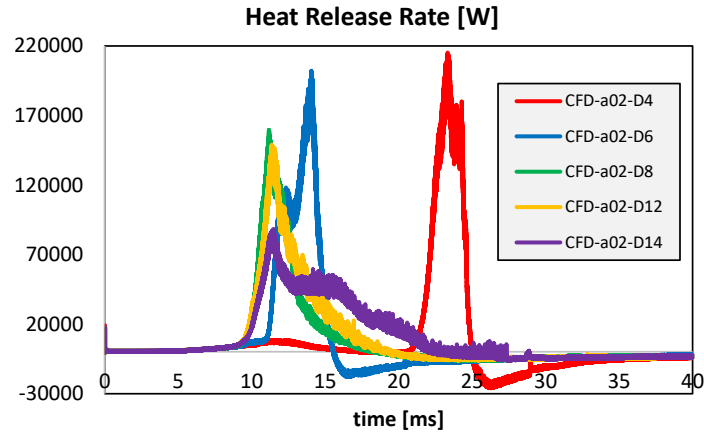


Figure 56 Heat Release Rate calculated based on average pressure trace, $a=20\%$

Figure 56 summarizes what has been said so far and, as it can be seen, the heat release rate trace is strictly related to the orifice diameter. Particularly, increasing the nozzle diameter the heat is released earlier, according to the jet ejection phasing, and less impulsively.

A comparison of the flame shape at MFB10 is shown in Figure 57 and Figure 58. A strong correlation between the jet front structure and the orifice area is notable: as the orifice diameter decreases, the flame reaches higher velocities inside the nozzle, resulting in longer penetrating jets. Also, the spread of chemical species which characterizes the smallest orifice, promoted by the long choked-flow phase, is not visible in the larger nozzle cases on equal burn duration. The $D = 4 \text{ mm}$ case is both the smallest orifice one and the only choked case, resulting in the lowest mass flow rate and the wider dispersion of reactive species in the MC.

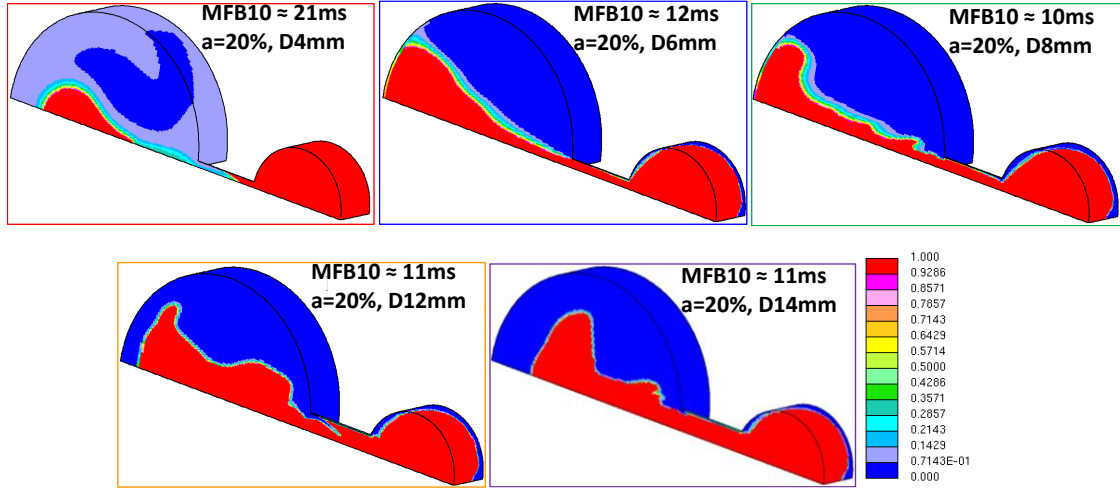


Figure 57 Progress variable at MFB10 for orifice diameters of 4mm, 6mm, 8mm, 12mm and 14mm, $a=20\%$

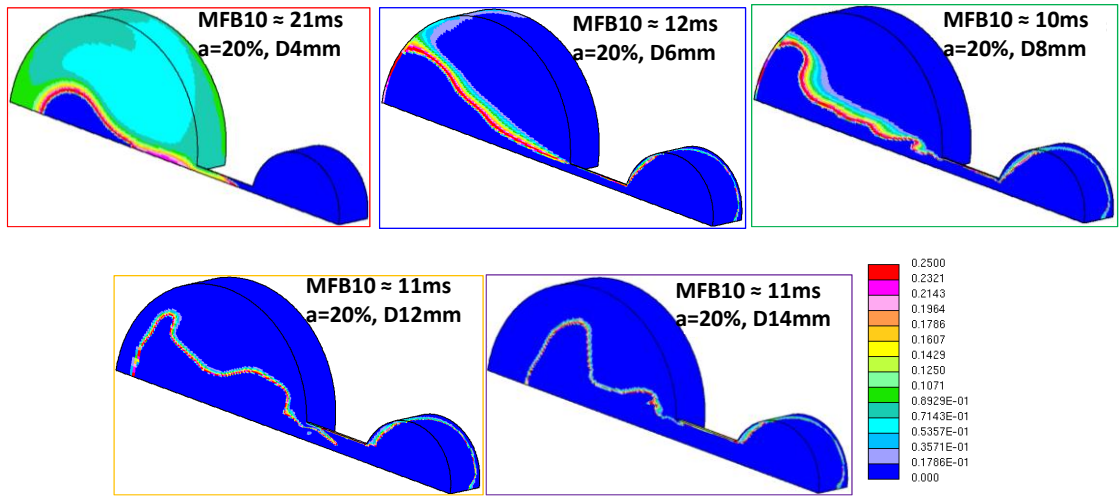


Figure 58 Reaction front at MFB10 for orifice diameters of 4mm, 6mm, 8mm, 12mm and 14mm, $a=20\%$

The analysis of the probes located within the domain shows that a $D \geq 6 \text{ mm}$ orifice is sufficient to considerably reduce the pressure unbalance between PC and MC (Figure 59), approaching null values for $D \geq 8 \text{ mm}$ orifices. Consequently, the flow velocity through the nozzle drops significantly with larger nozzles.

As depicted in Figure 60, the turbulent kinetic energy at MFB5 is strongly related to the orifice diameter. Particularly, the larger the nozzle diameter, the lower the turbulence encountered by the flame within the nozzle. A similar scenario may apply to Figure 61, in which a decreasing dissipation rate is shown progressively reducing the orifice diameter.

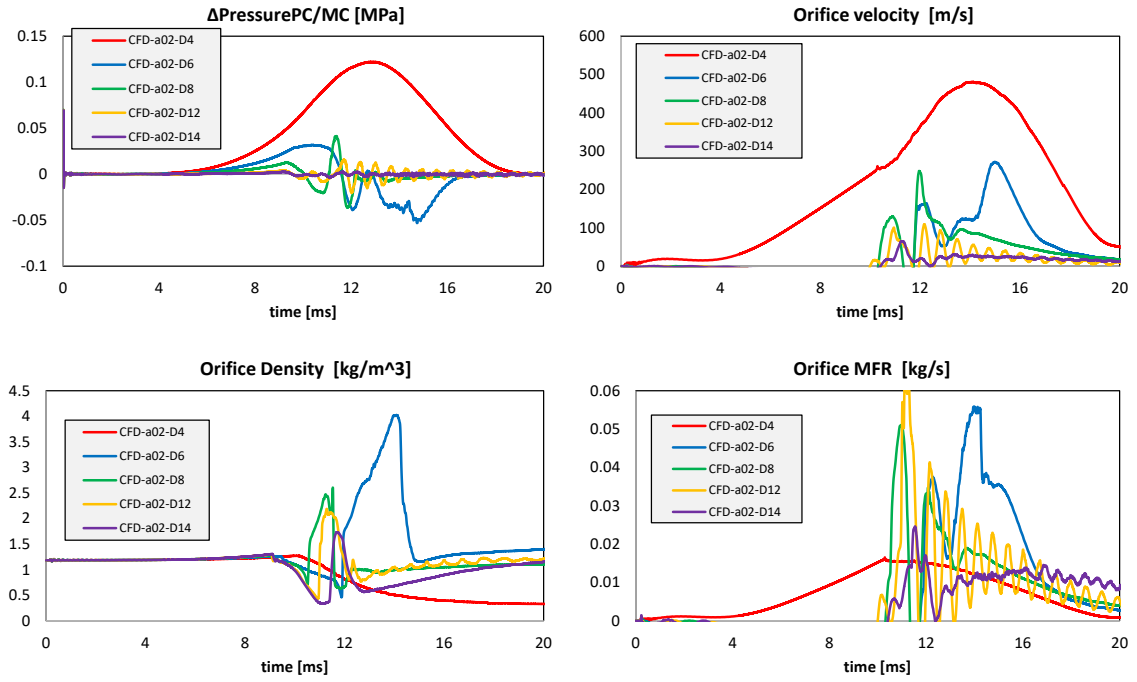


Figure 59 ΔP between the chambers. Axial velocity, orifice density and MFR through the nozzle, $\alpha=20\%$

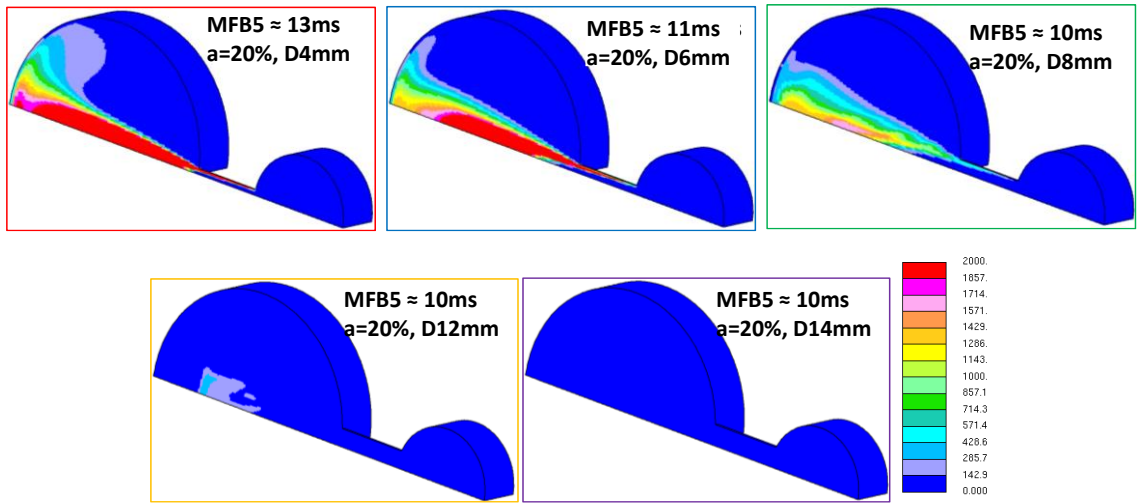


Figure 60 Turbulent Kinetic Energy @MFB5, orifice diameters of 4mm, 6mm, 8mm, 12mm, 14mm, $\alpha=20\%$

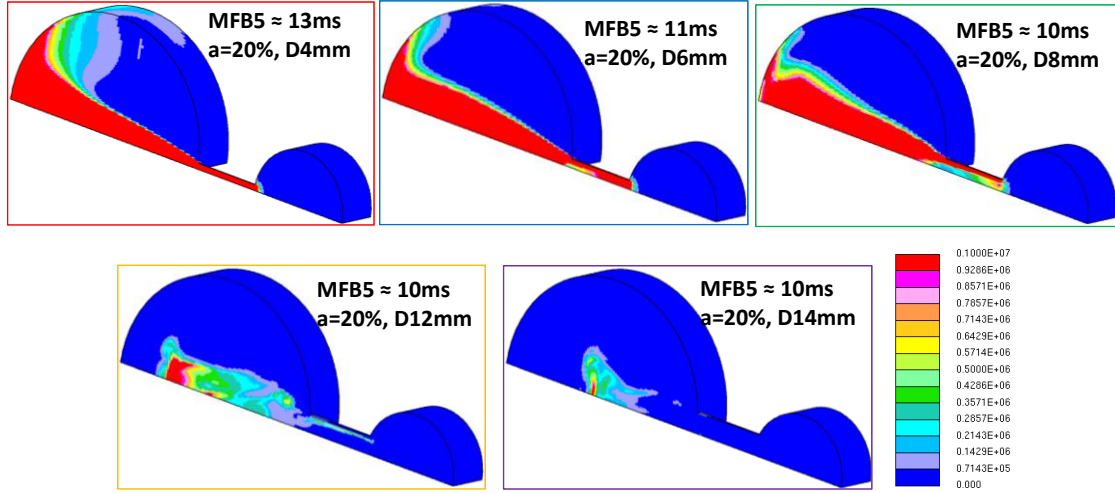


Figure 61 Dissipation rate @MFB5, orifice diameter of 4mm, 6mm, 8mm, 12mm, 14mm, $a=20\%$

The quantities evaluated in the subset $0.05 < \tilde{c} < 0.95$ range are analyzed for all the cases and reported in Figure 62 and Figure 63. Particularly, such quantities are reported as a function of mass fuel burnt for a clearer distinction of what concerns pre-chamber combustion, identified by $MFB < 5 - 10\%$

The Damköhler numbers seen by the flame inside the PC are little affected by the increasing orifice area. As for the turbulent kinetic energy encountered by the flame in the $D = 4 - 6 \text{ mm}$ cases, values one order of magnitude higher than those with the largest nozzles are measured. Due to the very fast combustion of the D4 case, an almost constant average Da is found during its main chamber combustion. Conversely, Da increases during combustion progress for D6-14 cases.

These considerations can be also applied to combustion regime analysis (Figure 64), further confirming the reduced dependency on the orifice diameter on the PC flame regime. The lower turbulent kinetic energy found for large D cases moves the MFB50 and MFB90 towards the thin reaction zone: this could cause the burn duration misalignment with experimental data for these cases. However, most of the combustion process develops in the well-stirred reaction zone, emphasizing the importance of using a chemistry-based combustion model.

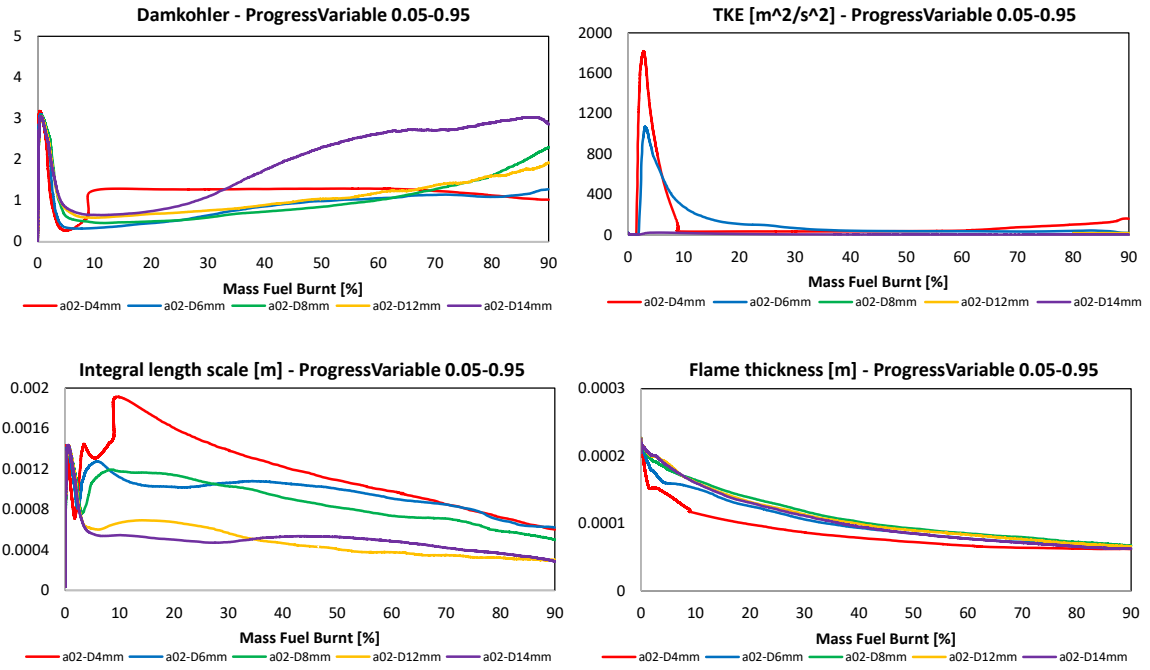


Figure 62 Average quantities seen by the flame, calculated based on the progress variable between 5% and 95%, $a=20\%$

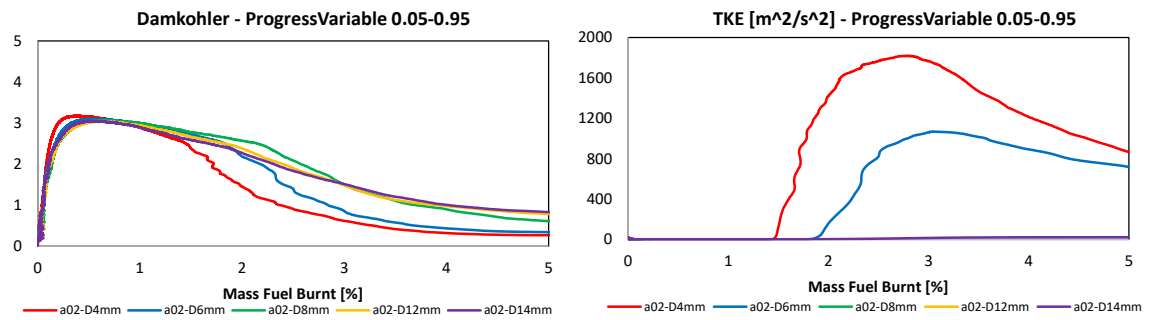


Figure 63 Average quantities seen by the flame until the jet ejection, calculated based on the progress variable between 5% and 95%, $a=20\%$

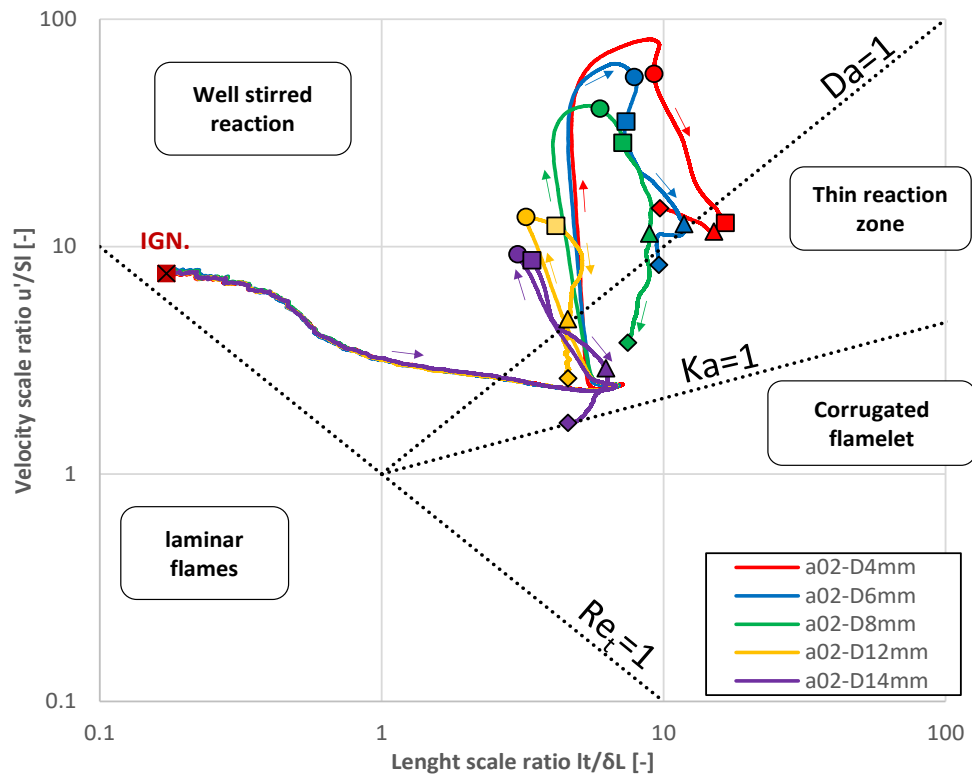


Figure 64 The combustion outcome of all test conditions involving $\alpha=20\%$ are presented in the Borghi-Peters diagram. Circles correspond to MFB5, squares to MFB10, triangles to MFB50 and rhombus to MFB90.

5.8. Chambers volume ratio analysis

A deeper investigation of the effect of the geometrical features of the PC on the flame regime is carried out. The same modelling ignition/combustion setup, already applied for the previously described cases, is also applied for the cases with the lower PC/MC volume ratio, i.e. 10% (Figure 65). The analysis of the experimental data shows an earlier jet ejection, as well as a slower combustion. Also in these cases the D4 configuration (a01-D4) is the only outlier, with a notably delayed jet ejection compared to the larger nozzles.

The numerical pressure traces show that from a qualitatively standpoint the sensitivity on the orifice area is well reproduced by numerical simulations. The delayed jet exit of the a01-D4 case is appreciable, even if a slight underestimation is observed. In this case $Ma < 0.8$, but the delayed jet ejection is motivated by the obstruction of the small-area orifice. A further confirmation comes from the analysis of the burnt fraction (Figure 66): both 5% and 10% combustion indicators are very close even for the D4 case, excluding any evidence of choking conditions. Comparing the numerical results with the experiments in terms of 5% and 10% phasing, a good agreement is found, and the experimental trend is well reproduced by numerical simulations. From a quantitative standpoint, 10-50% burn duration well matches the experimental reference for all cases except a01-D14.

As depicted in Figure 67, the heat release rate, calculated considering the average pressure trace, is closely related to the nozzle diameter and chambers volume. In particular, a smaller pre-chamber results in lower heat release peaks and less steep traces, leading to longer combustion duration. Similar effect can be obtained increasing the orifice diameter which again results in longer combustion.

The analysis of the combustion regime throughout the simulation duration (Figure 68, Figure 69) highlights that the combustion in the PC is still dominated by a well-stirred reaction condition, further motivating the need for a kinetic-driven modelling framework. A small PC volume leads to lower Δp_{PC-MC} , in turn promoting low turbulent kinetic energy values inside the nozzle. The Da values in the MC with $a=10\%$ are generally higher than those for $a=20\%$, and the combustion regime early moves into the thin reaction zone. Unlike cases with $a=20\%$, the combustion phasing (MFB50) of each case now falls into the thin reaction zone. As it can be noted, the end of combustion (e.g. MFB90) of the larger-D cases moves in the corrugated flamelet regime, pointing to the possibility of an out-of-range

condition of the adopted kinetics-driven approach in this case as the reason for the combustion velocity underestimation for the a01-D14 case.

Figure 70 summarizes the comparison of all the simulated cases (a01/02, D4/14) with the experimental counterparts in terms of ignition delay and combustion durations. As can be observed, very good alignment is found for MFB5 and MFB10 indicators, considerably driven by PC combustion. The agreement of combustion velocity is appreciable also in the MC combustion phase with no model tuning, especially for the a02 and low-D cases. Only the a01-D14 case shows a slower combustion compared to the experimental data, for the reasons discussed above.

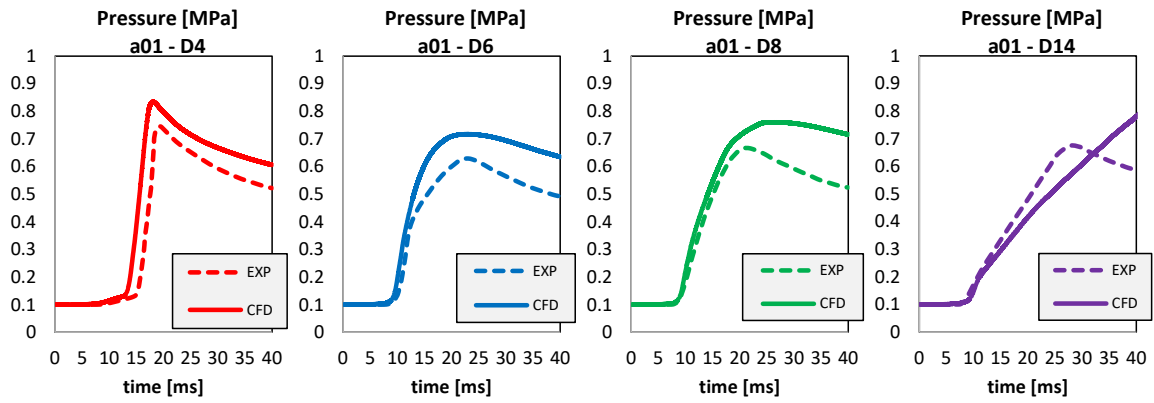


Figure 65 Experimental and numerical pressure traces for $a=10\%$, measured at main chamber probe

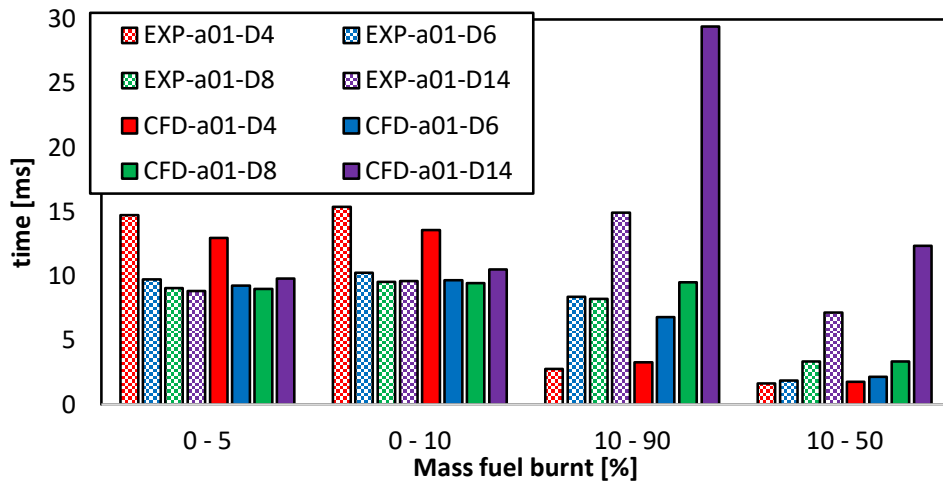


Figure 66 Ignition Delay and Combustion Durations for different orifice diameters, $a=10\%$

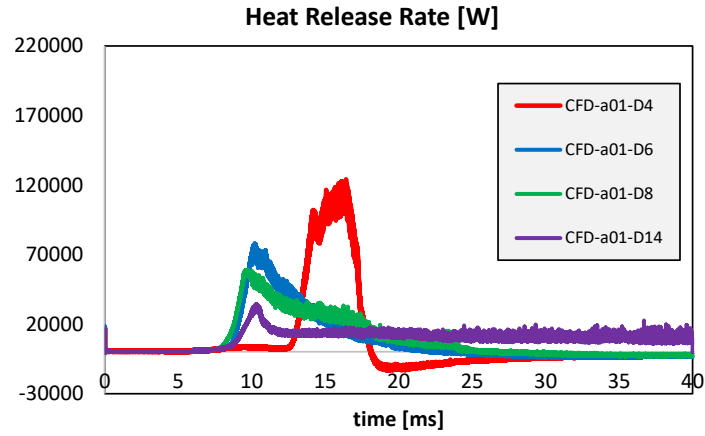


Figure 67 Heat Release Rate calculated based on average pressure trace, $a=10\%$

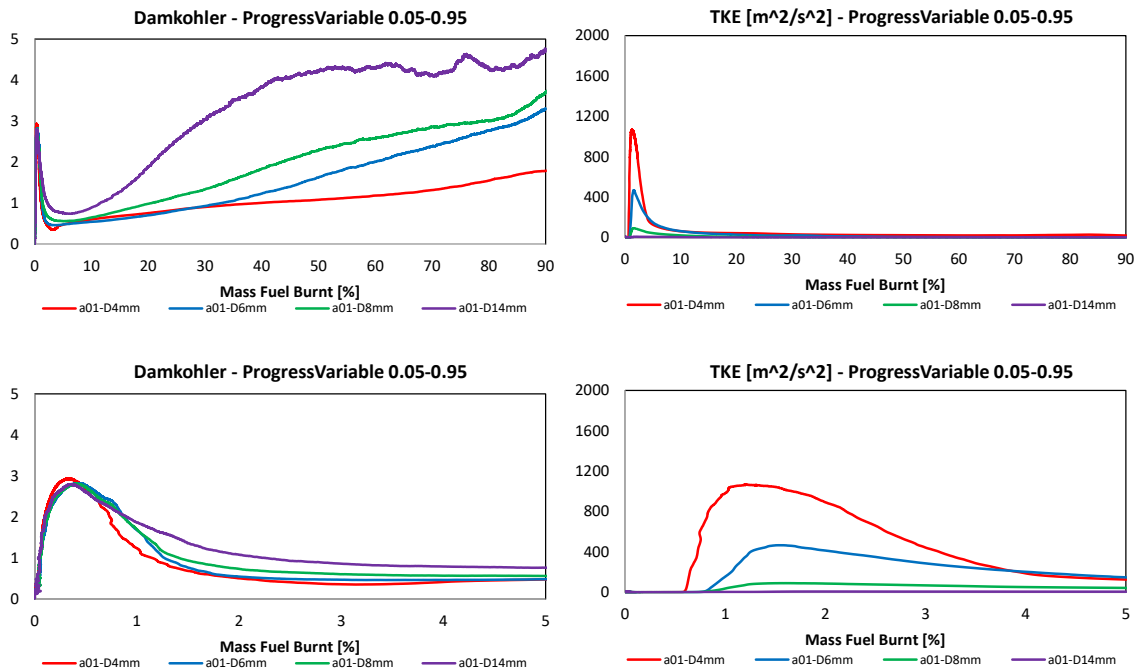


Figure 68 Average quantities seen by the flame, calculated based on the progress variable between 5% and 95%, $a=10\%$

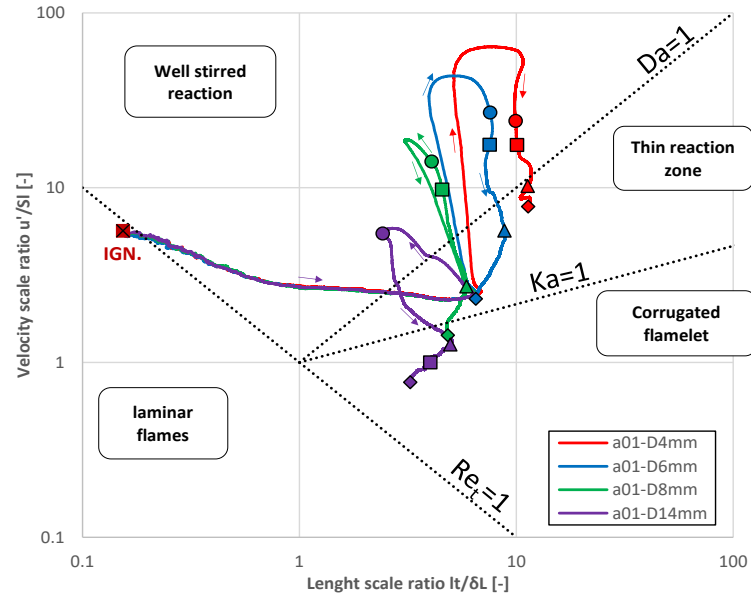


Figure 69 The combustion outcome of all test conditions involving $a=10\%$ are presented in the Borghi-Peters diagram. Circles correspond to MFB5, squares to MFB10, triangles to MFB50 and rhombus to MFB90. Most of the combustion inside the pre-chamber falls into the well-stirred reaction zone. Increasing the orifice area, the main chamber combustion regime tends to move into the thin reaction zone and even into the corrugated flamelet zone.

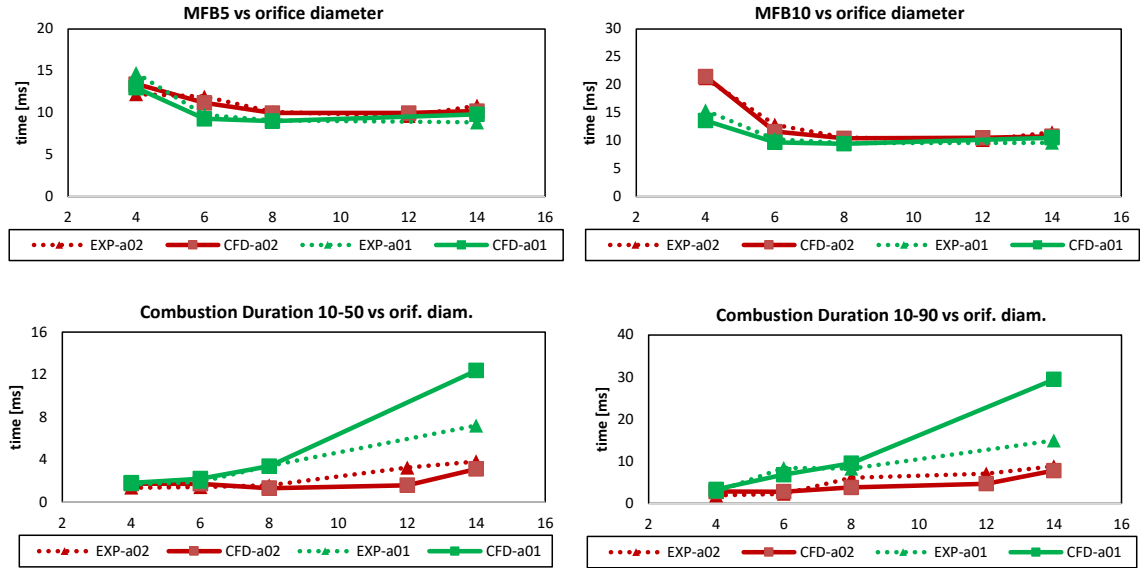


Figure 70 Ignition delay and Combustion durations varying the orifice diameter, $a=10\%$

Chapter 6. Conclusions and Further Work

A typical workflow to be followed as part of motor sport analyses is described in depth and two detailed numerical investigations are carried out. Initially, it is firstly described in detail a sloshing analysis of a lubricant tank of a high-performance sports car under racetrack maneuvers experienced during multiple lap-tracks. Therefore, it is described a detailed numerical investigation on the flame combustion regime of premixed propane/air mixture in a quiescent pre-chamber under ambient conditions. Both analyses are compared and validated against experimental measurements.

The sloshing application concerning the oil lubricant tank is initially validated in a simplified rectangular tank with internal baffles and under pitch oscillations, showing the capability of the model to handle phase interactions. The experimental data consisted of pressure traces measured in different positions within the tank and provided by literature. Numerical outcomes of the proposed EMP model are compared to those from a commonly adopted VoF approach to emphasize its superior capability to handle phase mixing and separation, as well its lower sensitivity to time step and grid size. Secondly, a static simulation of a current production lubricating oil tank for high-performance sports car is performed, where the effectiveness of the model in representing phase interactions is validated against test-rig measurements of foam formation at the suction side of the feed pump which have been provided (under confidentiality agreement) by a collaborating entity. Once again, EMP model shows trend-wise agreement with experimental data, i.e. the model is able to correctly capture the nature of the phenomenon, predicting an increase in foam when the total oil quantity is reduced. Thirdly, an analysis of a current production lubricating oil tank for high-performance sports car is carried out under racetrack accelerations. Two full laps are taken into account in order to reduce the dependence on the initial position of the free surface assumed horizontal at the beginning of the first track-lap.

The experimental data used in this application for model validation consisted of the experimental oil pump delivery pressure signal throughout the lap and have been provided (under confidentiality agreement) by a collaborating entity. This signal is compared to the numerical air percentage which reaches the feed pump suction side.

Again, the model effectiveness in representing both phase interactions and sloshing dynamics allows the CFD analysis to agree properly with the experimental data. The described applications, examined with an increase level of complexity, demonstrate the superiority of the EMP approach with respect to more widespread approaches such as VoF. In addition, EMP simulations are distinguished by fewer constraints in terms of grid density and Courant numbers, thus returning reliable and robust results with much lower computational costs and times. Such consideration is particularly relevant when dealing with computationally-expensive tasks such as transient analysis of long maneuvers. Finally, the methodology shows high capability to effectively handle phase mixing and subsequent separation both quantitatively and qualitatively with massive computational-time savings, far exceeding the constraints of the VoF approach and further extending the field of application of CFD analysis in multi-phase problems.

The numerical investigation on the flame combustion regime typically found in Turbulent Jet Ignition technology is performed. A premixed air/propane mixture is investigated in a quiescent divided chamber bomb under ambient conditions. The experimental data used in this analysis for model development and validation have been provided by literature and consisted of several pressure traces for different nozzle diameters and pre-chamber volumes. The availability of data and the absence of flow field and mixing uncertainties within the chambers make this application suitable for a robust investigation. Because of the low temperatures and the quiescent initial state, detailed chemical kinetics is integrated in the model setup and a kinetics-driven combustion modelling framework is used. A common CFD setup is shared by all the cases both in terms of ignition and combustion, i.e. avoiding case-by-case tuning of the modelling framework, thus developing a versatile methodology as robust as those needed in the design phase of modern pre-chamber systems. A remarkable agreement is found with the experimental jet outflow and main combustion event, both varying the chamber volume ratio and the nozzle diameter. First, a preliminary investigation is carried out with the highest PC/MC volume ratio and the smallest orifice diameter, resulting in a long choking phase within the orifice, which in turn causes the combustion in the MC to be more explosive, due to the simultaneous effect of fully dispersed active species and hot-jet-induced turbulence. Despite the complexity of the phenomena concerning this case, both jet outflow and combustion duration show good agreement with experimental counterparts. Consequently, high robustness of the model setup in terms of combustion duration and ignition delay time is

found increasing the orifice diameter while maintaining the same pre-chamber. Similarly, when the small pre-chamber and the narrower orifices of 4, 6 and 8mm are analyzed, the combustion duration shows good agreement with the experimental counterparts. However, the combustion duration tends to be underestimated when the nozzle diameter is increased up to 14 mm. This may be explained by the flame regime which affects such case: due to the much lower turbulence resulting from the decrease of the pre-chamber volume and the increase of the nozzle diameter, the Damköhler number concerning the flame tends to move the flame regime outside the well-stirred reaction zone. Particularly, when the largest nozzles are considered, the main chamber flame regime mostly falls into the corrugated flamelet zone, bringing out the limitations of a purely kinetics-driven modelling framework.

The conducted investigation aimed at understanding the flame behavior and its regimes using a pre-chamber combustor, without the uncertainties with respect to turbulence and mixing which may affect real-engine cases. It is shown that the higher the turbulence encountered by the flame when passing through the nozzle, the higher the reliability of a kinetics-driven framework in predicting jet exit and combustion duration, both fundamental in the design phase of modern pre-chamber systems. Further studies should be performed to investigate the use of kinetics-driven approaches under actual engine-like conditions as well as the use of a more traditional flamelet combustion framework. However, in modern pre-chamber systems a key role is played by the design of the pre-chamber and the orifice(s), whom the whole combustion process considerably depends on and for which the proposed methodology shows high reliability. Further improvements in reliability of the numerical setup could be obtained with a hybrid kinetics/flamelet models, whose adoption seems worthwhile in pre-chamber combustion systems where a-priori assumptions on specific combustion regimes throughout the flame propagation from PC to MC are very likely to fail.

Chapter 7. Bibliography

- 1 O. Baker, *Simultaneous flow of oil and gas*, *Oil Gas J.* 53 (1954) 185–195.
- 2 Sandra C.K., De Schepper, Geraldine J. Heynderickx, Guy B. Marin, *CFD modeling of all gas-liquid and vapor-liquid flow regimes predicted by the Baker chart*, Ghent University <https://doi.org/10.1016/j.cej.2007.06.007>
- 3 Kong S.H.H., Pao, W., *Numerical Solution of Passive and Partial Wet Gas Separation in T-Junction*, Universiti Teknologi Petronas, Malaysia
- 4 G.P. Celata, *Flow boiling*, in: *Lecture Given at Course: Boiling Heat Transfer and Boiling Equipment*, CISM, Udine, Italy, 2005.
- 5 Bergles AE, Collier JG, Delhaye JM, Hewitt GF, Mayinger F. *Two-phase Flow and Heat Transfer in the Power and Process Industries*. Hemisphere Publ. Corp., Washington, 1981
- 6 Hetsroni G. *Handbook of Multiphase Systems*. Hemisphere Publ. Corp., Washington, 1982
- 7 Gosman AD. *Developments in industrial computational fluid dynamics*. *Chem. Eng. Res. Des.*, 76(A2): 153-161, 1998. <https://doi.org/10.1205/026387698524721>
- 8 Casey M, Lang E, Mack R, Schlegel R, Wehrli M. *Applications of computational fluid dynamics for process engineering at Sulzer*. *Speedup J.*, 12(1): 43-51, 1988
- 9 Kuipers JAM, van Swaaij WPM. *Computational fluid dynamics applied to chemical reaction engineering*. *Adv. Chem. Eng.*, 24: 227-328, 1998. https://doi.org/10.1007/978-3-642-60185-9_45
- 10 Hjertager BH. *Computational fluid dynamics (CFD) analysis of multiphase chemical reactors*. *Trends Chem. Eng.*, 4: 45-92, 1998
- 11 Kusche H. *Computational Fluid Dynamics of Dispersed Two-Phase Flows at High Phase Fractions*. PhD Thesis, 2002. Imperial College of Science, Technology & Medicine
- 12 Jafari A, Hasani M, Hosseini M, Gharibshahi R. *Application of CFD technique to simulate enhanced oil recovery processes: current status and future opportunities*. *Pet. Sci.* 17, 434–456 (2020). <https://doi.org/10.1007/s12182-019-00363-7>

- 13 Raynal L, Augier F, Bazer-Bachi F, Haroun Y, Da Fonte CP. CFD applied to process development in the oil and gas industry—A review. *Oil Gas Sci Technol-Revue d'IFP Energies nouvelles*. 2016;71:42. <https://doi.org/10.2516/ogst/2015019>
- 14 Shelke, N., Bade, A., and Mukhopadhyay, S., “Fluid-Structure Interaction-Based Simulation Methods for Fluid Sloshing in Tanks,” *SAE Technical Paper 2019-01-5091*, 2019, doi:10.4271/2019-01-5091
- 15 Negrut, D., Mazhar, H., Serban, R., and Rakhsha, M., “On Simulating Sloshing in Vehicle Dynamics,” *SAE Technical Paper 2018-01-1110*, 2018, doi:10.4271/2018-01-1110
- 16 Brizzolara S., Serra F. “Accuracy of CFD codes in the prediction of planing surfaces hydrodynamic characteristics”. *2nd International Conference on Marine Research and Transportation*, 2007
- 17 Jafari A, Mousavi S. M., Kolari P, “Numerical investigation of blood flow. Part I: In microvessel bifurcations”. *Communications in Nonlinear Science and Numerical Simulation*. doi:10.1016/j.cnsns.2006.09.017
- 18 Krata, P., Jachowski, J., Towards a modification of a regulatory framework aiming at bunker oil spill prevention from ships – A design aspect of bunker tanks vents location guided by CFD simulations, *Reliability Engineering & System Safety*, 208, 2021. <https://doi.org/10.1016/j.ress.2020.107370>
- 19 Xue, M., Chen, Y., Zheng, J., Qian, L., Yuan, X., Fluid dynamics analysis of sloshing pressure distribution in storage vessels of different shapes, *Ocean Engineering*, 192, 2019. <https://doi.org/10.1016/j.oceaneng.2019.106582>
- 20 Farley FJM, Rainey RCT, Chaplin JR. “Rubber tubes in the sea”. *Phil. Trans. R. Soc. A*.370:381–402. 2012. <http://doi.org/10.1098/rsta.2011.0193>
- 21 Chaplin JR, Heller V, Farley FJM, Hearn GE, Rainey RCT. “Laboratory testing the anaconda”. *Phil. Trans. R. Soc. A*. 370:403–424. 2012. <http://doi.org/10.1098/rsta.2011.0256>
- 22 Zabihi M, Mazaheri S, Namin M, Mazyak A. “Irregular wave interaction with an offshore OWC wave energy converter”. *J. Ocean. Eng.* 222:108619. 2021. <https://doi.org/10.1016/j.oceaneng.2021.108619>
- 23 Wei Z, Zhai G, Wang Z, Wang W, Ji S. “An artificial intelligence segmentation method for recognizing the free surface in a sloshing tank”, *J. Ocean. Eng.* 220:108488. 2021. <https://doi.org/10.1016/j.oceaneng.2020.108488>

- 24 Lu, D., Yin, T., Liu, H., “Experimental and numerical investigation of sloshing behavior in annular region separated by several cylinders related to fast reactor design”. *Nucl. Eng. Des.* 339:235-243. 2018. <https://doi.org/10.1016/j.nucengdes.2018.09.009>
- 25 Cheng S, Li S, Li K, Zhang N, Zhang T. “A two-dimensional experimental investigation on the sloshing behavior in a water pool”. *Ann. Nucl. Energy* 114: 66-73. 2018. <https://doi.org/10.1016/j.anucene.2017.12.026>
- 26 Liu, Z., Feng, Y., Lei, G., Li, Y. “Thermal physical process in a liquid oxygen tank under different sloshing excitations”. *Int. Commun. Heat Mass.* 117:104771. 2020. <https://doi.org/10.1016/j.icheatmasstransfer.2020.104771>
- 27 Martinez-Carrascal J, González-Gutiérrez LM, “Experimental study of the liquid damping effects on a SDOF vertical sloshing tank”. *J. Fluid Struct.* 100:103172. 2021. <https://doi.org/10.1016/j.jfluidstructs.2020.103172>
- 28 Saltari F, Traini A, Gambioli F, Mastroddi F. “A linearized reduced-order model approach for sloshing to be used for aerospace design”. *Aerosp. Sci. Technol.* 108:106369. 2021. <https://doi.org/10.1016/j.ast.2020.106369>
- 29 Grossi E, Shabana A. “ANCF analysis of the crude oil sloshing in railroad vehicle systems”. *J. Sound Vib.* 433:493-516. 2018. <https://doi.org/10.1016/j.jsv.2018.06.035>
- 30 Fontanesi S, Cicalese G, De Pasquale G. “A methodology for the reduction of numerical diffusion in sloshing analyses through automated mesh adaptation”. *Energy Procedia.* 81:856-865. 2015. <https://doi.org/10.1016/j.egypro.2015.12.096>
- 31 Le Metayer O, Massoni J, Saurel R. “Modelling evaporation fronts with reactive Riemann solvers”. *J. Comput. Phys.*, 205:567-610. 2005. <https://doi.org/10.1016/j.jcp.2004.11.021>
- 32 Gavriluk S, Favrie N, Saurel R. “Modelling wave dynamics of compressible elastic materials”. *J. Comput. Phys.*, 227:2941-2969. 2008. <https://doi.org/10.1016/j.jcp.2007.11.030>
- 33 Iranmanesh A, Nikbakhti R. Numerical study on suppressing liquid sloshing of a rectangular tank using moving baffles linked to a spring system. *Ocean Eng.*, 229, 109002, 2021. <https://doi.org/10.1016/j.oceaneng.2021.109002>
- 34 Dervieux A, Thomasset F. “A finite element method for the simulation of Rayleigh-Taylor instability”, in: “Approximation methods for Navier–Stokes problems”, *Proceedings of the Symposium of the International Union of Theoretical and Applied Mechanics (IUTAM)*, pp. 145–158. Springer-Verlag, 1980. <https://doi.org/10.1007/BFb0086897>

- 35 Mulder W, Osher S, Sethian JA. "Computing interface motion: the compressible Rayleigh-Taylor and Kelvin-Helmholtz instabilities". *J. Comput. Phys.* 100(2):209-228. 1992. [https://doi.org/10.1016/0021-9991\(92\)90229-R](https://doi.org/10.1016/0021-9991(92)90229-R)
- 36 Sethian J. "Evolution, implementation and application of level set and fast marching methods for advancing fronts". *J. Comput. Phys.* 169(2):503-555. 2001. <https://doi.org/10.1006/jcph.2000.6657>
- 37 Fedkiw R, Aslam T, Merriman B, Osher S. "A non-oscillatory Eulerian approach to interfaces in multimaterial flows (the ghost fluid method)". *J. Comput. Phys.* 152(2):457-492. 1999. <https://doi.org/10.1006/jcph.1999.6236>
- 38 Abgrall R, Karni S. "Computations of compressible multifluids". *J. Comput. Phys.* 169(2):594-623. 2001. <https://doi.org/10.1006/jcph.2000.6685>
- 39 Liu TG, Khoo BC, Wang CW. "The ghost fluid method for compressible gas-water simulations". *J. Comput. Phys.* 204(1):193-211. 2005. <https://doi.org/10.1016/j.jcp.2004.10.012>
- 40 LeVeque RJ, Shyue K-M. "Two-dimensional front tracking based on high resolution wave propagation methods". *J. Comput. Phys.* 123(2):354-368. 1996. <https://doi.org/10.1006/jcph.1996.0029>
- 41 Abgrall R. "How to prevent pressure oscillations in multicomponent flow calculations: a quasi-conservative approach". *J. Comput. Phys.* 125(1): 150-160. 1996. <https://doi.org/10.1006/jcph.1996.0085>
- 42 Saurel R, Petitpas F, Berry RA. "Simple and efficient relaxation methods for interfaces separating compressible fluids, cavitating flows and shocks in multiphase mixtures". *J. Comput. Phys.* 228(5):1678-1712. 2009. <https://doi.org/10.1016/j.jcp.2008.11.002>
- 43 Favrie N, Gavriluk SL, Saurel R. "Solid-fluid diffuse interface model in cases of extreme deformations". *J. Comput. Phys.* 228(16):6037-6077. 2009. <https://doi.org/10.1016/j.jcp.2009.05.015>
- 44 Jacqmin D. "An energy approach to the continuum surface tension method". 34th Aerospace Sciences Meeting and Exhibit. AIAA. 1996. <https://doi.org/10.2514/6.1996-858>
- 45 Dumbser M. "A diffuse interface method for complex three-dimensional free surface flows". *Comput. Methods Appl. Mech. Eng.* 257:47-64. 2013. <https://doi.org/10.1016/j.cma.2013.01.006>
- 46 Tiwari A, Freund JB, Pantano C. "A diffuse interface model with immiscibility preservation". *J. Comput. Phys.* 252:290-309. 2013. <https://doi.org/10.1016/j.jcp.2013.06.021>

- 47 Green MD, Zhou Y, Dominguez JM, Gesteira MG, Peiró J. Smooth particle hydrodynamics simulations of long-duration violent three-dimensional sloshing in tanks. *Ocean Engineering*, 229, 108925, 2021. <https://doi.org/10.1016/j.oceaneng.2021.108925>
- 48 Longshaw SM, Rogers BD. Automotive fuel cell sloshing under temporally and spatially varying high acceleration using GPU-based Smoothed Particle Hydrodynamics (SPH). *Advances in Engineering Software*, 83:31-44, 2015 <https://doi.org/10.1016/j.advengsoft.2015.01.008>
- 49 Monaghan JJ. "Simulating free surface flows with SPH". *J. Comput. Phys.*, 110(2):399-406. 1994. <https://doi.org/10.1006/jcph.1994.1034>
- 50 Violeau D, Issa R. "Numerical modelling of complex turbulent free-surface flows with the SPH method: an overview". *Int. J. Numer. Methods Fluids*, 53(2):277-304. 2007. <https://doi.org/10.1002/fld.1292>
- 51 Oger G, Doring M, Alessandrini B, Ferrant P. "An improved SPH method: towards higher order convergence". *J. Comput. Phys.*, 225(2):1472-1492. 2007. <https://doi.org/10.1016/j.jcp.2007.01.039>
- 52 Ferrari A, Dumbser M, Toro EF, Armanini A. "A new 3D parallel SPH scheme for free surface flows". *Comput. Fluids* 38(6):1203-1217. 2009. <https://doi.org/10.1016/j.compfluid.2008.11.012>
- 53 Idelsohn SR, Oñate E, Del Pin F. "The particle finite element method: a powerful tool to solve incompressible flows with free-surfaces and breaking waves". *Int. J. Numer. Methods Eng.* 61(7):964-989. 2004. <https://doi.org/10.1002/nme.1096>
- 54 Del Pin F, Idelsohn SR, Oñate E, Aubry R. "The ALE/Lagrangian particle finite element method: a new approach to computation of free-surface flows and fluid-object interactions" *Comput. Fluids*, 36(1):27-38. 2007. <https://doi.org/10.1016/j.compfluid.2005.06.008>
- 55 Oñate E, Idelsohn SR, Celigueta MA, Rossi R. "Advances in the particle finite element method for the analysis of fluid-multibody interaction and bed erosion in free-surface flows". *Comput. Methods Appl. Mech. Eng.* 197:1777-1800. 2007. <https://doi.org/10.1016/j.cma.2007.06.005>
- 56 Larese A, Rossi R, Oñate E, Idelsohn SR. "Validation of the particle finite element method (PFEM) for simulation of the free-surface flows". *Eng. Comput.* 25(4): 385-425. 2008. <https://doi.org/10.1108/02644400810874976>
- 57 Wang T, Li W, Wu Z, Zhang J. Sloshing analysis and mechanical modeling of liquid cargo during lifting. *Procedia Computer Science*, 183:655-660, 2021. <https://doi.org/10.1016/j.procs.2021.02.111>

- 58 Tiwari A, Framke NH. "Modeling Fuel Tank Draining/Sloshing in a Typical Transiently Accelerating Vehicle using GT-SUITE for Reliable Tank Designing". SAE Technical Paper 2019-01-1262. 2019. <https://doi.org/doi:10.4271/2019-01-1262>
- 59 Tiwari A, Harrison J. "Simulation of Aircraft Fuel System with Complex 3D Tank Geometry Using a 1D Flow Solution". AIAA Propulsion and Energy 2019 Forum Session: Propellant Storage and Management II. AIAA Paper 2019-4360. 2019. <https://doi.org/10.2514/6.2019-4360>
- 60 Nichani V, Zhou Z, Gao H, Gendron S, Shlautman J, Aihara R et al. "A Direct 1D/3D (GT-SUITE/SimericsMP+) Coupled Computational Approach to Study the Impact of Engine Oil Pan Sloshing on Lubrication Pump Performance," SAE Technical Paper 2020-01-1112. 2020. <https://doi.org/10.4271/2020-01-1112>
- 61 Li Q, Dong F, Li K. "An interface-sharpening method with adaptive mesh refinement for volume-of-fluid simulations of two-phase compressible flows". Comput. Fluids. 210:104648. 2020. <https://doi.org/10.1016/j.compfluid.2020.104648>
- 62 Wen HL, Yu CH, Sheu T. "On the development of LS-assisted VOF method for incompressible interfacial flows". J. Comput. Phys. 406:109188. 2020. <https://doi.org/10.1016/j.jcp.2019.109188>
- 63 Rodrigues A, Wardle KE, Weller HG. Hybrid Multiphase CFD Solver for Coupled Dispersed/Segregated Flows in Liquid-Liquid Extraction, 2013. International Journal of Chemical Engineering <https://doi.org/10.1155/2013/128936>
- 64 Močilan, M., Žmindák, M., Pecháč, P., Weis, P., CFD Simulation of Hydraulic Tank, Procedia Engineering, 192, 2017. <https://doi.org/10.1016/j.proeng.2017.06.105>
- 65 Courant, R., Friedrichs, K. Lewy, H., Uber die partiellen Differenzengleichungen der mathematischen Physik, Mathematische Annalen, vol. 100, n.1, pp. 32-74, 1928.
- 66 Tomiyama, A., Kataoka, I., Zun, I., Sakaguchi, T. 1998, "Drag coefficients of single bubbles under normal and micro gravity conditions", JSME International Journal, Series B, 41(2), pp. 472 – 479.
- 67 Schiller, L., Naumann, A., 1933, "Uber die grundlegenden Berechnungen bei der Schwerkraftaufbereitung", VDI Zeits, 77(12), pp. 318-320.
- 68 Akyildiz H, Unal E. "Experimental investigation of pressure distribution on a rectangular tank due to the liquid sloshing". Ocean Eng. 32(11-12):1503-1516. 2005. <https://doi.org/10.1016/j.oceaneng.2004.11.006>

- 69 Akyildiz H, Unal E. "Sloshing in a three-dimensional rectangular tank: Numerical simulation and experimental validation". *Ocean Eng.* 33(16):2135-2149. 2006. <https://doi.org/10.1016/j.oceaneng.2005.11.001>
- 70 Jia Ying Yong, et al., "A review on the state-of-the-art technologies of electric vehicle, its impacts and prospects", *Renewable and Sustainable Energy Reviews*, <https://doi.org/10.1016/j.rser.2015.04.130>.
- 71 Mangeruga, V., et al., "Design of a Hybrid Power Unit for Formula SAE Application: Packaging Optimization and Thermomechanical Design of the Electric Motor Case", *SAE Technical Papers*, DOI: 10.4271/2019-24-0197
- 72 M.D. Hageman et al., "Determination of soot onset and background particulate levels on a spark-ignition engine", *Proc. Comb. Inst. Vol 35*, 2015, Pag 2949-2956, doi.org/10.1016/j.proci.2014.06.105.
- 73 E.S. Cho, H.C. Suk, Improvement of flame stability and NOx reduction in hydrogen-added ultra-lean premixed combustion, *J. Mech. Sci. Technol.* 23 (3) (2009) 650–658, <http://dx.doi.org/10.1007/s12206-008-1223-x>.
- 74 d'Adamo, A., et al., "A RANS knock model to predict the statistical occurrence of engine knock", *Applied Energy*, doi.org/10.1016/j.apenergy.2017.01.101.
- 75 D'Adamo, A., Breda, S., Fontanesi, S., and Cantore, G., "A RANS-Based CFD Model to Predict the Statistical Occurrence of Knock in Spark-Ignition Engines," *SAE Int. J. Engines* 9(1):618-630, 2016, doi:10.4271/2016-01-0581.
- 76 Breda, S., et al., "CFD Analysis of Combustion and Knock in an Optically Accessible GDI Engine", *SAE International Journal of Engines*, 2020. www.jstor.org/stable/26284845.
- 77 A.A. Quader, *Lean Combustion and the Misfire Limit in Spark Ignition Engines*, *SAE Technical Paper* 741055, 1974, <http://dx.doi.org/10.4271/741055>.
- 78 Germane, G., Wood, C., and Hess, C., "Lean Combustion in Spark-Ignited Internal Combustion Engines - A Review," *SAE Technical Paper* 831694, 1983, <https://doi.org/10.4271/831694>.
- 79 d'Adamo, A., et al., "Understanding the origin of cycle-to-cycle variation using large-eddy simulation: Similarities and differences between a homogeneous low-revving speed research engine and a production DI turbocharged engine", *SAE International Journal of Engines*, 2018. 12(1): p. 1-22. <https://doi.org/10.4271/03-12-01-0007>.

- 80 P. Chinnathambi, B. Thelen, D. Cook, E- Toulson, "Performance metrics for fueled and unfueled turbulent jet igniters in a rapid compression machine", *Applied Thermal Engineering*, <https://doi.org/10.1016/j.applthermaleng.2020.115893>.
- 81 S. Benekos, C. E. Frouzakis, G. K. Giannakopoulos, M Bolla, Y. M. Wright et al., "Prechamber ignition: An exploratory 2-D DNS study of the effects of initial temperature and main chamber composition", *Combustion and Flame*, doi:<https://doi.org/10.1016/j.combustflame.2020.01.014>
- 82 Distaso E, Amirante R, Cassone E, Catapano F, De Palma P, Sementa P et al. Experimental and numerical analysis of a pre-chamber turbulent jet ignition combustion system. *SAE Tech. Pap.* 2019-24-0018; 2019.
- 83 Attard, W., Fraser, N., Parsons, P., and Toulson, E., "A Turbulent Jet Ignition Pre-Chamber Combustion System for Large Fuel Economy Improvements in a Modern Vehicle Powertrain," *SAE Int. J. Engines* 3(2):20-37, 2010, doi:10.4271/2010-01-1457.
- 84 Attard, W., Kohn, J., Parsons, P., "Ignition Energy Development for a Spark Initiated Combustion System Capable of High Load, High Efficiency and Near Zero NOx Emissions," *SAE Journal Paper JSAE 20109088*, 2010.
- 85 Attard, W. and Blaxill, H., "A Gasoline Fueled Pre- Chamber Jet Ignition Combustion System at Unthrottled Conditions," *SAE Int. J. Engines* 5(2):315-329, 2012, doi:10.4271/2012-01-0386.
- 86 Attard, W. and Blaxill, H., "A Lean Burn Gasoline Fueled Pre-Chamber Jet Ignition Combustion System Achieving High Efficiency and Low NOx at Part Load," *SAE Technical Paper* 2012-01-1146, 2012, doi:10.4271/2012-01-1146.
- 87 Tsuboi S, Miyokawa S, Matsuda M, Yokomori T, Iida N. Influence of spark discharge characteristics on ignition and combustion process and the lean operation limit in a spark ignition engine. *Appl Energy* 2019;250:617–32. <https://doi.org/10.1016/j.apenergy.2019.05.036>.
- 88 KYRTATOS, P., BARDIS, K., BOLLA, M. et al. Transferability of insights from fundamental investigations into practical applications of prechamber combustion systems. *Ignition Syst. Gasol. Engines – 4th Int. Conf.* December 6-7, 2018, Berlin, IAV, 442-459.
- 89 Bunce, M., Blaxill, H., Kulatilaka, W., and Jiang, N., "The Effects of Turbulent Jet Characteristics on Engine Performance Using a Pre-Chamber Combustor," *SAE Technical Paper* 2014-01-1195, 2014, doi:10.4271/2014-01-1195.
- 90 Gentz, G., Thelen, B., Gholamisheeri, M., Litke, P., Brown, A., Hoke, J., Toulson, E., "A study of the influence of orifice diameter on a turbulent jet ignition system through combustion visualization

and performance characterization in a rapid compression machine”, *Applied Thermal Engineering* no. 81:399-411. Doi: <http://dx.doi.org/10.1016/j.applthermaleng.2015.02.026>.

91 Gerald Gentz, Bryce Thelen, Masumeh Gholamisheeri, Paul Litke, Adam Brown, John Hoke, Elisa Toulson, “A study of the influence of orifice diameter on a turbulent jet ignition system through combustion visualization and performance characterization in a rapid compression machine”, <http://dx.doi.org/10.1016/j.applthermaleng.2015.02.026>

92 Biswas, S., Tanvir, S., Wang, H., Qiao, Li, “On ignition mechanisms of premixed CH₄/air and H₂/air using a hot turbulent jet generated by pre-chamber combustion”, *Applied Thermal Engineering* no. 106:925-937. Doi: <http://dx.doi.org/10.1016/j.applthermaleng.2016.06.070>.

93 Muller, M, Freeman, C, Zhao, P, & Ge, H. "Numerical Simulation of Ignition Mechanism in the Main Chamber of Turbulent Jet Ignition System." Proceedings of the ASME 2018 Internal Combustion Engine Division Fall Technical Conference. Volume 2: Emissions Control Systems; Instrumentation, Controls, and Hybrids; Numerical Simulation; Engine Design and Mechanical Development. <https://doi.org/10.1115/ICEF2018-9587>

94 Bunce, M., Blaxill, H., Kulatilaka, W., and Jiang, N., “The Effects of Turbulent Jet Characteristics on Engine Performance Using a Pre-Chamber Combustor,” *SAE Technical Paper* 2014-01-1195, 2014, doi:10.4271/2014-01-1195.

95 Bunce, M. and Blaxill, H., "Methodology for Combustion Analysis of a Spark Ignition Engine Incorporating a Pre-Chamber Combustor," *SAE Technical Paper* 2014-01-2603, 2014, doi:10.4271/2014-01-2603.

96 Q. Malé, G. Staffelbach, O. Vermorel, et al. “Large Eddy Simulation of Pre-Chamber Ignition in an Internal Combustion Engine”, *Flow, Turbulence and Combustion*, Springer Verlag (Germany)

97 Masumeh Gholamisheeri, IndrekS. Wichman, Elisa Toulson, “A study of the turbulent jet flow field in a methane fueled turbulent jet ignition (TJI) system”, <http://dx.doi.org/10.1016/j.combustflame.2017.05.008>

98 Masumeh Gholamisheeri, BryceC Thelen, GeraldR Gentz, IndrekS Wichman, Elisa Toulson, “Rapid compression machine study of a premixed, variable inlet density and flow rate, confined turbulent jet”, <http://dx.doi.org/10.1016/j.combustflame.2016.05.001>

99 Bolla, M., Shapiro, E., Tiney, N., Kyrtatos, P. et al., “Numerical Simulations of Pre-Chamber Combustion in an Optically Accessible RCEM,” *SAE Technical Paper* 2019-01-0224, 2019, doi:10.4271/2019-01-0224.

- 100 Shapiro, E., Tiney, N., Kyrtatos, P., Kotzagianni, M. et al., "Experimental and Numerical Analysis of Pre-Chamber Combustion Systems for Lean Burn Gas Engines," SAE Technical Paper 2019-01-0260, 2019, doi:10.4271/2019-01-0260.
- 101 Guoqing Xu, Maria Kotzagianni, Panagiotis Kyrtatos, Yuri M. Wright, Konstantinos Boulouchos, "Experimental and numerical investigations of the unscavenged prechamber combustion in a rapid compression and expansion machine under engine-like conditions", <https://doi.org/10.1016/j.combustflame.2019.01.025>
- 102 Yamaguchi, S., Ohiwa, N., Hasegawa, T., "Ignition and Burning Process in a Divided Chamber Bomb", *Combustion and flame* 59: 177-178 (1985)
- 103 Berni F., Cicalese G., Fontanesi S., "A modified thermal wall function for the estimation of gas-to-wall heat fluxes in CFD in-cylinder simulations of high performance spark-ignition engines", <http://dx.doi.org/10.1016/j.applthermaleng.2017.01.055>.
- 104 Berni F., Fontanesi S., "A 3D-CFD methodology to investigate boundary layers and assess the applicability of wall functions in actual industrial problems: A focus on in-cylinder simulations", <https://doi.org/10.1016/j.applthermaleng.2020.115320>.
- 105 Berni, F., et al., "On the existence of universal wall functions in in-cylinder simulations using a low-Reynolds RANS turbulence model", DOI: 10.1063/1.5138752
- 106 Zhiwei Qin, Vitali V. Lissianski, Huixing Yang, William C. Gardiner, Scott G. Davis, Hai Wang, "Combustion chemistry of propane: a case study of detailed reaction mechanism optimization", *Proceedings of the Combustion Institute*, Volume 28, 2000/pp. 1663-1669
- 107 Blint RJ. The relationship of the laminar flame width to flame speed. *Combustion Science and Technology* 1986;49(1):79-92
- 108 Vancoillie, J., Demuyneck, J., Galle, J., Verhelst, S., van Oijen, J.A., "A laminar burning velocity and flame thickness correlation for ethanol-air mixtures valid at spark-ignition engine conditions", <http://dx.doi.org/10.1016/j.fuel.2012.05.022>.
- 109 D'Adamo, A., Del Pecchia, M., Breda, S., Berni, F. et al., "Chemistry-Based Laminar Flame Speed Correlations for a Wide Range of Engine Conditions for Iso-Octane, n-Heptane, Toluene and Gasoline Surrogate Fuels," SAE Technical Paper 2017-01-2190, 2017, doi:10.4271/2017-01-2190.
- 110 Del Pecchia, M., Breda, S., D'Adamo, A., Fontanesi, S. et al., "Development of Chemistry-Based Laminar Flame Speed Correlation for Part-Load SI Conditions and Validation in a GDI Research Engine," SAE Int. J. Engines 11(6):2018, <https://doi.org/10.4271/2018-01-0174>.

- 111 Del Pecchia, M., Pessina, V., Berni, F., d'Adamo, A., Fontanesi, S., "Gasoline-ethanol blend formulation to mimic laminar flame speed and auto-ignition quality in automotive engines", <https://doi.org/10.1016/j.fuel.2019.116741>
- 112 A. d'Adamo, C. Iacovano, S. Fontanesi, "Large-Eddy simulation of lean and ultra-lean combustion using advanced ignition modelling in a transparent combustion chamber engine", *Applied Energy*, Volume 280, 15 December 2020, 115949, <https://doi.org/10.1016/j.apenergy.2020.115949>
- 113 N. Peters, *Turbulent Combustion*, first edition., Cambridge University Press, 2000. ISBN-13: 978-0521660822.
- 114 N. Peters, *Laminar flamelet concepts in turbulent combustion*, in: *Twenty First International Symposium on Combustion*, The Combustion Institute, Pittsburgh, 1986, pp. 1231–1250, [http://dx.doi.org/10.1016/S0082-0784\(88\)80355-2](http://dx.doi.org/10.1016/S0082-0784(88)80355-2).
- 115 R.G. Abdel-Gayed, D. Bradley, F.K. Lung, *Combustion regimes and the straining of turbulent premixed flames*, *Combust. Flame* 76 (2) (1989) 213–218, [http://dx.doi.org/10.1016/0010-2180\(89\)90068-0](http://dx.doi.org/10.1016/0010-2180(89)90068-0).
- 116 T. Poinso, D. Veynante, S. Candel, *Diagrams of premixed turbulent combustion based on direct simulation*, in: *Twenty-Third International Symposium on Combustion*, The Combustion Institute, Pittsburgh, 1990, pp. 613–619, [http://dx.doi.org/10.1016/S0082-0784\(06\)80308-5](http://dx.doi.org/10.1016/S0082-0784(06)80308-5).
- 117 Clara Iacovano, Alessandro d'Adamo, Giuseppe Cantore, "Analysis and Simulation of Non-Flamelet Turbulent Combustion in a Research Optical Engine," *Energy Procedia*, Volume 148, August 2018, Pages 463-470, <https://doi.org/10.1016/j.egypro.2018.08.121>

Chapter 8. Abbreviations

CA	Crank Angle
CFL	Courant-Friedrichs-Lewy
D	Diameter
DMZ	Dynamic Multi-Zone
EMP	Eulerian MultiPhase
HRR	Heat Release Rate
ICE	Internal Combustion Engine
IMEP	Indicated Mean Effective Pressure
MFB	Mass Fuel Burnt
MC	Main-Chamber
PC	Pre-Chamber
RANS	Reynolds-Averaged Navier-Stokes
RCEM	Rapid Compression and Expansion Machine
SA	Spark Advance
SI	Spark Ignited
TJI	Turbulent Jet Ignition
TKE	Turbulent Kinetic Energy
VoF	Volume of Fluid
VF	Volume Fraction
WOT	Wide-Open Throttle