Behaviour of fuel droplet evaporation injected in a thermal convective counter flow

A. Amoresano[1], C. Allouis[2]

1. Dipartimento di Ingegneria Meccanica per l'Energetica - Università di Napoli "Federico II", Via Claudio 21 - 80125 Naples- Italy
2. Istituto di Ricerche sulla Combustione-CNR Piazzale Tecchio 80 125 Napoli

The aim of this paper was to study the interaction between kerosene droplets and a hot counter air flow. Fast imaging was employed to better understand what were the mechanisms that govern the self-sustaining combustion in Mild Combustion condition. Working in counter flow conditions allowed us to reduce the area of the evaporation and burning phenomena and verify with fine precision the condition of ignition and burning phenomena. This preliminary work allowed to recover numeric information about droplet size, thermal field and dynamic field in counter flow injection condition.

Context and needs

Heavy fuel oils accounted for about 100 million tonnes in oil consumption in Europe in 1999, mainly for boiler use and for power generation, and about 90 million tonnes in North America, according to BP Amoco Statistical Review of World Energy 2000 (see Fig. 1); although the consumption of heavy fuel oils has been reduced by two thirds in the last thirty years due to the increasing use of natural gas, it represents a widely exploited source of energy, accounting yearly almost half of the consumption of gasoline.

Heavy oils constitute a cheap energy source: average price in Europe per energy unit supplied is about 20% lower than the price of natural gas (8 Euro/GJ against 10 Euro GJ – January 2001).

The exploitation of heavy fuel oils or other not precious liquid combustibles for heat generation through conventional combustion technologies is limited *de facto* by environmental policies, since they contain heavy hydrocarbons and undesired amounts of nitrogen and sulphur, which form NO_X and SO_X during standard combustion processes in an oxygen rich environment.

Nevertheless, they are economical alternatives for power generation due to their low cost, provided that they can be used in compliance with environment protection, heavy oils constitute a major energy source, which could be used for medium/large sized heat/power generation, providing an improved and full exploitation of natural resources.

In order to face up to the above mentioned constraints, in the last decade a new mode of developing combustion has been developed, referred to as MILD COMBUSTION [1] in the present document, which shows an outstanding potential to be still adequately developed.

The basic design concept of mild combustion is contrary to the traditional burner design where fuel and oxidant are intimately mixed in a burner port for stable and rapid combustion.

Mild combustion is characterised by the recovery of effluent gases, generally used for diluting and pre-heating the reacting stream; fuel combustion takes place in higher volumes, creating *flameless* or *monochromatic* flames, where the gradient temperature is limited and the average temperature is lower than the flame temperature relative to classical combustion systems; in

these conditions, all the elements, even the most complex, have the required time to undergo a full combustion.
Mild combustion allows to extract the whole energetic power of the fuel and in the same way to reduce polluting emissions.

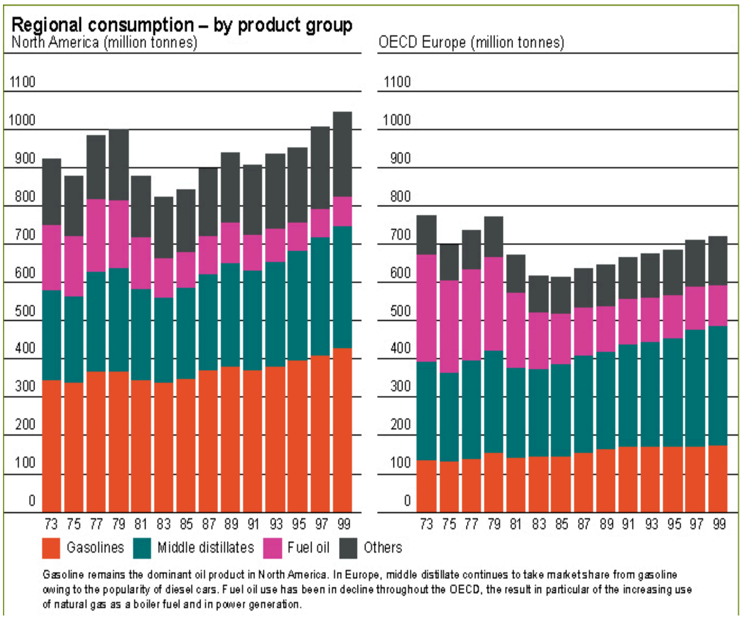

Figure 1 – Regional consumption of oil products (BP Amoco Statistical Review of World Energy 2000)

Fluid dynamic aspects
The way to manage the high temperature of the reactants and to keep the product temperature low will be dealt taking into account plant constraints, like fan characteristic at high temperature and chemical physical processes, like reaction rates of the possible fuel. The first aspects are dominant in the analysis of the strategy chosen for mixing the air with flue gases, because the material resistance to the temperature is the fundamental criterion to be followed. In the second case the challenging target is to consider the unique possibility to create conditions comparable to those obtained in Well Stirred Reactors (WSR).
In this case it is possible to extract heat from the reaction volume and to lower the product temperature straightly in the reaction zone. In other words the feasibility of the creation of an adiabatic WSR in this regime is the fundamental answer to the requirement of keeping as low as possible the products stream temperature , because this means that the reaction times are sufficiently small that fluid-dynamic controlled process, like complete mixing of the products with the reactants and the convective heat exchange, can reasonably take place.
The practical feasibility of a WSR is related to the feasibility of high level of back mixing [2].
A general rule to yield such condition is to create reverse flows, which are obtained by impinging jet on walls or on other jets. This allows to split the main streams in parts, which are comparable, whereas shedding vortices, swirled flows can ensure only fractional recirculation of the main stream.
A partial increase of the kinetic energy of the recirculated stream can be obtained with the injection of the fuel inside the reverse flow, but this increases only partially the mass fraction of the revere flow.
The creation of impinging jets is made difficult by the high temperature of undiluted combustion because the reactor confinement has to be performed by means of cooled surfaces. This constraint is partially released in mild combustion systems, because the product

temperature is kept relatively low. Therefore the first possible design of a WSR can neglect cooling systems, which can be taken into consideration in the second detailed design only when the heat extraction can be also considered as an additional improvement of the combustion stability and pollutant control.
The second aspect, which pertains the specific conditions of mild combustion, is that this regime entails an increase of the ignition delay, but it keeps the reaction time of the oxidation process order of magnitudes lower than the ignition delay times. The chance, that slow combustion can be sustained in some particular intermediate temperature range for other types of fuels, has to be considered still an open question, but it is a challenging task to show that this can occur with a suitable rate for practical application The great difference between the pre-combustion and combustion time entails that the WSR has to be designed with a very long average residence time either inside the reactor itself or in the inlet mixing zones. A high value inside the reactor is desirable, because it allows keeping a high level of tolerance respect to the mixing time and consequently respect to fuel types, dilution level and initial temperature. The third aspect is related to the application of the mild combustion in the cleaning process of the flue gas streams. In this case it is suitable to have a high level of external accessibility in the sense that injection both of the fuel and of other possible additives as well as heat extraction is favoured when this can performed on the external confinement.

Experimental

In respect with the previous considerations, it appears evident that it becomes important to understand better the dynamic flow field inside the burner or how to manage the flame set-up when working in mild combustion condition. A low air index the flame is not able to self-sustain the combustion process.
This aspect is a very atypical and complicated process. The aim of this paper is to study the auto burning phenomenon and to define the parameters able to maintain the flame in burning conditions. In figure 2 is presented the facility used to analyse the behaviour of the fuel jet injected in a hot environment at atmospheric pressure.

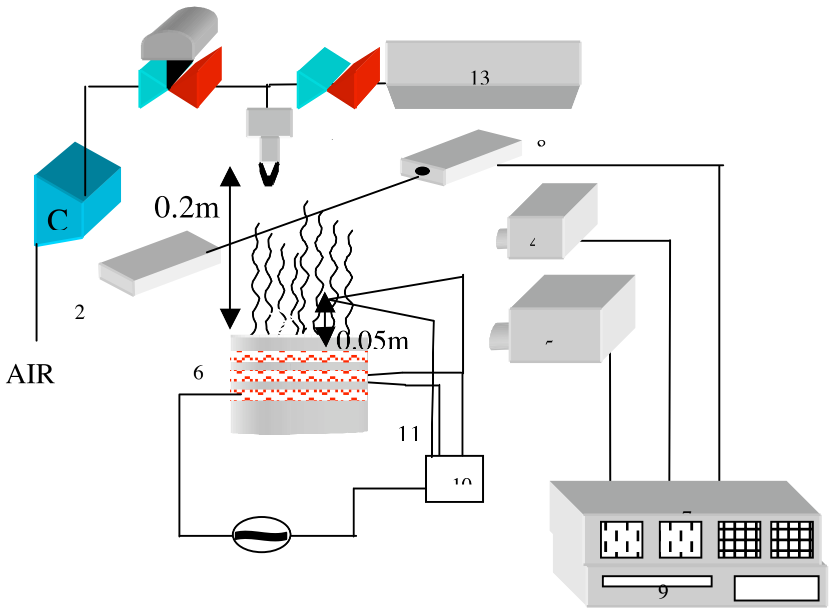

Figure 2 1.droplet generator; 2.laser;3.photodiode;4. CCD camera 1;5. Infrared camera 6. electric furnace; 7. image recorder; 8. external trigger; 9. personal comp. –

Experimental apparatus

The furnace (6) generates a hot air in free convection or in turbulent convection in a range of temperature up to 1000 K. The setup is controlled by means of a thermocouple inserted in the furnace. A second thermocouple measures the temperature near the hot surface and a third one the temperature at 100 mm high from the hot plate.

To manage the injector, a remote control system was manufactured .It used a pressure regulator with an electrovalve. In this way, it was possible to have, with a good confidence 1-2% of accuracy, the same pressure value and so the same flow dynamic behaviour at the outlet of the injector. It this way it is possible to control the following parameter:

1. Pressure of the nozzle
2. Temperature inside the furnace
3. Temperature outside the furnace and near the spray cone angle
4. Spray velocity
5. Particle diameter

Droplets and convective flow field analysis

The tests done with the previous described apparatus were performed using Kerosene as fuel. The aim of these test is to understand the behaviour of the cone profile distribution when a hot stream of air or nitrogen (to avoid the combustion of the fuel) is fluxed in opposition with the fuel injected.

The tests have been carried out changing time by time the limit conditions of the thermal flow field generated by the furnace. In figure are reported the most significant results, especially those related to the qualitatively behaviour. The following diagram shows the behaviour of the convective flow generated by the furnace taking into account the expression (1):

$$\Delta p = g(\rho_{in} - \rho_{out})H \qquad (1)$$

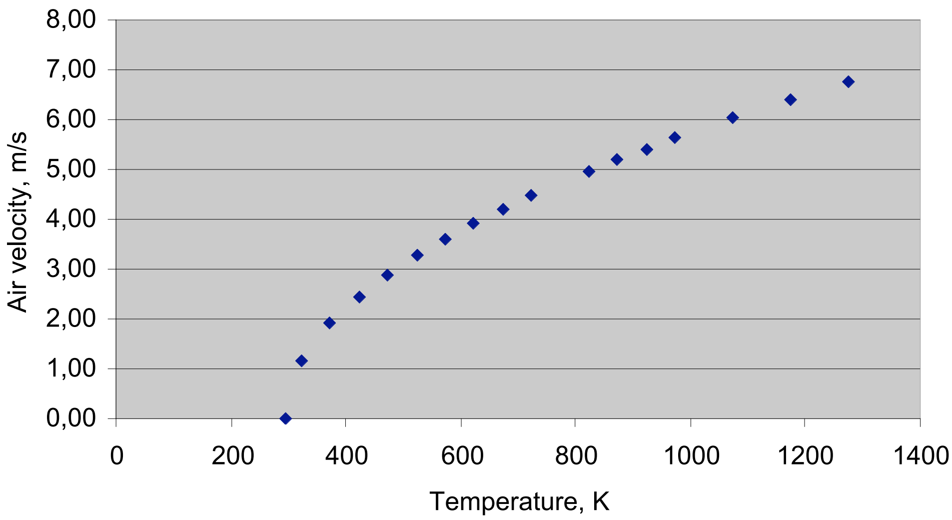

Figure 3 - Air velocity in free convection mode.

An analysis of the behaviour of the kerosene droplet diameters ranging between 0.05-0.005 mm, permits to verify where the dynamic behaviour of the droplets depends on the convective flux of the hot air coming from the furnace.
Analysing the following formula of the local force [3,4] on the droplets:

$$M\cdot\frac{D_a\overline{V}_a}{Dt}=-\frac{3m}{4d}\cdot C_d\cdot(\overline{V}_a-\overline{W}_a)\cdot\left|\overline{V}_a-\overline{W}_a\right|-\frac{m}{\rho}\cdot\nabla p+$$
$$+(m-M)\cdot g\cdot\nabla(r\cdot sen\varphi)+m\cdot C_v\left(-\frac{1}{\rho}\cdot\nabla p-\frac{D_a\overline{V}_a}{Dt}\right)+\qquad(2)$$
$$+\frac{9m}{d}\sqrt{\frac{\nu}{\pi}}\int_{\tau=t_0}^{\tau=t}\frac{d\cdot\left(\overline{V}_a-\overline{W}_a\right)}{d\tau}\frac{1}{\sqrt{t-\tau}}d\tau$$

Analysing each term of the equation it is possible, in a first approximation to rewrite the equation (2) as following [5]:

$$d\mathbf{u}_p/dt=-\alpha(\mathbf{u}_p-\mathbf{u})+\mathbf{g}\qquad(3)$$

Where α is:

$$\alpha=3\mu C_d Re/4\rho_p d_p^2\qquad(4)$$

and $1/\alpha$ is the time that the droplet takes to match the velocity of the gas phase. In figure 4 are reported the values of $1/\alpha$ of the droplets generated by the nozzle.

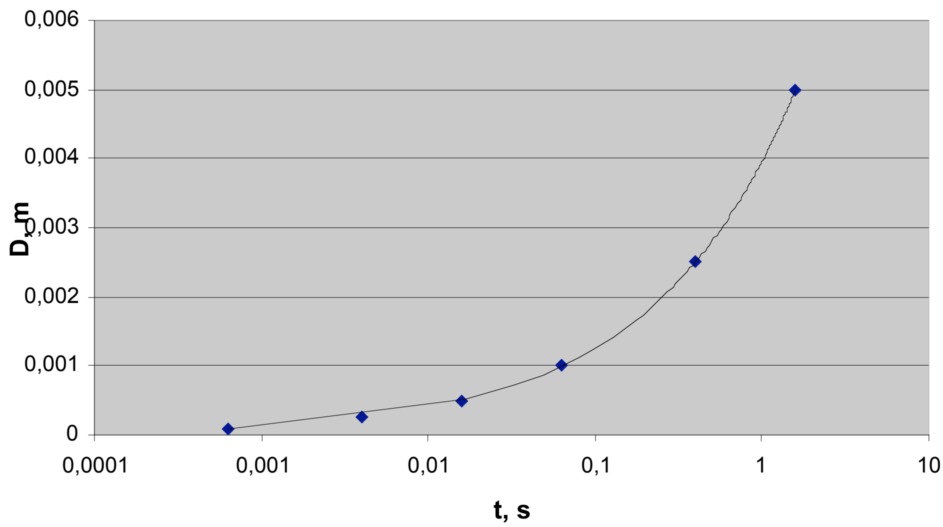

Figure 4 – Droplet equalization time.

The equalization time is regulated by the following equation

$$d_p=0.004t^{0.5}$$

In this way, it is possible to verify, in a first approximation, which are the droplets that are affected by the thermal convective field. A verification is presented in the figure 5 where the distance from the nozzle vs. droplets diameter.

This figure evidences a particular region where the equalization distance from the nozzle is lower than the distance between the nozzle and the throat of the furnace and, far from this last one about 0.05 m. In fact, the sensitive zone of the convective thermal field finishes after 0.05 m due to the great heat transfer with the surrounding environment. This small region is crossed by droplets with diameter greater than 0.001 m, while there is a stagnation region for smaller droplets. In respect to the boundary conditions considered here for the tests, the droplets with this diameter, or smaller, are very sensitive to the convective flow.

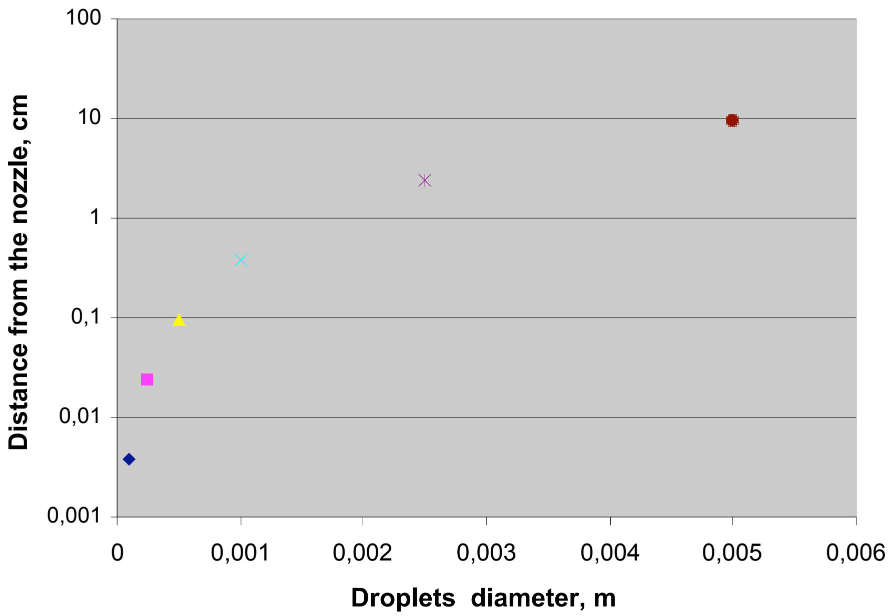

Figure 5 – Equalization distance vs. droplet diameter.

Results

The tests were performed with a fast CCD camera operating at 300 frame/s. The diameters generated by this nozzle ranged between0.001-0.005 m.

The boundary conditions of the first test are: fuel temperature, T_f = 293 K; atomisation pressure of the air was fixed at 2 atm. In this case, liquid fuel and gas had the same temperature, and the convective effect of the air lifting up from the furnace exit was completely negligible. In this case the instability of the spray was only due to the stress between the droplets and the external air.

he photo in fig 6 shows the behaviour of the spray in cold conditions. It is possible to observe the perfect development of the entire spray cone. It appears symmetrical and has an undistorted shape. The droplets arrive undisturbed up to the furnace exit.

The second photo (Fig 7.) represents the spray visualization with the following conditions: T_f = 293°K, T_{in}= 723 K. The frame evidences that the spray does not touch the exit hole of the furnace. The droplets equalise their velocity nearby the exit of the furnace at 0.03-0.06 m far from this one. In this condition and at this distance from the furnace exit, the stagnation condition arises. In this region, the droplets evaporate because of their permanence in the hot stream due to the convective flow field, that in this case reached a temperature of T_{out}= 500 K and a velocity of 2.3 m/s.

The figure 8 represents the behaviour of the spray when T_f = 293 K, T_{in}=993 K. The velocity of the hot air due to the free convection flow field is now of 5 m/s. The droplets are strongly applied by the counter hot air stream. The evaporation or stagnation region is 0.1 m far from the mouth of the furnace. The little droplets evaporate and begin to be very reactive. In this case burning conditions are present.

Fig.6

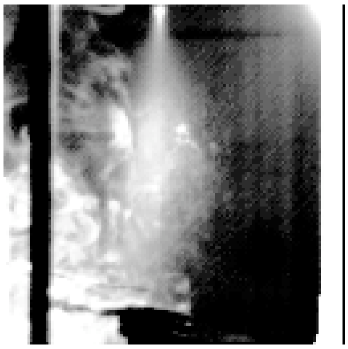

Fig.7

Fig.8

Conclusions

In this paper, we evidenced the behaviour of a spray injected in a free convective counter flow. Although this kind of application had a small industrial impact, we considered that this analysis could be very interesting to understand what are the mechanisms that govern the free combustion. It was an important input for the knowledge of the role of the heat and mass fluxes inside just before the droplet combustion. Thus, working in counter flow conditions allowed us to reduce the area of the evaporation and burning phenomena and verify with fine precision the condition of ignition and burning phenomena. The extension of this initial work foresee to recover numeric information about droplet size, thermal field, and dynamic field, to support the choices especially in the applications for mild combustion with more complex fuel like heavy oil.

References

[1] de Joannon M., Langella G., Beretta F., Cavaliere A., Novello C., 2000, *Combust. Sci. Tech.*, **153**, 33-50.

[2] de Joannon M., Saponaro A., Cavaliere A., 2000, *Proc. Comb. Inst.*, **28** .

[3] Clift R., Grace J.R. and Weber M.E., 1978, Bubble Drops and Particle, Academic Press.

[4] Durst F., Melling A. and Whitelaw J.H., Principles and Practice of Laser Doppler Anemometry, 1976, Academic Press.

[5] Adinolfi Falcone R., Amoresano A., Novello C., 1992, *Proceeding of the 47th ATI Congress, Florence – Italy*, 1233.