CHALLENGES IN MODELLING INDUSTRIAL BURNERS FOR DESIGN PURPOSE: A CASE STUDY FOR GASEOUS AND LIQUID FUELS

G. Rossiello*, L. Morandi, D. Carucci**, D. Ettorre***, T. Vela*, S.B. Ahmadpanah*, and M. Torresi*****

Corresponding author: gianluca.rossiello@seamthesis.com (Gianluca Rossiello)
* SEAMTHESIS Srl, Via IV Novembre, 156 – 29122 Piacenza, ITALY
** Termotecnica Industriale Srl, via Dante, 16 – 20121 Milano, ITALY,
*** DMMM, Department of Mechanics, Mathematics and Management, Polytechnic University of Bari, Via Re David, 200 – 70125 Bari, ITALY

Abstract
This article deals with the use of CFD simulations as design tool for the development of heavy-duty burners. The newly designed 50 MW burner operates, singularly in a combustion chamber, with both gaseous and liquid fuel. The focus is on two aspects of modelling: one related to gaseous fuel, and the other to low boiling point liquid.
First, for refinery gas feeding, attention is drawn on the geometry and boundary conditions suitable for the injection nozzles of external spuds, considering either the injection directly on exit surfaces of nozzles on spud tip, or the internal volume of the spuds and each hole geometrically included in the computational domain. The two simulations show no relevant difference for the given ratio between hole diameter and depth.
As a second example, the liquid fuel feeding is considered for the same burner: in this case the challenges come from light naphtha feeding, due to low boiling point. Indeed, it is shown how, employing a standard atomizer designed for light oil, the functioning of the burner inside the combustion chamber is totally unsatisfactory, with delayed combustion and carbon monoxide at furnace exit. For modelling validation purpose, the same numerical test is run with a gasoil, obtaining a suitable solution, well known for the coupling of this kind of air/steam assisted atomizer with gasoil. This second test proves the suitability of such numerical modelling: indeed, in the end it is shown a third result obtained by completely redesigning the atomizer to operate with low boiling point fuel, showing a completely satisfactory behavior.
Finally, this study underlines how CFD simulations give quantitative and affordable support to designers for such complex and challenging problems, anticipating and replacing, to a certain extent, experimental testing, in the early stages of design and when testing is not viable for some reason.

Context and motivation
The ongoing energy transition from fossil to renewable sources is a real challenge for industry and for the energy market, requiring a combination of fuel flexibility,

thermal efficiency, and low emissions to properly manage fossil fuels, waste, and new and renewable energy carriers, both gaseous and liquid, reducing fuel consumption and expensive after-treatment of flues [1].
In this context, advanced design tools are needed to meet market needs: in particular, for steam boiler design, it is essential to improve and refine simulation capabilities to properly handle a wide range of liquid and gaseous fuels with performance even exceeding that of standard fuels [2-4]. Experimental campaigns are extremely expensive, with few plants available for full scale testing, and require careful commissioning and management to give accurate measurements. Moreover, the testing give back, as best option, affordable performance parameters, but nothing regarding the interpretation of such results, neither in depth understanding of flame behavior or physical phenomena taking place in general. In this context, CFD analysis can be an extremely useful and strategic tool both for the analysis and development of individual equipment – for example burners, air intake openings, etc. – both for the optimization of the whole system.

Problem description
The object of the present study is the modelling of a single heavy-duty industrial burner of a boiler furnace fed with both gas and liquid fuel. The burner has a total of 9 gas lances with three nozzles each, and a Y-jet air/steam assisted spray atomizer. In the case of liquid fuel supply, a standard atomizer is first studied, comparing the performance obtained with light naphtha supply with that resulting using the physical properties of standard diesel fuel. The operating conditions considered are 100% load corresponding to 48.71 MW of thermal input for refinery gas, and 48.35 MW thermal input for liquid fuel. The compositions and main physical properties of gaseous and liquid fuels are summarized in Table 1 and Table 2, respectively.

Table 1. Refinery gas composition [% Volume] and physical properties.

	CH4	H2	C2H6	C3H8	N2	MW	LHV
Units	%	%	%	%	%	kg/kmole	MJ/kg
	51	25.5	4	16.2	3.3	17.967	47.818

Table 2. Liquid fuel composition [% Volume] and physical properties.

	C6H6	C5H12	C6H14	MW	LHV
Units	%	%	%	kg/kmole	MJ/kg
	54	23	23	78.312	42.717

Numerical modelling and boundary conditions.
CFD analyses are conducted with the Ansys Fluent-v17.2 [5]. The computational domain is discretized with a polyhedral mesh of about 7.5 million elements with orthogonal prismatic cells next to the walls. For each configuration, a steady state computation is performed with turbulence modeled by a RANS approach and

standard 2 equations closure. The non-premixed model is used to account for gas phase reaction, with tabulated chemistry by a probability-density-function approach based on the hypothesis of chemical equilibrium, with tabulated data (PDF-Table) computed in pre-processing. Compressible ideal gas approach is employed for the equation of state and in addition, the domain-based weighted sum of grey gases model is used to calculate the gas radiative properties and the radiant heat transfer is calculated by the Discrete Ordinates method (DO). The Discrete Phase Model available in Ansys Fluent is employed for description of liquid droplets, with constant velocity of 70 m/s and direction perpendicular to the nozzle exit surface and with a Rosin-Rammler distribution for the diameter of particles, with 70 μm for mean value, 21 μm as minimum one, and 140 μm for the maximum.
The vaporization temperature and boiling temperature used in the simulations are respectively 262 K and 353 K for the light naphtha, and 400 K and 589 K for the fuel-oil. The flow rate of combustion air input is calculated by the design value of the excess air, which is 12.5% for both gaseous and liquid fuel full load operation. For the furnace walls, a convective heat transfer coefficient of 2800 W/m^2-K is considered, which also accounts for the phenomenon of fouling, with water-cooling temperature of 255 C. For radiation, internal emissivity coefficient is set to 0.6.

Directionality testing with gaseous fuel

The first case study shown in this work concerns the gaseous fuel modelling and operation of the burner in the combustion chamber of the steam generator. The correct representation of boundary condition is at least as essential as the proper and modelling in order to obtain accurate results, with reliable assessment of flame behavior affordable estimation of relevant performance parameters. Hence, the focus here is on the injection of fuel jets into the furnace from external gas spuds nozzles. The physical phenomenon under investigation is the supercritical discharge of a high-pressure gas flow from convergent nozzles into the low-pressure volume of the combustion chamber. The well-known flow is characterized by conical oblique shocks, high Mach number internal jets, and successive normal shocks alternated with expansion fans. Through this complex flow structure, the adaptation of fuel streams to the downstream low-pressure field occurs. The direction and momentum of such fuel jets is affected by the modelling of flow and the representation of boundary conditions: while the former aspect is described in the previous section, the latter is discussed here. In particular, two alternatives are considered. In a first simulation, the internal volume of gas spuds is included in the domain, and the nozzles are modelled as passage ducts, discharging into the furnace. In a second simulation just the outlet section of the nozzle on the tip of each spud are considered as inlet surfaces. So, in the first case an overall mass flow rate is imposed for each of eh 9 spuds, while in the second it is imposed on each of the 27 nozzles (3 nozzles on each spud). The two simulations show similar results, almost overlapping, both for the streamlines of jests (see Figure 2) and for the flame behavior. This proves that the ratio of nozzle diameter to the length of the hole (12 mm and 15 mm, respectively) is sufficiently high to make upstream inertial effect and 3D flow behavior through the drilled volumes almost negligible. Finally, to illustrate the flame shape and the burner-furnace behavior, we show in Figure 3 the contour plot of temperature and carbon monoxide fields.

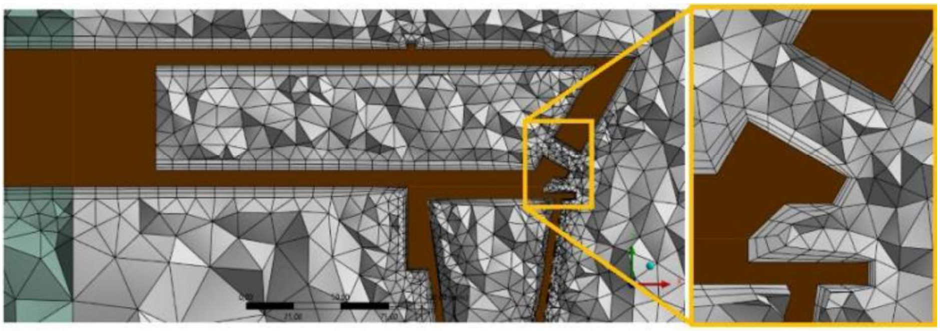

Figure 1:Geometry used for case 1, with nozzle volume modelled.

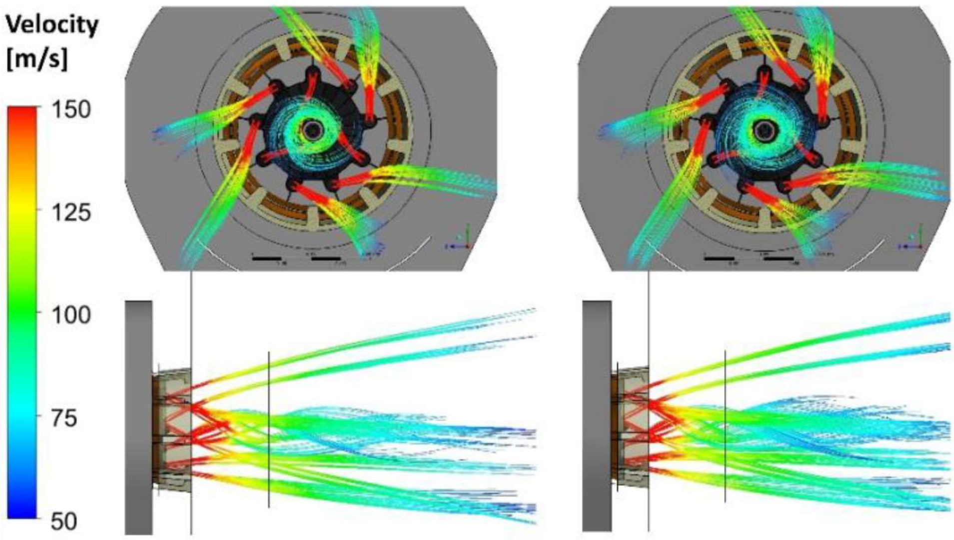

Figure 2 Streamline of fuel (refinery gas) coming out from nozzles, coloured by velocity magnitude: model with holes included (left) vs model with exit surfaces.

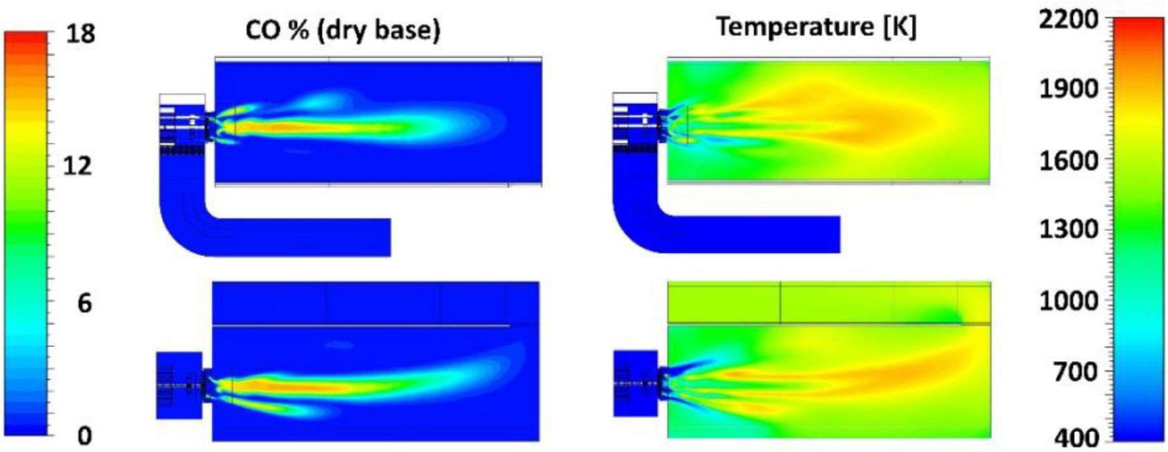

Figure 3:Refinery gas CFD solution: temperature (right) and CO (left) fields.

Analyses of spray combustion for liquid fuel feeding

The second part of this study is focused on liquid fuel combustion modelling of the burner inside the combustion chamber. In particular, three results are shown. The first one refers the simulation with light naphtha feeding, a low boiling point liquid fuel, using a "standard" Y-jet air-assisted atomizer. The solution obtained for the flame shape is completely unsatisfactory, as shown in the first column of Figure 4, since the combustion is delayed, and CO oxidation is not complete at combustion chamber exit. This behaviour is due to extremely fast evaporation of liquid fuel at nozzle atomizer exit, as is clearly visible in the picture in the third (last) row: this issue can be imputed either to modelling inaccuracy, or to bad atomizer operation.

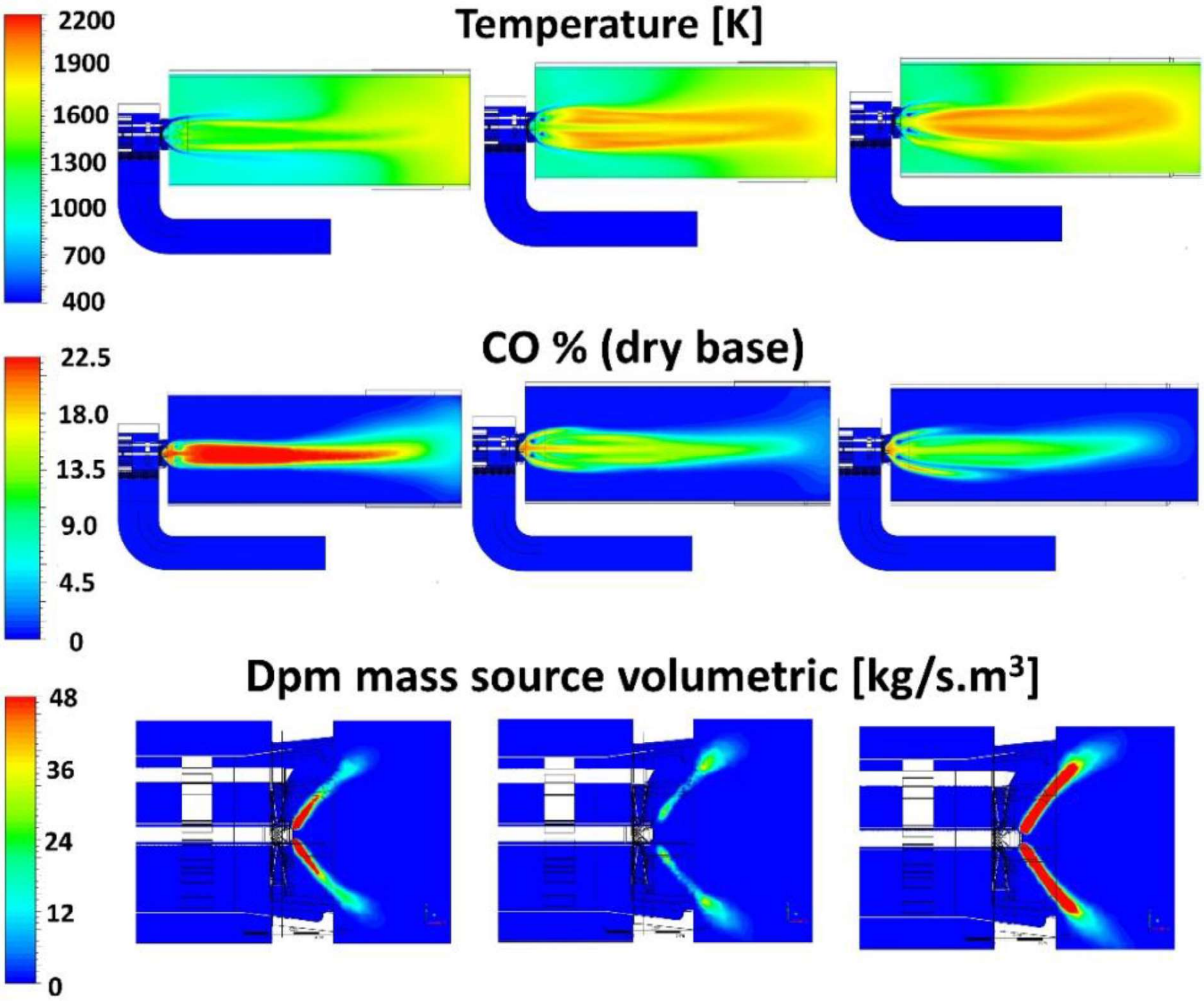

Figure 4: Liquid fuels CFD solutions: temperature (first row), CO (second row) and vapor mass source (third row) fields; the first column refers to Case-1 (light naphtha and initial atomizer), the second column refers to Case-2 (fuel-oil and initial atomizer) third column refers to Case-3 (light naphtha, redesigned atomizer).

To address this point, a second simulation is taken into consideration, which is identical to the first one except for the liquid droplets, for which fuel-oil physical properties replace those of light naphtha. The idea is to replace a low-boiling point liquid fuel, with a diesel oil for which this kind of atomizers has been developed, giving a well-known and satisfactory flame behaviour. Indeed, in the second column of Figure 4 the numerical solution of this second case is shown, proving that the overall model captures the expected flame characteristics. This step is used as a validation of the modelling approach, and allows then a proper redesign of the atomizer, finally giving with light naphtha a completely satisfactory combustion behaviour, as illustrated in the last column of Figure 4. The final flame behaviour is similar to that of the fuel oil case, even if the light naphtha is a low boiling point fuel, and even if the evaporation pattern of the newly designed atomizer is completely different from the initial Y-jet atomizer used in the case-1 and case.2 simulations.

Conclusion

In this work an example is given of how CFD modelling and simulation can be exploited for the design and optimization of a heavy-duty burner. In particular, gaseous and liquid fuel operation are simulated, focusing on modelling and functioning aspects.

For the refinery gas feeding it is proven that the modelling of internal spud volumes and nozzle holes in not required for the considered geometry, and that the burner design is completely satisfactory as for its flame shape and coupling with the combustion chamber.

Concerning liquid fuel, the CFD simulations proves the inadequacy of a standard Y-jet air assisted atomizer to operate with low boiling point fuel like light naphtha. A second simulation with the same atomizer and fuel-oil is used to verify the modelling of liquid droplets and spray combustion. Finally, the numerical simulation are exploited to redesign the atomizer for low boiling point liquid fuel, obtaining a completely satisfactory behavior of flame and complete combustion at furnace exit.

References

[1] WEO-2021 World Energy Outlook, International Energy Agency, Paris, France, 2021.

[2] Park, J. K., Park, S., Kim, M., Ryu, C., Baek, S. H., Kim, Y. J., Kim, H. H., Park, H. Y., "CFD analysis of combustion characteristics for fuel switching to bioliquid in oil-fired power plant," Fuel, 159, 2015, 324-333.

[3] Liu, H., Zhang, L., Li ,Q., Zhua, H., Deng, L., Liu, Y., Che, D., "Effect of FGR position on the characteristics of combustion, emission and flue gas temperature deviation in a 1000 MW tower-type double-reheat boiler with deep-air-staging", Fuel, 246, 2019, 285-294.

[4] Laubscher, R., van der Merwe, S., "Heat transfer modelling of semi-suspension biomass fired industrial water tube boiler at full- and part-load using CFD", Thermal Science and Engineering Progress, 25, 2021, 100969.

[5] ANSYS Fluent, Users and Theory Guide, ANSYS, Inc., v.17.2, 2017.