



CFD simulations on marine burner flames

Cafaggi, Giovanni; Jensen, Peter Arendt; Glarborg, Peter; Clausen, Sønnik; Dam-Johansen, Kim

Publication date:
2017

Document Version
Peer reviewed version

[Link back to DTU Orbit](#)

Citation (APA):

Cafaggi, G., Jensen, P. A., Glarborg, P., Clausen, S., & Dam-Johansen, K. (2017). *CFD simulations on marine burner flames*. Paper presented at Nordic Flame Days, Stockholm, Sweden.

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

CFD simulations on marine burner flames

**Giovanni Cafaggi^{1*}, Peter Arendt Jensen¹, Peter Glarborg¹, Sønnik Clausen¹,
Kim Dam-Johansen¹**

*¹Technical University of Denmark, Department of Chemical and Biochemical Engineering, Søtofts
Plads, 2800-Kgs.Lyngby, Denmark*

** Corresponding author: gioc@kt.dtu.dk*

Background and aim

The marine industry is changing with new demands concerning high energy efficiency, fuel flexibility and lower emissions of NO_x and SO_x. A collaboration between the company Alfa Laval and Technical University of Denmark has been established to support the development of the next generation of marine burners. The resulting auxiliary boilers shall be compact and able to operate with different fuel types, while reducing NO_x emissions.

The specific boiler object of this study uses a swirl stabilized liquid fuel burner, with a pressure swirl spill-return atomizer (Fig.1). The combustion chamber is enclosed in a water jacket used for water heating and evaporation, and a convective heat exchanger at the furnace outlet super-heats the steam.

The purpose of the present study is to gather detailed knowledge about the influence of fuel spray conditions on marine utility boiler flames. The main goal of work presented in this paper was to obtain a spray description to setup a particle injection region in the CFD simulations of the boiler.

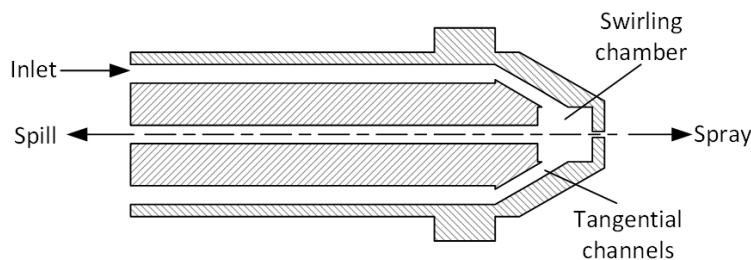


Figure 1. Spill-return atomizer cross-section.

CFD simulations

Experimental and computational methods have been used in the past for simulating both liquid fuel flames and the atomization processes [1] [2] [3]. Results obtained via Computational Fluid Dynamics (CFD) provide detailed information about the local temperatures, compositions and flow fields within the furnace chamber [4]. However, due to its stochastic [5] nature and to the very small length- and time-scales at which it takes place, to simulate the droplet formation process with reasonable accuracy, huge computational resources would be required. The commercial software ANSYS CFX has been used to do 3D CFD simulations of the boiler. As the main objective is to simulate the flame itself, the computational domain has been restricted

to the furnace, thus leaving out the stack and the convective heater. A grid independency study led to the use of a grid with approximately 12 million cells. During the simulation campaign, several models have been tested for turbulence, radiation and reaction chemistry. These modes have been chosen according to the ANSYS manuals [5]. For turbulence and radiation, the Shear Stress Transport model and the discrete radiation model proved to be adequate. Being the CO emissions of interest for the project, the combustion was simulated with a two-step global reaction (Reactions 1-2) [6] [7], while the NO_x formation can be calculated by post-processing of the results.



The Eddy Dissipation Model and Finite Rate Chemistry model are used as reaction models for both steps of the global reaction. The fuel is injected into the computational domain as droplets with a fixed velocity and a specified particle size distribution. As expected, a sensitivity analysis showed that the particle size distribution plays a crucial role in the flame stability, to the point of leading to flame lift off in some cases.

The nozzle manufacturers give some of the macroscopic characteristics of the spray, such as spray angle at maximum and minimum flow, design differential pressure through the nozzle, and inlet, return and spray flows as functions of the pressure in the return line. No documentation is given regarding droplet size and velocity distribution. It is possible to retrieve in literature [8] correlations to calculate these parameters for Simplex atomizers, but, depending on the correlation used, the range of results obtained is broad (meaning using one instead of another would drastically change the CFD results). It has therefore been decided to build a spray characterization setup to observe the atomization quality of the nozzle and gain a better understanding of the atomization process at different operating conditions and for different fluids.

Spray setup

A spray setup was built to reproduce the operating conditions of the burner (Fig.2). In order to deliver the desired flow, a three-piston reciprocating pump is used. To ensure a constant flow a membrane tank precharged with compressed air is used to dampen pulsations in the flow. The flow meters on the pump line and on the return line make possible to calculate the exact flow through the nozzle. The gauge positioned just before the nozzle measure the differential pressure through the nozzle. A needle valve is used to regulate the flow through the return line and, together with the pump, it gives full control of the flow through the nozzle. This means that the regulation of the pump and the needle valve also controls the differential pressure through the nozzle.

For safety, versatility and simplicity, the setup is built to use water and water-glycerol mixtures as working fluid instead of the actual fuel. Mixing glycerol with water is known procedure to reproduce the physical characteristics of fuels for spray analysis [9] [10]. A rheometer is used

to measure the mixture viscosity, even if plenty of data is found in literature, regarding water glycerol mixtures, including viscosity for different concentration at different temperatures. This is done both to ensure that the right viscosity is achieved, but also because it enable us to add surfactants in the mixtures, thus considering possible effects of surface tension on the spray.

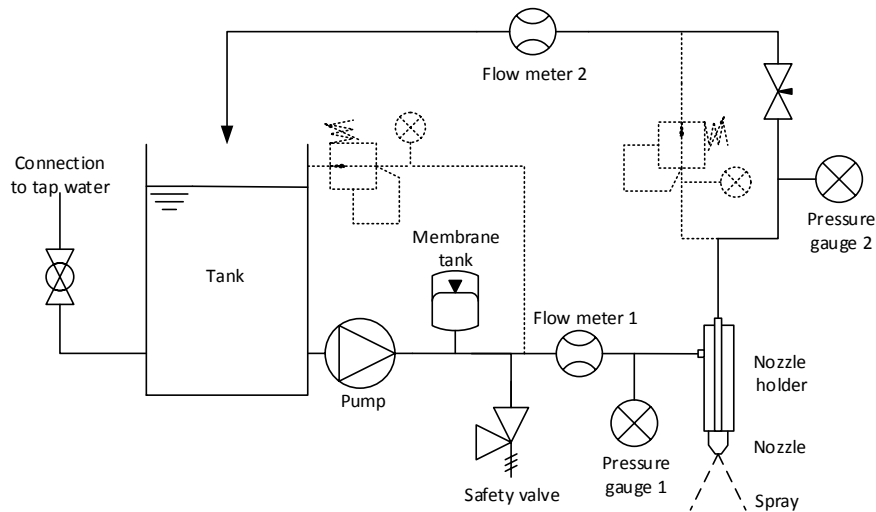


Figure 2. PI diagram of the spray setup: the dotted lines represent planned future additions.

The spray generated with this setup is then captured with an optical system and the pictures analysed with a tailor made software to obtain information about droplet size and velocity distributions.

Optical imaging setup

The optical imaging setup used is an adaptation of a system used to observe solid particles in erosion experiments [11]. A double exposure CCD camera, largely used for Particle Image Velocimetry (PIV), with a telecentric lens is employed to capture couples of frames of a spray region. The volume observed is determined by the lens magnification, camera sensor size and depth of focus. In the current setup, the volume is a rectangular cuboid of $2.41 \times 3.24 \times 0.86 \text{ mm}^3$. The pulsed LED light source is placed behind the focal point and sends two pulses with an adjustable delay as low as $0.9 \mu\text{s}$. The LED pulser is synchronized with the camera using a delay generator for accurate timing of signals. The result is two consecutive frames showing the same region after a known delay, thus making it relatively easy to calculate the individual droplets velocity. The images are then stored and analysed on a dedicated computer.

Results and conclusion

To prepare the CFD simulations a number of different models, parameters and algorithms has been tested. While not universal and limited to the software used, the obtained combination can be considered as a starting point for future modelling. During the CFD study insights in the boiler operation has been obtained: fuel conversion (Fig.3b), vortex shedding frequencies in different parts of the geometry and positions of the recirculation zones have been determined (Fig.3a). In addition, a sensitivity analysis showed that under the same conditions, a change from 10 to $25 \mu\text{m}$ of droplet mean diameter, leads to flame lift off.

Using the spray setup described, it was possible to measure droplet size in different regions of the spray. The data thus obtained can now be used to improve the CFD simulation of the boiler. Some of the results gathered from spraying water, at the same pressure and flow of the burner at full load, are shown in Fig.5.

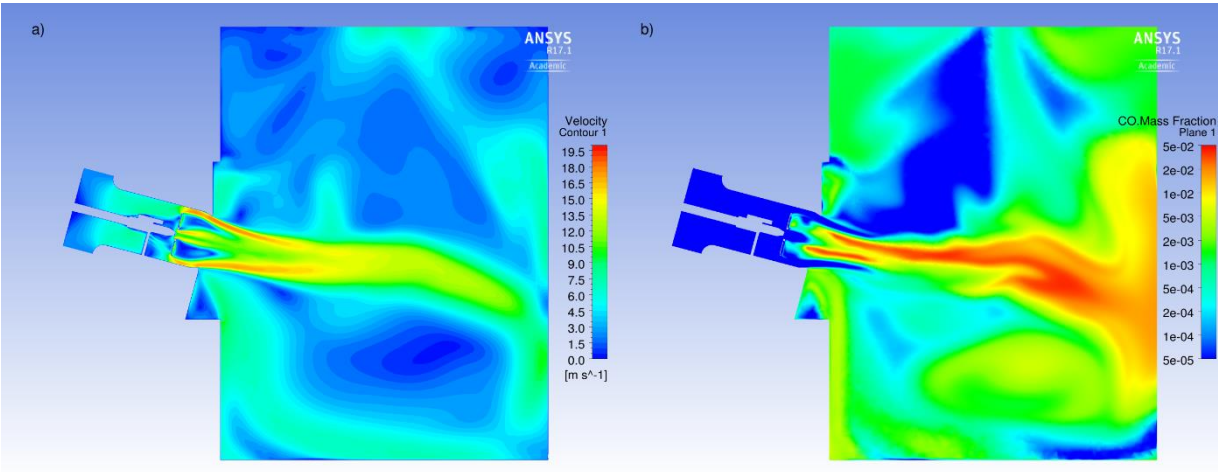


Figure 3. a) Velocity [m/s] and b) CO [%] contours on the central vertical cross section of the boiler.

Also an interesting result of the spray characterization is the strong dependence of velocity magnitude and angle on the droplet size. This will have to be taken into account when setting a particle injection region in the CFD simulations. Another item of relevance for setting up future CFD simulations is the direct observation of spray regimes: from the pictures taken it is easily seen where the liquid still forms complex structures and where instead only droplets are present, and therefore can be modelled as a particle injection.

Lastly, the spray setup gave us the possibility to directly observe the droplet formation mechanisms with a relatively inexpensive equipment (Fig.4). This could prove to be an interesting opportunity for further studies on spray characterization.

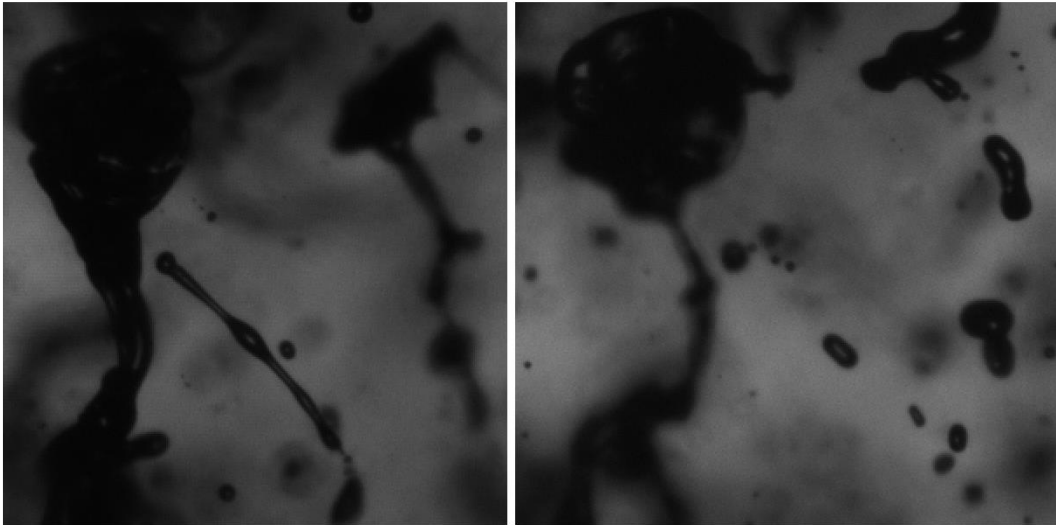


Figure 4. Detail of an image pair taken at an axial distance of 12 mm at 5 mm radius (directly on the spray cone) from the nozzle orifice with a 40 μs delay.

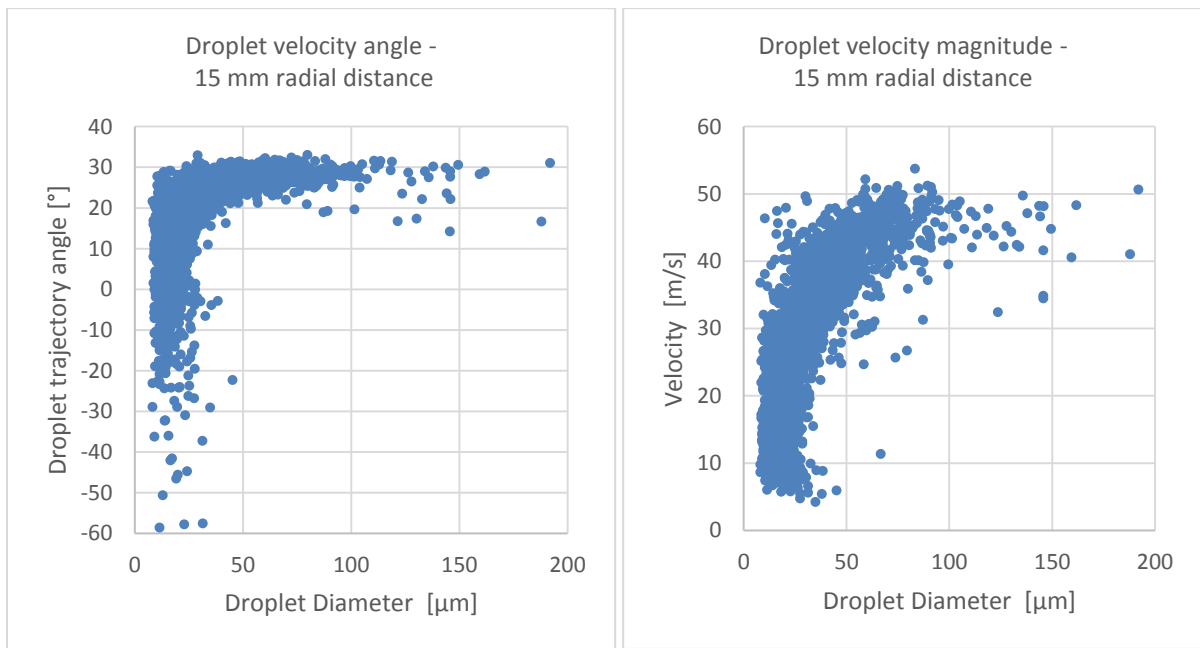
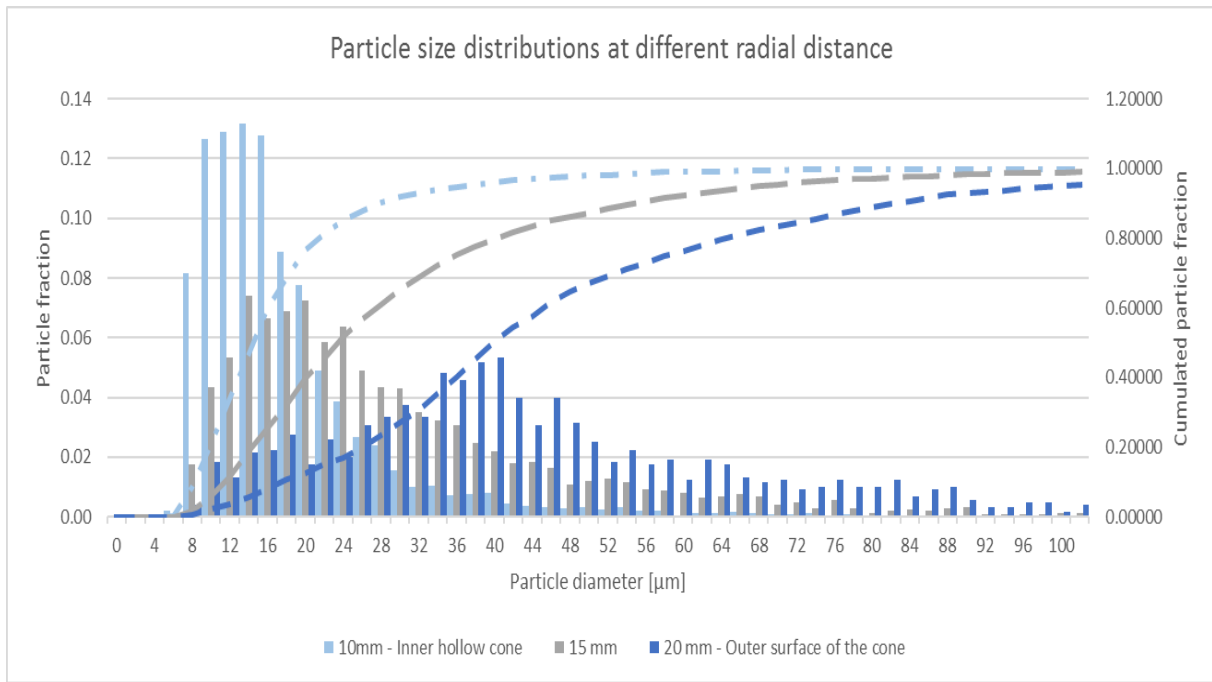


Figure 5. Data from a sample of 6628 droplets, 24 mm downstream of the nozzle orifice. The upper plot shows the particle size distribution based on the number of droplets at three different radial distance from the cone axis. Below the data for 15 mm radius is shown in detail.

References

- [1] P. Jenny, D. Roekaerts och N. Beishuizen, "Modeling of turbulent dilute spray combustion," *Progress in Energy and Combustion Science*, nr 38, pp. 846-887, 2012.

- [2] M. Linne, "Imaging in the optically dense regions of a spray: A review of developing techniques," *Progress in Energy and Combustion Sciences*, nr 39, pp. 403-440, 2013.
- [3] Z. Ling, X. Zeng, T. Ren och H. Xu, "Establishing a low NO_x and high-burnout performance in a large-scale, deep-air-staging laboratory furnace fired by a heavy-oil swirl burner," *Applied Thermal Engineering*, nr 79, pp. 117-123, 2015.
- [4] I. Bonefačić, W. Igor och P. Blecich, "Improvement of fuel oil spray combustion inside a 7 MW industrial furnace: A numerical study," *Applied Thermal Engineering*, nr 110, pp. 795-804, 2017.
- [5] ANSYS CFX User's Manual, 2017.
- [6] C. K. Westbrook och F. L. Dryer, "Simplified Reaction Mechanisms for the Oxidation of Hydrocarbon Fuels in Flames," *Combustion Science and Technology*, vol. 27, pp. 31-43, 1981.
- [7] B. Franzelli, E. Riber, M. Sanjosé och T. Poinso, "A two-step chemical scheme for kerosene-air premixed flames," *Combustion and Flame*, vol. 157, nr 7, pp. 1364-1373, 2010.
- [8] A. H. Lefebvre, *Atomization and Sprays*, Hemisphere publishing co., p. 214, 1989.
- [9] A. Sängler, T. Jakobs, N. Djordjevic och T. Kolb, "Effect of primary instability of a high viscous liquid jet on the spray quality generated by a twin-fluid atomizer," i *ILASS – Europe 2014, 26th Annual Conference on Liquid Atomization and Spray Systems, 8-10 Sep. 2014*, Bremen, Germany, 2014.
- [10] A. Davanlou, J. D. Lee, S. Basu och R. Kumar, "Effect of viscosity and surface tension on breakup and coalescence of bicomponent sprays," *Chemical Engineering Science*, nr 131, pp. 243-255, 2015.
- [11] F. Cernuschi, C. Rothleitner, S. Clausen, U. Neuschaefer-Rube, J. Illemann, L. Lorenzoni, C. Guardamagna och H. Engelbrecht Larsen, "Accurate particle speed prediction by improved particle speed measurement and 3-dimensional particle size and shape characterization technique," *Powder Technology*, nr 318, p. 95–109, 2017.
- [12] J. K. Park, S. Park, M. Kim, C. Ryu, S. H. Baek, Y. J. Kim, H. H. Kim och H. Y. Park, "CFD analysis of combustion characteristics for fuel switching to bioliquid in oil-fired power plant," *Fuel*, pp. 324-333, 2015.