A one-dimensional spray model to teach diffusion flame fundamentals for Engineering students

José María García-Oliver, Antonio García, Joaquin De la Morena* and Javier Monsalve-Serrano

CMT - Motores Térmicos, Universitat Politècnica de València, Camino de Vera s/n, 46022 Valencia, Spain

Corresponding author (*):
Dr. Joaquin De la Morena (joadela@mot.upv.es)
Phone: +34 963876530
Fax: +34 963879659

Abstract

This work presents an interactive application for teaching spray dynamics in engineering degrees. The model is based on spray momentum conservation, and can be used to evaluate both fuel-air mixing characteristics in inert condition as well as diffusion flame performance once combustion takes place. During a dedicated computer-lab session, the students perform parametric studies regarding the influence of the nozzle outlet diameter, the combustion chamber density and the spray cone opening angle on the mixing process, characterized by the maximum stoichiometric length. Later on, the effect of the combustion reaction on the mixing field is evaluated. The results are analyzed taking as a reference the theoretical development made by Spalding and Schlichting for diffusion gas jets. The outcomes of several years using this technique are reported.

Keywords

Spray; Combustion; Practical Session; Engineering teaching; Educational tool

1. Introduction

Combustion is a topic of interest for engineering students due to its application of many different fields, such as domestic heating, electric power plants and propulsion systems. Since most of these combustion processes are based on hydrocarbon fuels, they are one of the most significant sources for carbon dioxide emissions, contributing to greenhouse effect and global warming [1]. Additionally, other harmful pollutants such as particulate matter or nitric oxides are generated as side products of the combustion processes [2–4]. Although part of the energy currently generated by combustion processes will be displaced for alternative energy sources (especially in power plants and ground transportation), it is well established that combustion-based applications will still have a very significant share of the total energy generation in the upcoming decades [5]. At the same time, future combustion systems will face more stringent emissions requirements in order to minimize their environmental effects [6]. Considering this scenario, Combustion fundamentals subject is included in the academic curriculum of both Aerospace and Mechanical Engineering degrees in Universitat Politècnica de València (Spain) [7].

Nevertheless, it has to be noted that combustion processes are often complex. On the one hand, chemical mechanisms for standard petroleum-based fuels include hundreds of species and thousands of reactions [8]. On the other hand, many different physical phenomena including atomization, evaporation, heat and mass transport and turbulence play a significant role combustion development [9,10]. Often, the teaching approach consist of analyzing academic cases, so that the students can understand the

main physical drivers for the different kinds of combustion processes through relatively simple equations and calculations.

Taking this into account, it is a good option to complement the knowledge gained from these academic studies with other experiences based on real engineering problems. In this sense, the best approach would be to perform a series of experiments on different combustion system hardware. However, this is not always possible due to economical and practical constraints: such facilities are usually expensive to both build and operate, and the experimental tools required for a deep analysis of combustion (as, for instance, optical diagnostics [11–13]) are too sophisticated and time consuming to be developed on a teaching environment.

The other alternative consists of using computer-aided learning tools. Properly designed, these tools facilitate the students to obtain close to real-world experiences with much lower consumption of time and resources. In fact, a similar approach is used by the industry in the combustion system pre-design phase in order to save time and costs. From teaching perspective, it is known that these tools can help the students not only to learn faster but to retain this knowledge for longer time [14]. Perumal and Ganesan [15] concluded that modeling-based learning is useful to comprehend complex fluid-dynamic processes. Regueiro et al. [16] showed that the students appreciated computer-aided learning due to its "learning-by-doing" capabilities. Gutierrez-Romero et al. [17] stated that such experiences are perceived by the students as "closer to industry" compared to traditional work in the classroom. Burke et al. [18] also provided evidence that modeling tools can help the students to gain more deep understanding of

complex phenomena and interactions (in their case, engine calibration), with particular advantages for foreign students.

In the case of the current study, a computer-aided learning strategy is applied for teaching diffusion flame fundamentals during practical sessions in a computer lab. Diffusion flames are used in many industrial and propulsion applications. While they have disadvantages in terms of emissions (mostly particulate matter) compared to fully premixed flames, their higher stability and controllability make them suitable for cases where wide load control is needed. In the methodology proposed, the simple analytical solution provided by Spalding [19] and Schilichting [20] for laminar isodense gas jets is complemented with a one-dimensional spray model based on momentum conservation equation [21,22]. The students use this model to perform different parametric studies to comprehend the effects of nozzle diameter, combustion chamber density and spray cone opening angle on fuel-air mixing characteristics. Finally, they analyze the differences in the mixing field for non-reacting (inert) and reacting conditions.

As far as the paper structure is concerned, the paper is divided in 5 section. Section 2 provides the details of the developed 1D spray model tool used, including the main physical equations as well as the graphical interface available for the students. Section 3 describes the work that the students perform, detailing the methodology used for the practical session and its objectives as well as the parametric studies proposed. Section 4 summarizes the main results obtained by the students giving examples from their reports, complemented with a statistical review of their performance and satisfaction surveys regarding the methodology used. Finally, the main conclusions from the research are summarized in Section 5.

2. 1D mixing model description

This section describes the bases of the combustion tool used in the practical sessions, as well as the methodological approach followed.

2.1. Governing equations and assumptions

In this section, an overview of the model will be given. A more detailed description of the model can be found in [21,22].

The spray is assumed to be injected into a quiescent chamber, which is large enough so that the fuel injection does not modify air conditions far away from the nozzle. A uniform velocity profile is assumed at the nozzle exit. The fuel stream exchanges momentum with the ambient air and sets it in motion, inducing air entrainment into the spray region, which enlarges with the axial distance. In the model, it is assumed that the rate of this radial enlargement of the spray is constant, and defined by a certain spray cone angle or spreading angle θ. This parameter is generally a function of the internal nozzle flow characteristics and the chamber air density [23], being hard to estimate in a simple one-dimensional model, so it has been defined as an input parameter for the code. Together with the nozzle diameter d_0, the spray angle defines the virtual origin of the spray. All these conditions are summarized in Figure 1.

The spray domain is discretized with a constant axial spacing Δx, defining a control volume with a truncated-cone shape where momentum conservation equation is solved. At every time step, the spray tip penetration s is defined as the farthest control volume from the virtual origin location where the inlet velocity is different from zero and outlet velocity is zero.
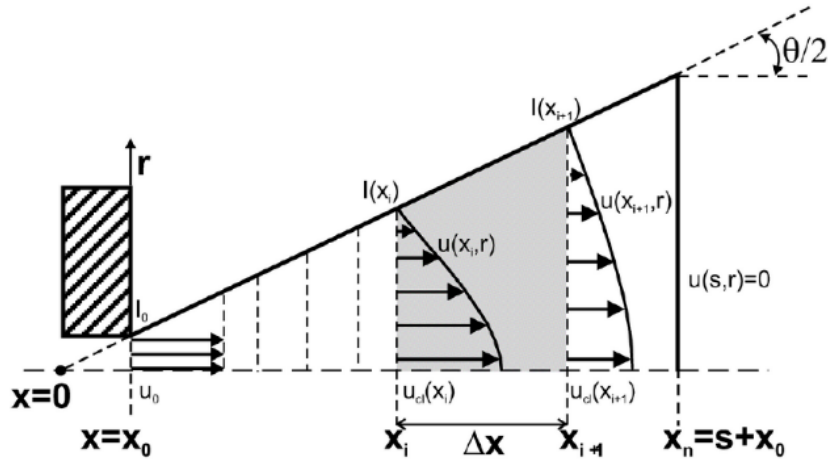
5

Figure 1: Model description and boundary conditions

The following hypotheses are made:

- All the flow conditions are symmetric to the spray axis. This implies that there are no convective streams in the air such as swirl or tumble motion, since the chamber is assumed to be quiescent.

- A fully developed turbulent flow is assumed, which means that the main solved variables (velocity and mixture fraction) have self-similar radial distribution profiles. In the present approach, a radial Gaussian or exponential profile is assumed, so that:

$$\frac{u(x,r)}{u_{cl}(x)} = \left[\frac{f(x,r)}{f_{cl}(x)}\right]^{1/Sc} = exp\left[-k\left(\frac{r}{x}\right)^2\right] \quad (1)$$

where $u_{cl}(x)$ and $f_{cl}(x)$ are the values on the spray axis or centerline for the axial velocity and the mixture fraction, respectively, k is a shape constant, and Sc is the turbulent Schmidt number.

- The spray cone angle is defined as the location where the axial velocity reaches 1% of the value at the spray axis. This imposes the condition for the calculation of the shape constant *k* as:

$$k = \frac{ln(1/\zeta)}{tan^2(\theta/2)} \quad (2)$$

- The turbulent Lewis number is assumed to be equal to 1. Consequently, the local enthalpy can be directly calculated as a direct function of the mixture fraction and the characteristic enthalpies of fuel and air streams:

$$h(x,r,t) = h_{a,\infty} + f(x,r,t)\big(h_{f,0} - h_{a,\infty}\big) \quad (3)$$

where *f(x, r, t)* is the local mixture fraction value and $h_{f,0}$ and $h_{a,\infty}$ are the enthalpy of pure fuel (at the nozzle outlet conditions) and pure air (far away from the nozzle), respectively. Thus, the mixture enthalpy is calculated as a weighted average of the fuel and air streams, and energy conservation equation does not need to be solved.

- The pressure is assumed to be constant all over the spray, which is consistent with the assumption that the chamber volume is assumed to be significantly larger than the spray volume.
- A locally homogeneous flow is assumed, i.e., local equilibrium exists both in thermal and velocity conditions. This allows for the consideration of the spray as a single-fluid jet.

The following expressions show the mass and momentum conservation equations solved on each cell or control volume:

7

$$I(x_i, t) - I(x_{i+1}, t) = \frac{d}{dx} \int \rho u \, dV \quad (4)$$

$$M_f(x_i, t) - M_f(x_{i+1}, t) = \frac{d}{dx} \int \rho f \, dV \quad (5)$$

I and M_f are the axial momentum (related to local axial velocity, *u*) and the mixture fraction (related to local mixture fraction, *f*) fluxes, respectively. *i* would represent the inlet section of a certain control volume, while *i+1* would represent the outlet surface. Thus, the left hand terms accounts for the variation of each of these fluxes across the control volume at a certain time step. The right-hand-side terms represent the temporal variation of the integral over the whole cell, which quantifies the process of accumulation or de-accumulation of momentum and fuel within a cell. Introducing the radial profiles previously seen in equation [1], the conservation equations can be expressed only in terms of u_{cl} and f_{cl}, which can be solved starting from the nozzle exit, where the injection conditions are known ($u_{cl}=u_0$ and $f_{cl}=1$).

Nevertheless, in order to solve these equations it is necessary to define the value of the local density, which is included in the integral terms. This is achieved by means of the so-called state relationships. Such relationships define the local composition, temperature and density inside the spray as a function of the mixture fraction. For this purpose, ideal adiabatic-mixing processes of pure fuel [nozzle conditions, subscripted as (*f*, 0)] and pure air [subscripted as (*a*, ∞)] are assumed, similarly as it was previously stated for the local enthalpy (equation [3]). The calculation of the state relationships is independent of the solution of the flow conservation equations.

Figure 2 summarizes the model calculation structure. The green boxes show the main boundary conditions or input variables needed for the model, including ambient

thermodynamic properties, the spray cone angle and the mass flow rate and momentum flux at the nozzle exit. The red boxes show the different equations that are solved inside the model. The conservation equations are solved at each cell to obtain u_{cl} and f_{cl}, from which u and f can be obtained at any other location using the radial distribution profiles. Eventually, once f is obtained at any location, the local temperature, density, and composition can also be calculated from the state relationships.
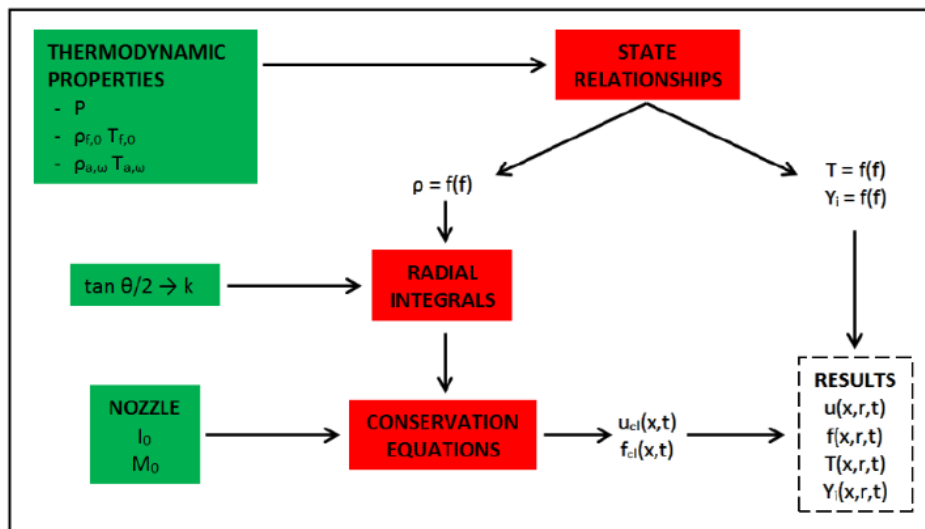

Figure 2: Overview of model inputs and outputs

2.2. Graphical interface

The previously described model is programmed into C++ code with a user-friendly graphical interface, meant to guide the students through the case setup process. This interface is divided in 4 main windows: "General", "Morphology", "Injection Rate" and "Mixing laws". The objective for this division is to guide the students on what aspects they are affecting when changing a specific parameter or software option.

Figure 3 shows the "General" window. Here, the students can select if the case study is done on steady-state or transient conditions. The selection by default is steady state,

which implies that the nozzle outlet mass flow and momentum flux are constant. Additionally, the user can select the path where the simulation results will be stored.
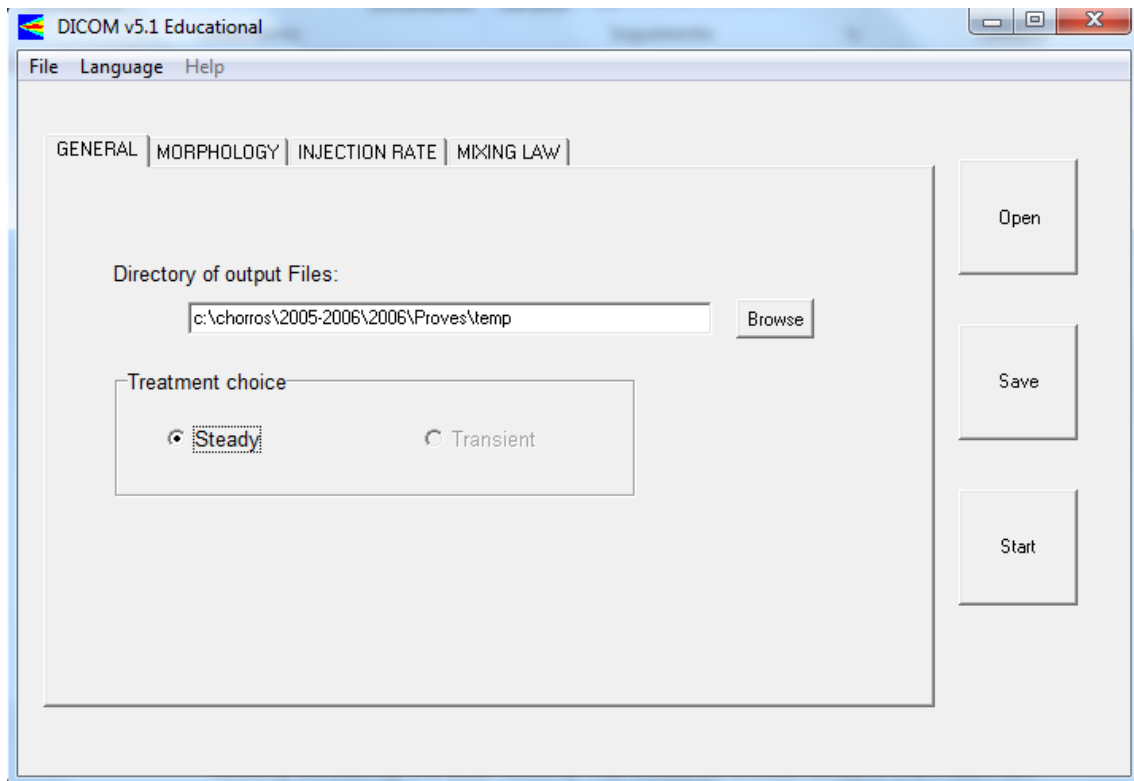

Figure 3: "General" window in the graphical interface

Figure 4 shows the "Morphology" window. Three main parameters can be varied:

- Velocity fraction for spray boundary: it represents the value of the local velocity divided by the axial velocity that would be encountered in the maximum spray radius, defined by the cone opening angle. The value by default is 0.01 (i.e. 1 percent of the axial velocity).

- Turbulent Schmidt number, represents the ratio between momentum and mass diffusivity in turbulent conditions. The value by default is 1.

- Spray cone opening angle. As previously described, it is assumed that the spray widens at a constant rate defined by this angle. This value depends mostly on

the nozzle geometrical characteristics and the density ratio between fuel and combustion chamber, and is typically determined through experimental correlations. In the case of the practical session, it will be one of the parameters varied by the students.

Apart from these three parameters, the user can select four different mathematical definitions for the radial velocity/mixture fraction profiles. As previously stated, in the current work Gaussian or exponential profile described by equation [1] is used, which is the option selected by default. All these parameters are related to the way in which the results of the one-dimensional calculations on the spray axis are extended to the radial direction.
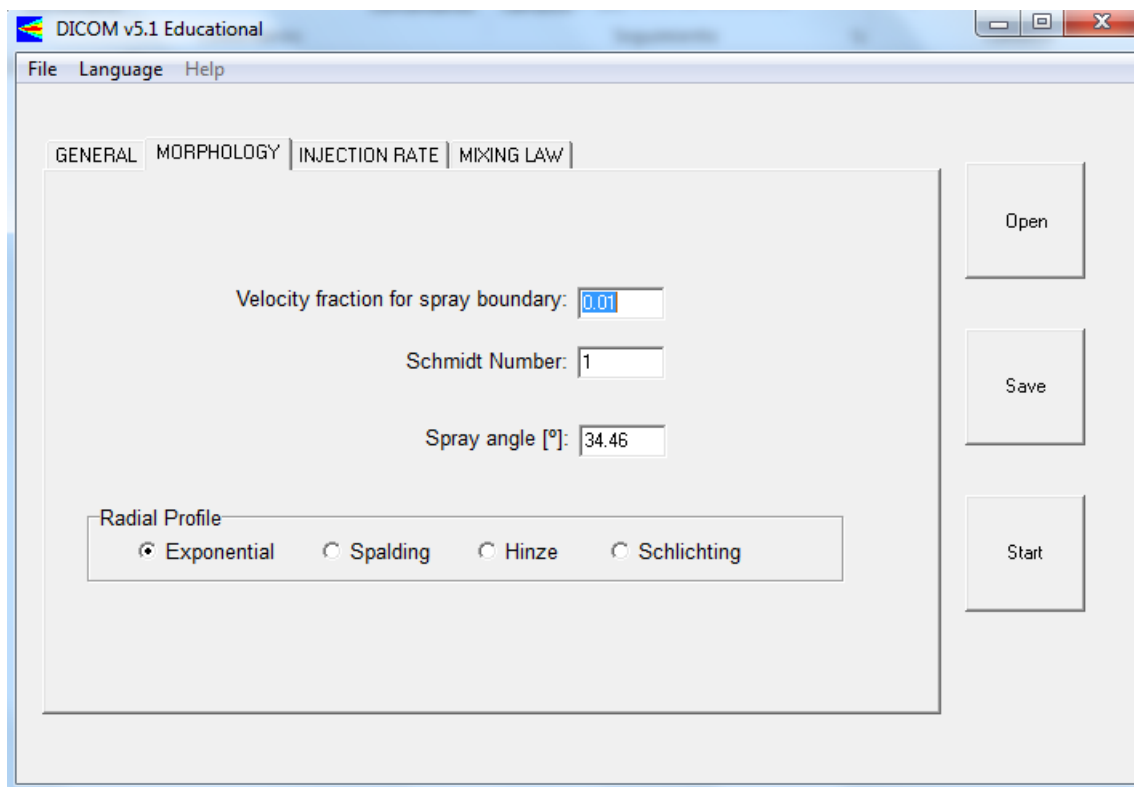

Figure 4: "Morphology" window in the graphical interface

In the "Injection rate" window (Figure 5) the user introduces the boundary conditions at the nozzle outlet. When the "direct input" option is selected, the user introduces directly the values of mass flow rate and momentum flux, together with the nozzle outlet diameter. This is the option that will be used for the practical session currently described. Additionally, there is the possibility to derive the momentum flux from a certain pressure difference across the nozzle and a velocity coefficient using the "derived input" option [24].
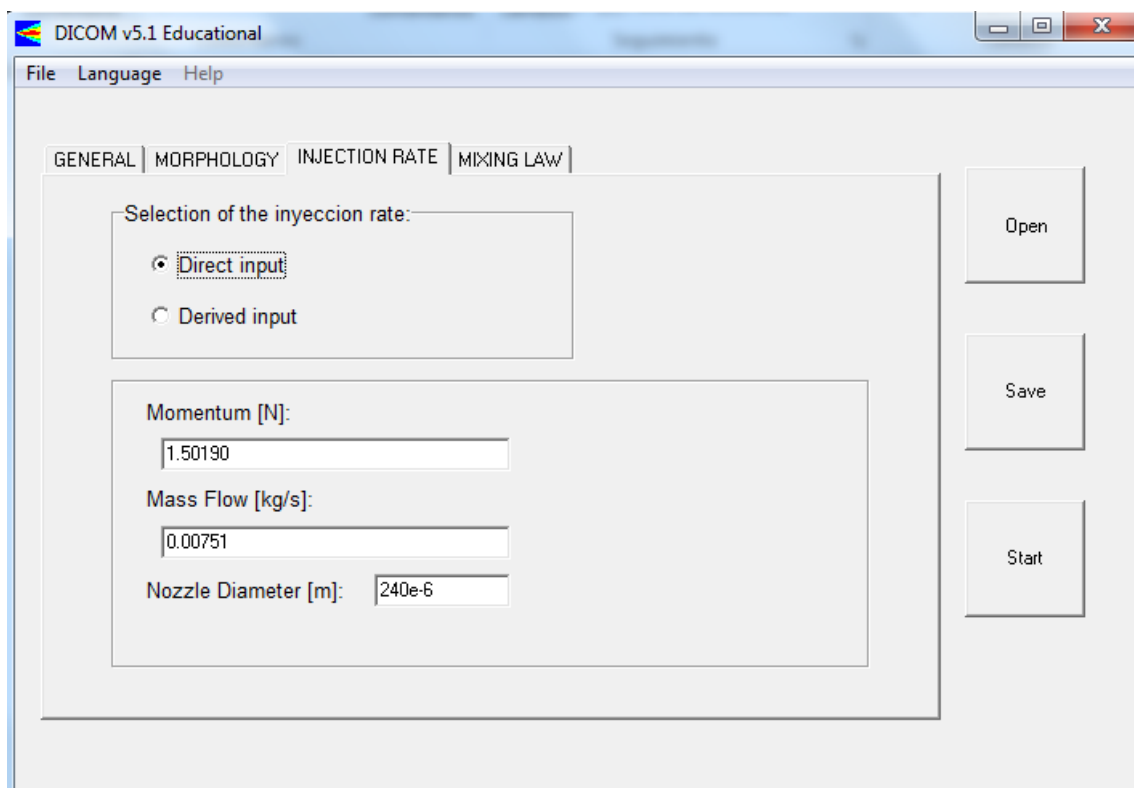

Figure 5: "Injection rate" window in the graphical interface

Finally, the main boundary conditions inside the combustion chamber are defined in the "Mixing law" window. Three main option can be selected:

- Isothermal (Figure 6). In this case, the simulation is performed in inert (non-reacting) conditions, and assuming that the fuel temperature is the same as the

gas temperature in the combustion chamber. In this case, only three physical parameters need to be defined: the combustion chamber density, the fuel density and the stoichiometric mass fraction, which depends on the fuel formulation used.
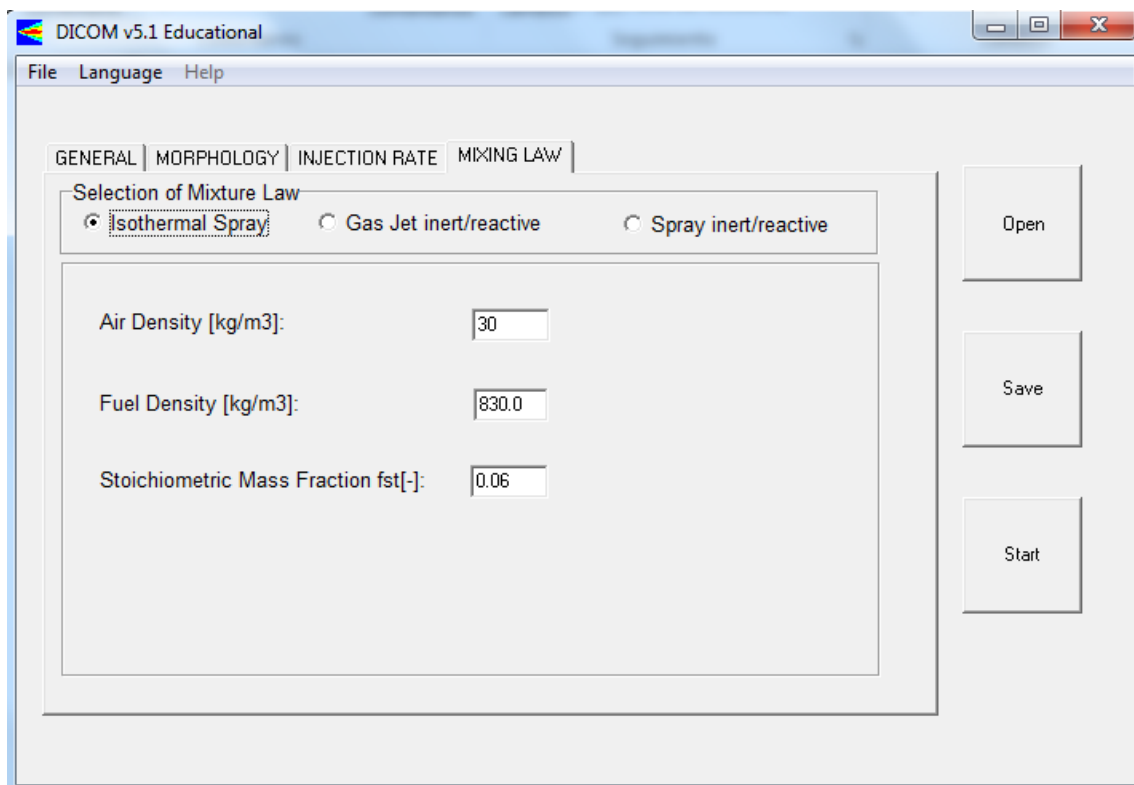
Figure 6: "Mixing law" window in the graphical interface for the isothermal calculation

- Inert/reacting gas jet (Figure 7). In this case, the temperature between fuel and combustion chamber can be different, but the fuel is considered to exit the nozzle in gas phase (so evaporation is neglected). In this case, apart from the combustion chamber density, it is necessary to introduce its pressure. The simulation can be done in either inert or reacting conditions through a binary

parameter called "Reactivity parameter" or *fLOL*. In order to compute reacting simulation, it is necessary to define the composition inside the combustion chamber gas. In this case, only nitrogen and oxygen are assumed as air constituents, but it is possible to manually introduce their mass fractions. Finally, the fuel temperature is also input to account for its enthalpy.

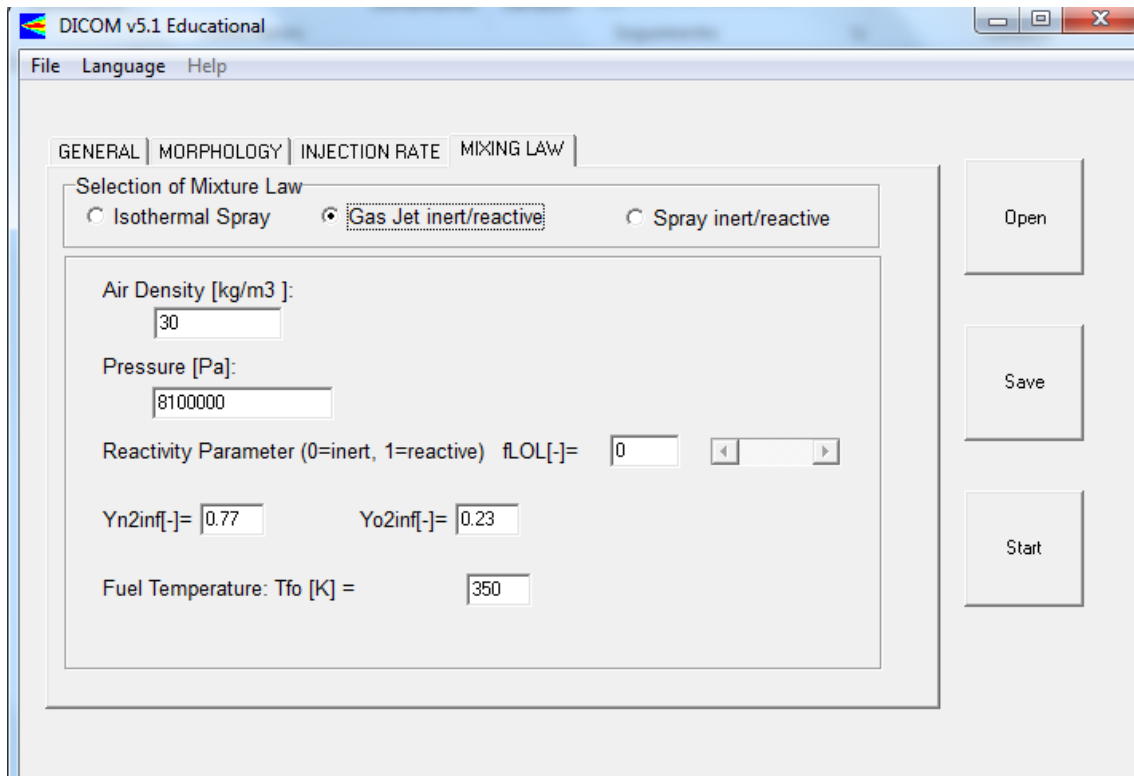Figure 7: "Mixing law" window in the graphical interface for the inert/reacting gas jet calculation

- Inert/reacting spray. This simulation is equal to the one previously described for the gas jet, but under the assumption that the fuel is injected in liquid phase, so the evaporation process is calculated.

14

3. Teaching methodology

In this section, the methodology used during the practical session is described.

3.1. Previous knowledge and teaching objectives

Before the software is introduced to the students, two standard classroom sessions with a total duration of four hours have been devoted to the description and analysis of gaseous diffusive flames. During these sessions, the theoretical solution provided by Spalding and Schilchting for describing the fuel-air mixing process in laminar and inert conditions is described. The maximum length of the stoichiometric surface is introduced as an estimator for the flame length. Once this theoretical model is fully described, the effects of turbulence on the mixing process are introduced and quantified thanks to the introduction of the turbulent viscosity. Finally, the similarities and differences between the stoichiometric length in inert conditions and the flame length are discussed.

However, when arriving to the end of these standard classroom sessions there are certain lacks in the students' technical knowledge. First, while most diffusive flames in realistic applications are performed with liquid fuel (due to its higher energy density), only gaseous fuel flames are described. In this sense, the software introduced in the current contribution represents a nice link since it is based on the same equations described for gaseous flames, but corrected to solve the dynamics of flames with liquid fuels. This allows to demonstrate that the contents described in the classroom sessions, which may seem oversimplified and may lead to a lack of motivation for the students, are useful to describe real-life conditions, improving their engagement.

Additionally, during the classroom session only one practical problem has been solved, but the sensitivities of the different operating parameters to the fuel-air mixing process

and flame length are not provided since making the calculations by hand is time consuming. Therefore, the usage of the previously described software is aimed at providing the students' with these sensitivities.

In summary, the main objectives covered during the computer session focused in the one-dimensional model are:

- Complement the theoretical examples seen in the standard classroom sessions for gaseous jets with more realistic applications with liquid fuels.

- Provide order of magnitude and sensitivities to the main operating parameters for liquid sprays.

- Improve the students' engagement and motivation.

3.2. Computer session organization

The practical sessions are performed in reduced groups, ranging from 15 to 20 students. The total duration of the session is 2.5 hours. Each practical session is divided in three parts:

a) Theoretical introduction (30 minutes). First, a brief explanation about the 1-dimensional model is provided by the teacher. This explanation includes the main assumptions used to simplify the mass and momentum conservation equations, with particular emphasis of the self-similarity assumption of the Gaussian velocity and mixture fraction radial profiles (as explained in section 2.1). Additionally, an example of validation of the code against experimental data coming from optical diagnostics applied to diesel inert and reacting sprays is

provided, aimed at proving the students that the tool accurately models the spray structure in realistic engine conditions.

b) Tool tutorial (15 minutes). Again, this part is done by the teacher. The objective is to describe the different configuration windows available in the software, highlighting the parameters that will be explored by the students in the rest of the session. Additionally, the structure output data files is detailed.

c) Parametric studies and analysis (105 minutes). In groups of 2-3 students, a guided report including different parametric variations is performed. The data analysis is guided in this report through a series of questions aimed at improving the understanding of the student about the physical phenomena behind the results obtained.

3.3. Report structure and parametric studies justification

The report filled out by the students is divided on three different subsections:

a) <u>Reference case analysis</u>. The students are asked to perform a baseline simulation with typical engine-like conditions:

Table 1: baseline case conditions

Condition	Spray cone angle [º]	d_0 [mm]	ΔP [bar]	ρ_f [kg/m^3]	ρ_a [kg/m^3]	f_{st} [-]
Reference	20	0.150	800	830	30	1/15

This case is computed using the isothermal calculation mode. It can be noticed immediately that the nozzle outlet mass flow rate and spray momentum are not provided. Instead, the students are expected to estimate these conditions from the

pressure drop across the nozzle, the outlet diameter and the fuel density. The equations to do so can be easily derived from Bernoulli's equation:

$$I_o = \rho_f A_o u_o^2 = \frac{\pi}{4} d_o^2 2\Delta P \quad (6)$$

$$M_o = \rho_f A_o u_o = \frac{\pi}{4} d_o^2 \sqrt{2\Delta P \rho_f} \quad (7)$$

Once the simulation is performed, the students are asked to plot the following mixing-related quantities against the axial position:

- Velocity at the centerline divided by outlet velocity (u_{cl}/u_o).

- Mixture fraction at the centerline (f_{cl}).

- Total spray radius and stoichiometric surface radius.

- Total mass flow (M).

From these results, they are asked to check the following aspects and explain why they appear:

- The total radius always increases with the axial position while the stoichiometric radius becomes zero at a certain distance.

- The velocity ratio (u_{cl}/u_o) and the mixture fraction are equal.

- The total mass flow increases linearly with the axial position.

The main objective for this reference case is to show the students that the same trends described during the classroom sessions are achieved in the software results, confirming that the analogy between gaseous jet and liquid sprays is valid.

b) <u>Parametric studies.</u> Once the baseline case is fully analyzed, the students perform the following parametric variations (all in isothermal calculation mode),

having to analyze if these parameters have a positive or negative effect on the fuel-air mixing process. The particular cases to be run are summarized in Table 2.

Table 2: parametric study

Condition	Spray cone angle [º]	d_0 [mm]	ΔP [bar]	ρ_f [kg/m³]	ρ_a [kg/m³]	f_{st} [-]
2	20	**0.125**	800	830	30	1/15
3	20	**0.175**	800	830	30	1/15
6	20	0.150	800	830	**20**	1/15
7	20	0.150	800	830	**40**	1/15
8	**15**	0.150	800	830	30	1/15
9	**25**	0.150	800	830	30	1/15

As it can be seen, the variation with respect to the baseline case is highlighted.

Looking at Table 2, for the inert sprays the following parameters are covered in the parametric studies:

- Nozzle outlet diameter. The objective is to show that the air entrainment for turbulent jets is inversely proportional to the diameter, so stoichiometric length is enlarged. Additionally, since the injection velocity is maintained constant, the total mass flow introduced also increases. This is equivalent to what happens in diesel fuel injectors.

- Air density. In this case, the change of air density is inducing a similar effect as the previous parameter, since it represents a change in the equivalent diameter, which is the parameter driving the gas jet behavior. The combination of this study together with the previous one should help the students' to arrive to this conclusion.

19

- Spray cone angle. This parameter introduces one of the main differences between gas and liquid sprays that need to be taken into account for real-life applications. While in turbulent gaseous jets the spray angle is independent from the operating conditions, in a liquid spray this angle is a consequence of complex physical phenomena involved, such as primary and secondary atomization and fuel evaporation, which do depend on the operating conditions. Therefore, the spray angle is a critical parameter for understanding a real spray behavior.

c) <u>Inert vs. reacting comparison.</u> Finally, the independent effect of reacting flow is studied by performing two equal simulations in the inert/reactive gas jet mode. The boundary conditions for these simulations are the following:

Table 3: inert vs. reacting simulation

Condition	Spray cone angle [º]	d_0 [mm]	ΔP [bar]	ρ_f [kg/m³]	ρ_a [kg/m³]	P[bar]	fLOL[-]
10	20º	0.150	800	830	30	50	0 (inert)
11	20º	0.150	800	830	30	50	1 (reactive)

Again, the following questions are proposed:

- What is the effect of combustion on the velocity distribution and the air entrainment?

- Is combustion beneficial or detrimental for the fuel-air mixing process?

In this case, the main objective is for the students to confirm the expansion induced by the combustion process due to the local density change produced by the higher temperatures achieved.

4. Results

4.1. Sample report answers from students

In this section, some excerpts from students reports are reported in order to overview the detail of the analysis achieved thanks to the methodology proposed.

a) <u>Reference case analysis.</u> The following charts and comments are obtained from one of the students reports:
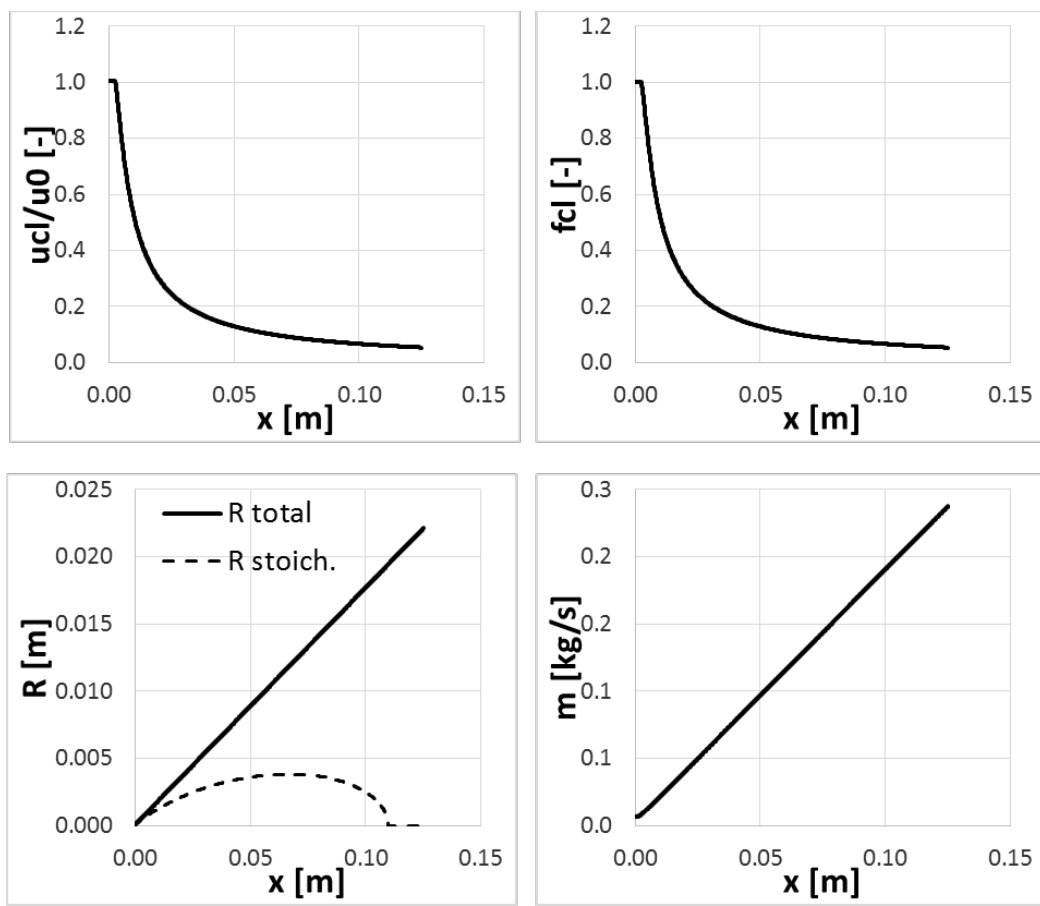

Figure 8: Extract from students report regarding the reference case analysis

"The axial evolution of the mixture fraction is equal to the one of the velocity ratio between centerline and outlet section. This is due to the fact that Schmidt number is

imposed to 1. As expected, both decrease with the axial distance as air is entrained into the jet".

"In a first step, we observe that both velocity and mixture fraction in the centerline are constant. This is partially related to the fact that the origin of the reference system is not at the nozzle outlet, but in an inside point where the spray radius would become zero. Additionally, at the centerline the first steps outside the nozzle are not affected yet by air entrainment."

"It can be seen that the total spray radius always increases with the axial distance because the spray cone angle is assumed to be constant on the simulations. On the contrary, the stoichiometric surface radius reaches a maximum and then decreases until arriving to zero. After this point, the high amount of entrained air results in the fact that equivalence ratio is always lower than one. The linear increase of the total mass flow is a consequence of the entrained air and the constant spray angle."

b) <u>Parametric studies:</u>
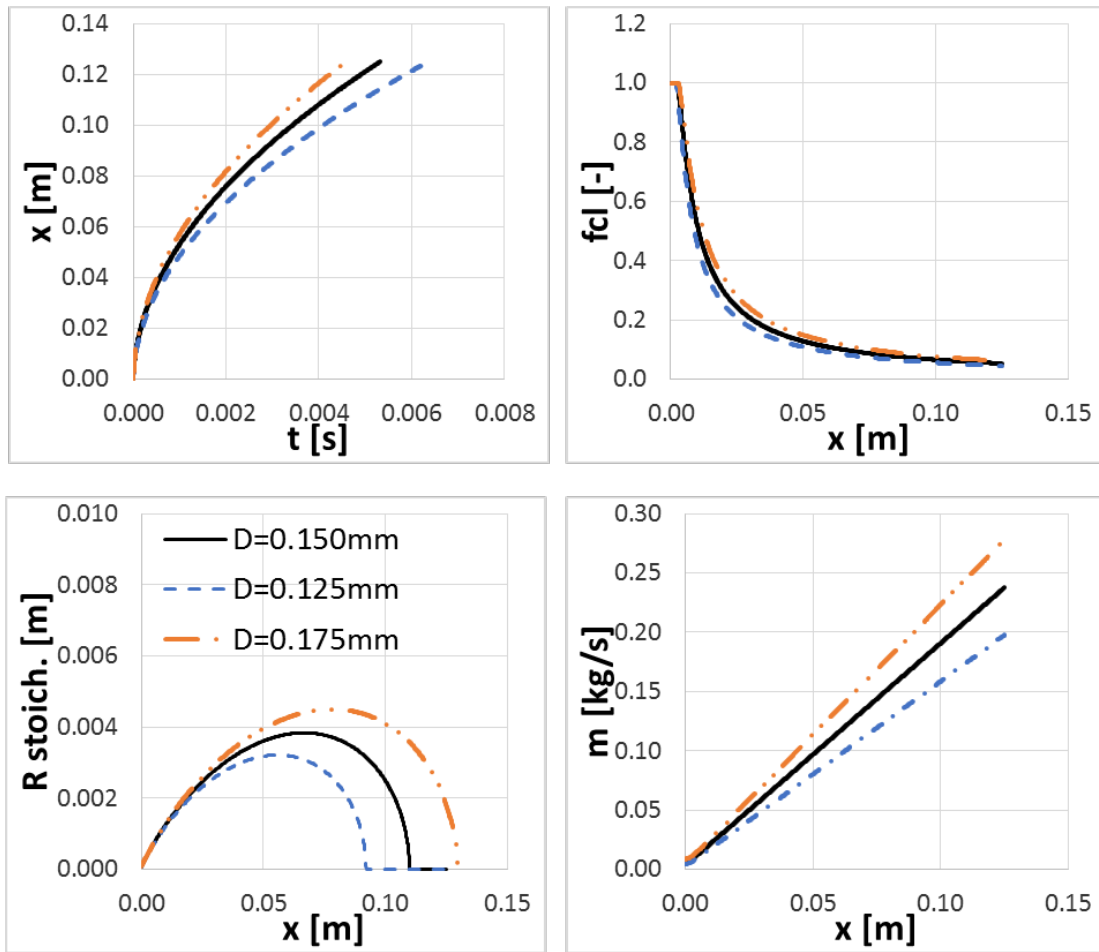
- Diameter effect (cases 1-2-3)

Figure 9: Extract from students report regarding the outlet diameter influence

"The trends observed in the charts can be explain making an analogy with the Spalding/Schlichting solution, even if it is derived for laminar isodense sprays (which is not the case of study). In this solution, the radial position for a given mixture fraction is defined as:

$$r = \frac{16}{\sqrt{3}} \frac{x}{R_{eo}} \sqrt{\sqrt{\frac{3}{32} \frac{R_{eo} d_o}{x} \frac{1}{f}} - 1} \qquad (8)$$

It is possible to look for the maximum radius by calculating the derivative of this radius in the axial direction and making it equal to zero:

$$x_{r\,máx} = \frac{27 d_o R_{eo}}{512 f} \tag{9}$$

$$r_{máx} = \frac{9 d_o}{32 f} \tag{10}$$

So it can be seen that increasing the outlet diameter results in an increase of the radius for any given mixture fraction, for instance the stoichiometric one, as seen in the previous graphs. Therefore, increasing the outlet diameter results in a lower mixing rate"
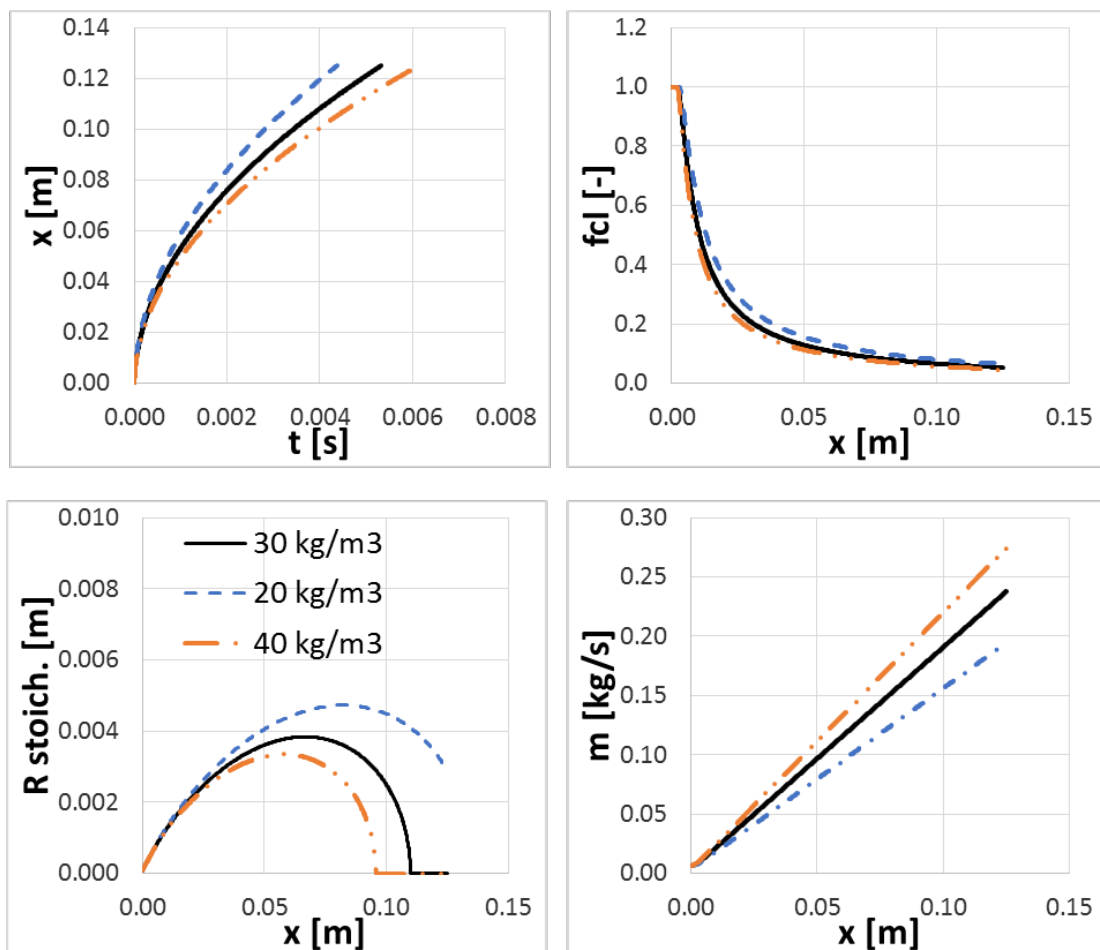
- Air density effect (cases 1-6-7):

Figure 10: Extract from students report regarding the air density influence

"In this case, the spray encounters a denser ambient, lowering its penetration due to aerodynamic effects and producing a shorter stoichiometric length. The air mass flow entrained is higher due to the fact that the air is denser, so similar volume entrainment results in higher mass. Overall, it can be seen that higher air density induces faster mixing."
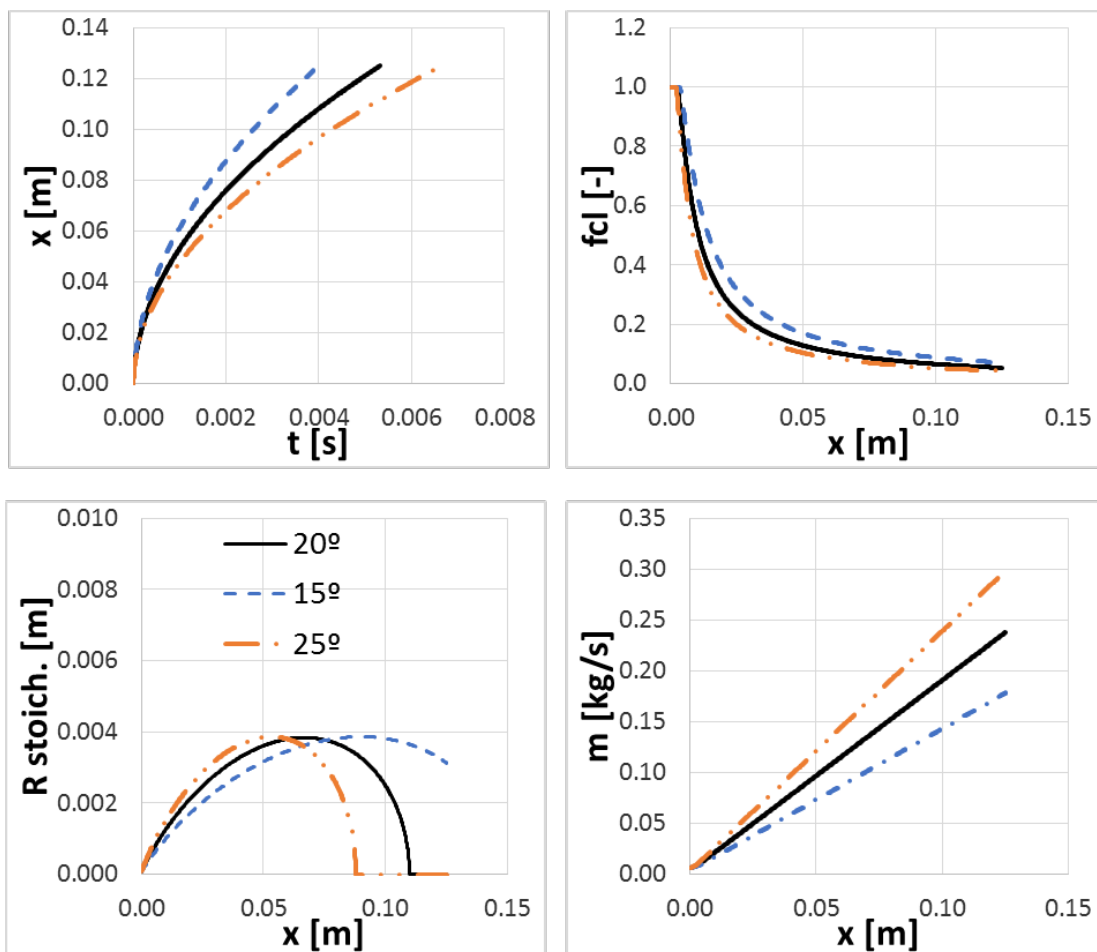
- Spray cone angle effect (cases 1-8-9)

Figure 11: Extract from students report regarding the spray cone angle influence

"On a turbulent spray, increasing the cone angle is equivalent to an increase of the turbulent viscosity, so it means that momentum transfer is enhanced and turbulent

Reynolds reduced. This is translated in higher entrained air (better mixing) and a subsequent reduction of the stoichiometric length."

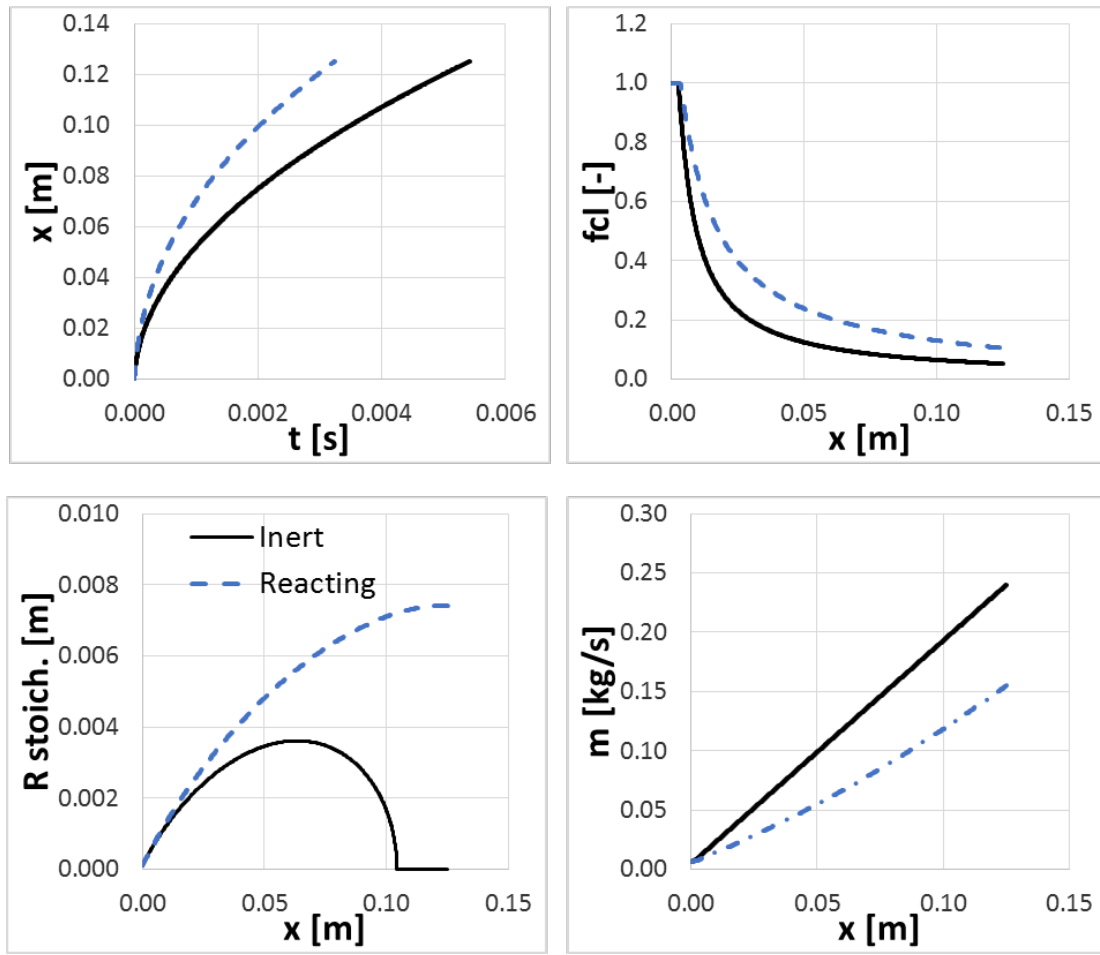c) <u>Inert vs. reacting comparison</u>

Figure 12: Extract from students report for the inert vs. reacting comparison

"From the graphs, it can be seen that combustion development inhibits air entrainment, which is perceivable by the lower total mass flow and the slower decay of velocity and mixture fraction. This can be due to effect of heat release lowering the local density, and hence the mixing rate".

As it can be seen in the extracts from the students' reports, the main objectives that were previously claimed for the different studies proposed are well fulfilled. This can be

seen as a first confirmation of the suitability of the methodology introduced to improve their learning process and achieving the teaching objectives established.

4.2. Assessment results

After the computer session is completed for the different groups of students, a specific test is made to better evaluate its impact on the students learning process. Figure 13 shows a historical view of the marks (in a scale 0-10) achieved by the Aerospace Engineering students on the practical session since academic year 2013/14, where the degree was implemented. During all these years the same methodology proposed in the current paper was applied. The blue line represents the average mark, which is plotted together with the standard deviation as error bars. As it can be seen, the average mark is very high, over 9/10 in all cases, which confirms the high level of learning achieved by the students during the practical session. Indeed, the minimum mark achieved is also represented in the orange line, showing all values over 7.5/10.
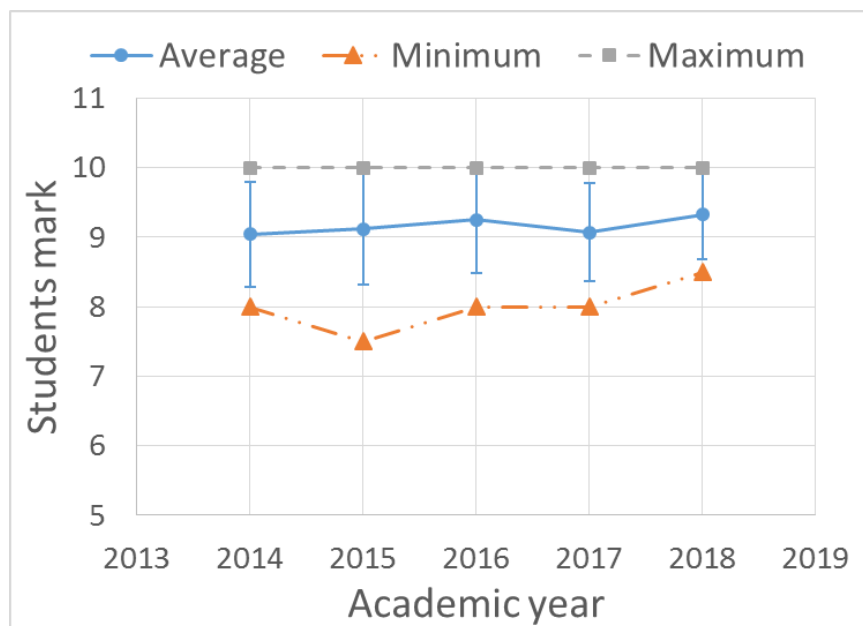

Figure 13: Historical marks achieved by the students

It has to be considered that the total average mark for the subject, including all different assessment performed along the semester, is on 6.6/10 average. Considering this, it can be concluded that the learning performance achieved thanks to the methodology described in the current paper is significantly over the one achieved by other conventional methodologies, highlighting the positive impact of its introduction.

4.3. Student opinion

At the end of the practical session, a satisfaction survey is provided to the students. Their responses are filled out anonymously and treated in a statistical manner. This survey has the following structure:

1. Global assessment of the practical session

2. The contents:

    2.1. Are interesting.

    2.2. Are related with the theoretical contents of the subject

    2.3. Apply the theoretical contents of the subject

3. The professor:

    3.1. Masters the subject

    3.2. Explains clearly

    3.3. Started and finished the session on time

4. The resources:

    4.1. Are appropriate for the session development

    4.2. Are enough

    4.3. The room is comfortable

5. The information regarding location and schedule:

    5.1. Is enough

    5.2. Was available sufficiently in advance

From the point of view of this paper, only the answers to questions 1, 2.1, 2.2 and 2.3 will be considered, since they are the ones related to the teaching methodology proposed. In this case, the scale for the responses is 0-5. This information is available in Figure 14. Looking at this figure, it is appreciable how the students' satisfaction is very high, always over 4.5/5. Therefore, not only the methodology is useful from a teaching-learning process perspective, but it is also well received by the students'.
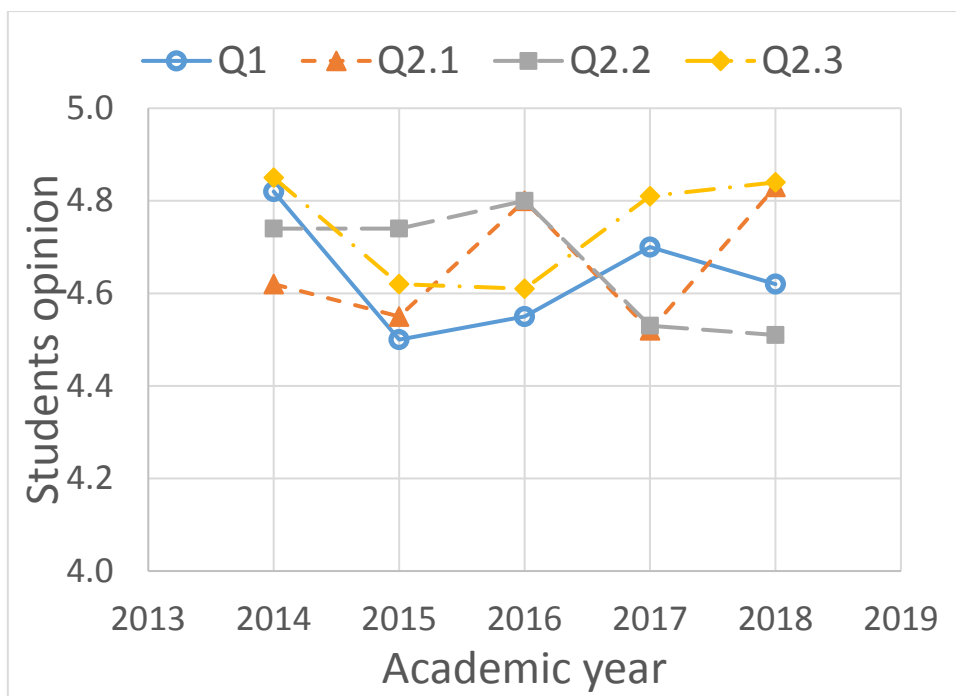

Figure 14: Historical evolution of students' opinion based on satisfaction surveys

5.   Conclusions

In the current work, a computer-aided learning methodology is proposed for teaching diffusion flame fundamentals for Aerospace and Mechanical Engineering students. For

this purpose, a one-dimensional spray model, previously developed for research purposes, has been adapted and coded in C++. Using this tool, the students are proposed to identify and analyze the effects of some critical parameters, such as the nozzle outlet diameter, the ambient density and the spray cone opening angle on the fuel-air mixing process on inert conditions. Additionally, the students perform also an analysis of the spray structure change once combustion takes place. This allows to complement the simple Spalding and Schlichting solution for laminar isodense gas jets developed during the standard classroom sessions.

From the students' report, it can be observed that the level of comprehension achieved by the combination of the theoretical work previously developed in the classroom and the use of the spray model is very high. The students are able to identify not only the main trends regarding the mixing field, but to analyze in a proper way the root causes for the behavior observed. This is translated in the very high marks achieved by the students on the assessment of the session, which in average is always over 9/10. Additionally, the students appreciation of the methodology is also high, as it can be seen for the average students satisfaction survey results, always over 4.5/5.

Nomenclature

A	Area
d	Diameter
f	Mixture fraction
fLOL	Binary flag to activate reacting calculation
h	Specific enthalpy
I	Momentum flux
i	Index for discretization step
K	Constant for Gaussian radial profile
P	Pressure
r	Radial position
S	Spray maximum penetration
Sc	Schmidt number
t	Simulation time
T	Temperature
u	Velocity
V	Volume
x	Axial position

Greek symbols

ΔP	Pressure difference across the nozzle

ζ	Non-dimensional velocity, $\zeta = u(x,r) \big/ u_{cl}(x)$
ϑ	Spray cone opening angle
ρ	Density

Subscripts

0	Standard conditions
∞	Ambient conditions
a	Air (in the combustion chamber)
cl	Centerline (spray axis)
f	Fuel
o	Nozzle outlet
st	Stoichiometric

References

[1]   IPCC. Intergovernmental Panel on Climate Change Working Group I. Climate

Change 2013: The Physical Science Basis. Long-term Climate Change:

Projections, Commitments and Irreversibility. Cambridge Univ Press New York

2013:1029–136. doi:10.1017/CBO9781107415324.024.

[2]   Weilenmann M, Soltic P, Saxer C, Forss AM, Heeb N. Regulated and

nonregulated diesel and gasoline cold start emissions at different temperatures.

Atmos Environ 2005;39:2433–41.

[3]   Kirchstetter TW, Harley RA, Kreisberg NM, Stolzenburg MR, Hering S V. On-road

measurement of fine particle and nitrogen oxide emissions from light- and

heavy-duty motor vehicles. Atmos Environ 1999;33:2955–68.

doi:10.1016/S1352-2310(99)00089-8.

[4]   Archer G. Briefing: Particle emissions from petrol cars. Transp Environ 2013:1–4.

[5]   Ben Naceur K, Cozzi L, Gould T. World Energy Outlook 2016. 2016.

doi:10.1787/weo-2016-en.

[6]   Nesbit M, Fergusson M, Colsa A, Ohlendorf J, Hayes C, Paquel K, et al.

Comparative Study on the differences between the EU and US legislation on

emissions in the automotive sector. 2016.

[7]   www.upv.es. Univ Politècnica València n.d.

[8]   Battin-Leclerc F. Detailed chemical kinetic models for the low-temperature

combustion of hydrocarbons with application to gasoline and diesel fuel

surrogates. Prog Energy Combust Sci 2008;34:440–98.

doi:10.1016/j.pecs.2007.10.002.

[9]   Aleiferis PG, Behringer MK, Malcolm JS. Integral Length Scales and Time Scales

of Turbulence in an Optical Spark-Ignition Engine. vol. 98. Flow, Turbulence and

Combustion; 2017. doi:10.1007/s10494-016-9775-9.

[10]   Dumouchel C, Cousin J, Triballier K. On the role of the liquid flow characteristics

on low-Weber-number atomization processes. Exp Fluids 2005;38:637–47.

doi:10.1007/s00348-005-0944-1.

[11]   Sick V, Drake MC, Fansler TD. High-speed imaging for direct-injection gasoline

engine research and development. Exp Fluids 2010;49:937–47.

doi:10.1007/s00348-010-0891-3.

[12]   Zhao H, Ladommatos N. Optical diagnostics for soot and temperature

measurement in diesel engines. Prog Energy Combust Sci 1998;24:221–55.

doi:10.1016/S0360-1285(97)00033-6.

[13]   Fansler TD, Parrish SE. Spray measurement technology: a review. Meas Sci

Technol 2015;26:012002. doi:10.1088/0957-0233/26/1/012002.

[14]   Edmonds E. Where Next in Computer Aided Learning? Br J Educ Technol

1980;11:97–104.

[15]   Perumal K, Ganesan R. CFD modeling for the estimation of pressure loss

coefficients of pipe fittings: An undergraduate project. Comput Appl Eng Educ

2016;24:180–5. doi:10.1002/cae.21695.

[16]   Regueiro A, Patiño D, Míguez C, Cuevas M. A practice for engineering students

based on the control and monitoring an experimental biomass combustor using

labview. Comput Appl Eng Educ 2017;25:392–403. doi:10.1002/cae.21806.

[17]   Gutiérrez-Romero JE, Zamora-Parra B, Esteve-Pérez JA. Acquisition of offshore

engineering design skills on naval architecture master courses through potential

flow CFD tools. Comput Appl Eng Educ 2017;25:48–61. doi:10.1002/cae.21778.

[18]   Burke RD, De Jonge N, Avola C, Forte B. A virtual engine laboratory for teaching

powertrain engineering. Comput Appl Eng Educ 2017;25:948–60.

doi:10.1002/cae.21847.

[19]   Spalding DB. Combustion and mass transfer. Pergamon Press; 1979.

[20]   Schlichting H. Boundary-Layer Theory. McGraw-Hill; 1978.

[21]   Desantes JM, Pastor JV, Garcia-Oliver JM, Pastor JM. A 1D model for the
description of mixing-controlled reacting diesel sprays. Combust Flame
2009;156:234–49. doi:10.1016/j.combustflame.2008.10.008.

[22]   Desantes JM, Pastor JV, Garcia-Oliver JM, Briceño FJ. An experimental analysis
on the evolution of the transient tip penetration in reacting Diesel sprays.
Combust Flame 2014;161:2137–50. doi:10.1016/j.combustflame.2014.01.022.

[23]   Payri R, Salvador FJ, Gimeno J, Novella R. Flow regime effects on non-cavitating
injection nozzles over spray behavior. Int J Heat Fluid Flow 2010;32:273–284.

[24]   Payri R, Garcia-Oliver JM, Salvador FJ, Gimeno J. Using spray momentum flux
measurements to understand the influence of diesel nozzle geometry on spray
characteristics. Fuel 2005;84:551–61. doi:10.1016/j.fuel.2004.10.009.