CFD Modeling and Validation For Once-Through Steam Generator Using OpenFOAM

E. Turco Neto[1], E. Askari Mahvelati[1], M. Forcinito[1], A. Chatel[2], L. Fitschy[2], A. Maesen*,[2]

[1] AP Dynamics, Calgary, Canada
[2] GDTech S.A., Alleur, Belgium

Abstract

In this paper, a small scale Once-Through Steam Generators (OTSG) is studied using the open-source CFD code OpenFOAM. The capability of the code to simulate the turbulent non-premixed combustion and the underlying heat transfer phenomena is assessed. The predicted heat fluxes to the water tubes are compared to field and Fluent data, in which a good agreement was obtained. The results presented in this paper prove that computational fluid dynamics (CFD) models can be a powerful tool available to assist in designing OTSGs.

Introduction

In Northern America, the Steam-Assisted Gravity Drainage (SAGD) oil extraction technology strongly relies on the use of water-tube boilers called OTSGs. Due to economical and environmental concerns, the pollutant emissions resulted from the operation of these boilers must be minimized while maintaining their high thermal efficiency. Typical OTSGs are composed of three main parts, namely the burner, the radiant, and the convective sections. The burner stabilizes a strongly turbulent flame produced by the combustion of natural gas and air. The heat generated by the combustion reactions is then transferred to the water tubes, which are distributed over the radiant and convection sections. Finally, the steam produced inside these pipes is injected into the ground to reduce the bitumen viscosity, and allowing its pumping to the surface.

Up to now, the design of OTSGs has been governed by simplified models as well as knowledge obtained from field experience in industry. However, numerical simulations are nowadays an additional tool that can bring powerful guidance in the design phase of OTSGs. In this context, CFD has become a flexible tool to study the complex underlying physics governing their operation. In other words, it is capable to provide accurate predictions of the flow and thermal behavior taking place inside the equipment, while drastically reducing the costs and time required to build and run experimental pilot-scaled units.

In this work, the capability of simulating the operation of a small scale OTSG with the open-source CFD code OpenFOAM is assessed. In order to do so, the work is organized as follows. The geometry and computational mesh of the considered OTSG are first presented. Then, the boundary conditions and the sub-models used in the simulation are listed. Finally, the results for the velocity and temperature profiles are presented. The validation of the model is made by comparing the computed heat fluxes for each tube section with corresponding values taken from available field data, and previous results obtained from running the model using the commercial CFD software Fluent [1].

CFD Modeling

Geometry and Boundary Conditions

The Once-Through Steam Generator (OTSG) shown in Figure 1 was considered as a test-case.

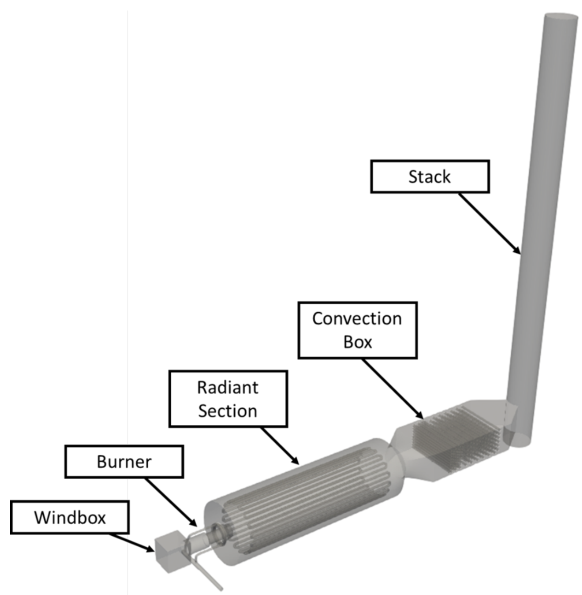

Figure 1: Once-Through Steam Generator.

This unit is equipped with a windbox, a burner, a radiant section, a convection box, and a stack. In order to carry out the combustion, natural gas is fed into the unit and enters the combustion chamber through two different types of nozzle tips. These tips are distributed over a gas ring in a staged "free-jet" type burner, which has the purpose to minimize the NOx generation. The burner geometry is represented in Figure 2.

*Corresponding author: arthur.maesen@gdtech.eu
Proceedings of the European Combustion Meeting 2021

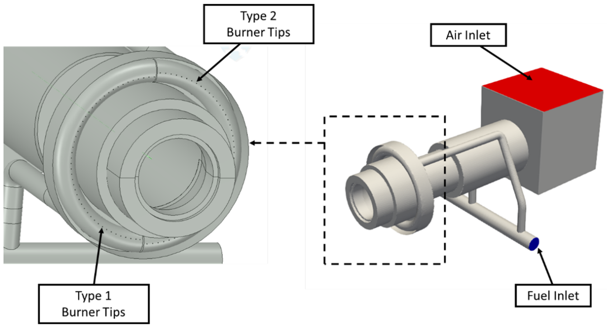

Figure 2: Burner geometry and configuration.

The type 1 burner tips are designed to project the fuel gas slightly towards the centerline of the burner, which creates jets that brush the edges of burner tile. Consequently, cold spots are formed at its surface, which assists in modulating the flame temperature to reduce NOx generation. On the other hand, the type 2 tips are positioned to direct the jets slightly outwards relative to the centerline. Consequently, the jets produced by these tips induce the formation of recirculation zones around the flame due to the additional turbulence. In this way, the local oxygen concentration is reduced due to the dilution imposed by the enhanced turbulent mixing, and consequently, the chemical reactions generating NOx species slow down. At the same time, the air stream is fed into unit by a windbox attached to the burner, whose flow rate is adjusted to maintain 15% excess of air.

Once combustion is initiated, the produced hot flue gases flow over several tubes distributed at both radiant and convection sections. The configuration of each set of tubes is shown in both Figures 3 and 4.

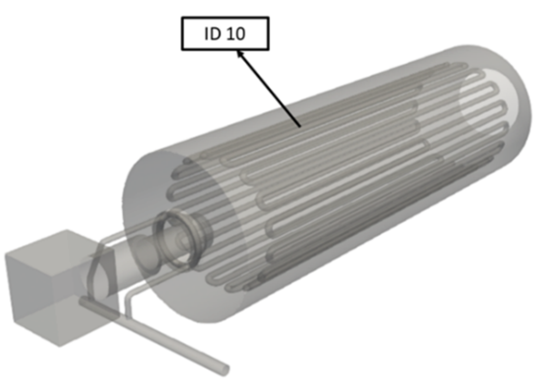

Figure 3: Steam Tubes Configuration in the radiant section.

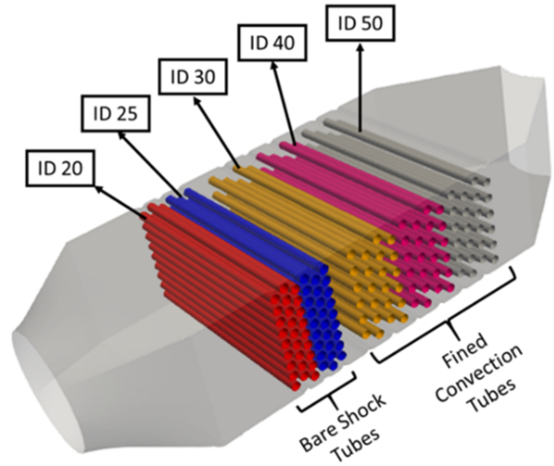

Figure 4: Steam Tubes Configuration in the convective section.

The boiler feed water enters the finned convection tubes at the inlet of ID 50, where it is fed in counterflow configuration relative to the flow direction of the flue gases. As a first assumption, the fins were not included in the computational domain in order to minimize the mesh size and reduce the computational time. At IDs 50, 40, and 30, the heat absorbed by the water is mainly due to convective heat transfer with the objective to enhance the heat recovery in the system. On the other hand, the water flow direction is changed to co-flow configuration at the bare shock tubes (ID's 20 and 25), where significant heat transfer rates are achieved in this section due to the high temperature of the flue gas. Consequently, water reaches its boiling point at this set of tubes, and evaporation starts. Finally, the boiling water enters the set of tubes in ID 10 located inside the radiant section, where the evaporation rate is significantly increased due to the high rate of radiative heat transfer emitted from flame.

Table 1 provides the nominal operating conditions used for running this unit, which are further used to properly set the boundary conditions for the CFD simulation. The thermal boundary conditions were set based on pre-defined values of heat transfer coefficient and bulk temperature.

Firing rate [MW]	1.7126
Air excess [%]	15
Product Steam Quality [wt %]	80

Table 1: Nominal operating conditions.

Meshing

The computational mesh was generated using a hexa-dominant grid with approximately 36 million cells. Local refinement was applied at regions near the burner tips to resolve the flow around the jets. Furthermore, additional refinement was applied around the burner tile in order to properly capture the flame characteristics, and accurately compute the reactions rates. Finally, fine cells were also placed around the tubes to ensure numerical stability and accurate computation of the heat fluxes. Representations of the mesh are sketched in Figures 5 to 8.

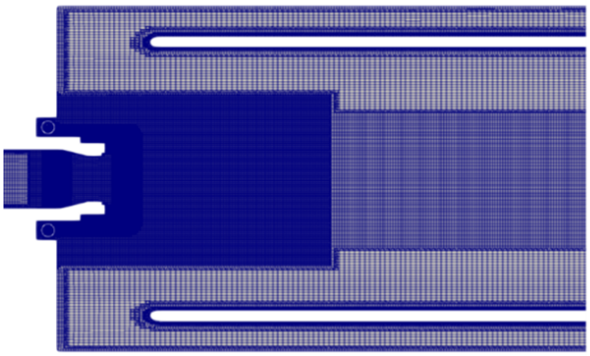

Figure 5: Computational Grid Generated for the Pilot-Scale OTSG - closeup in the radiant section.

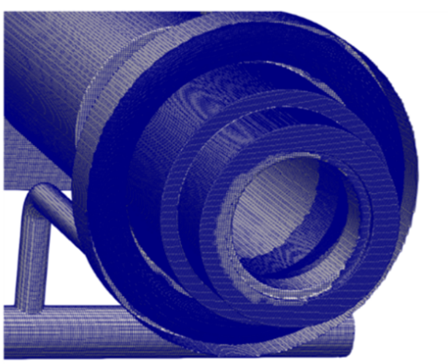

Figure 6: Computational Grid Generated for the Pilot-Scale OTSG - burner.

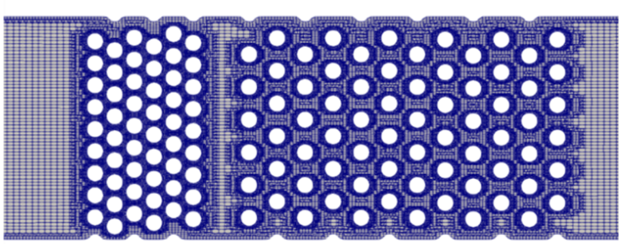

Figure 7: Computational Grid Generated for the Pilot-Scale OTSG - tubes in the convection section.

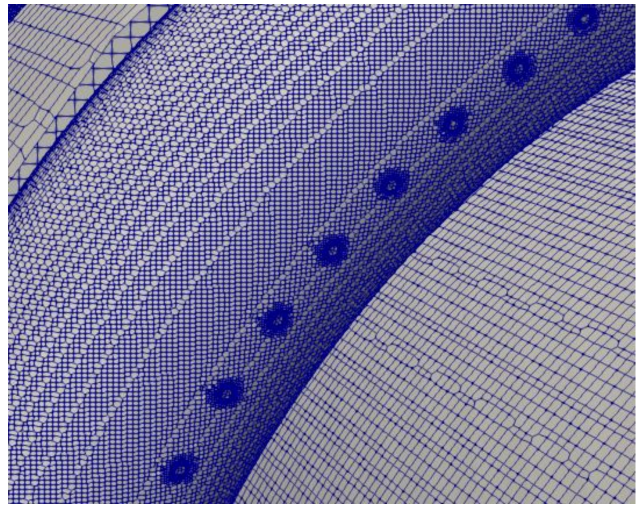

Figure 8: Computational Grid Generated for the Pilot-Scale OTSG - closeup on the injection nozzles.

Model Setup

The simulation presented in this section was conducted in OpenFOAM version 8 due to its flexibility in terms of combustion modeling and parallel computing. Moreover, the solver "simpleReactingFoam" was implemented and used to obtain the steady-state temperature, velocity, and species concentration profiles.

The fuel gas is assumed to be pure methane, whose combustion follows a one-step global reaction mechanism represented by Equation 1.

$$\mathrm{CH_4} + 20_2 \longrightarrow \mathrm{CO_2} + 2\mathrm{H_2}0 \tag{1}$$

Since the fuel and air streams are injected at high speed inside the radiant section, it is reasonable to assume that the turbulent mixing is the rate-controlling step, and thereby, the eddy-dissipation model (EDM) [2] was chosen to compute the reaction rates.

The SST k-ω turbulence model [3] was used to compute the turbulent effects in the mean flow quantities, since it accurately captures recirculation zones. Also, the discrete ordinate method was selected as the radiation model as the accuracy of the estimated radiative heat fluxes might depend on the directionality of the radiation intensity emitted from the flame [4, 5].

Results and discussion

Figures 9 and 10 show the steady-state velocity and temperature profiles obtained from the OTSG simulation.

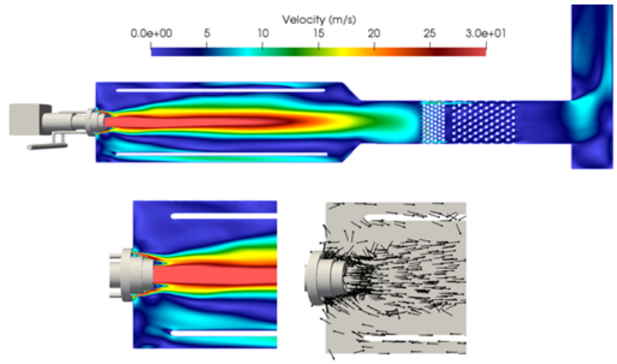

Figure 9: Velocity profile.

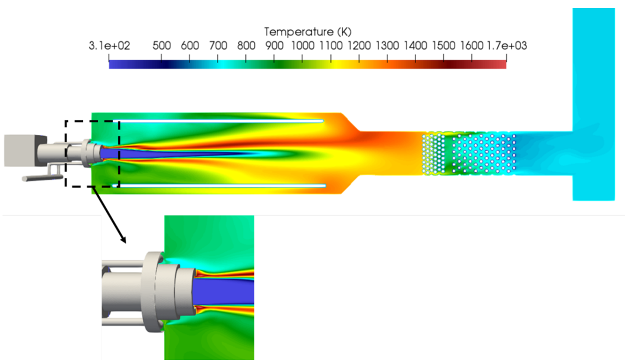

Figure 10: Temperature profile.

It can be observed that the simulations capture the brushing effect and flow patterns created by the two types of burner tips. In this case, a recirculation zone is formed around the flame, which brings cold fluid towards the hot temperature zone and assists in modulating the temperature to potentially reduce the NOx generation. These profiles also show that the flame forms over the surface of the burner tile, where both fuel gas and air come into contact. Additionally, hot spots are formed around the tubes located close to the entrance of the convection box.

Under these conditions, it is possible that operating and safety issues might arise with the current design of this unit. For example, the lifetime of the burner tile might be reduced due to the constant exposure to the high flame temperature. Moreover, the hot spots formed around the tubes can lead to thermal cracks over time as flame impingement is likely occurring at those locations. These results show that the CFD simulation has the capability to unveil potential design issues before commissioning of the unit.

Validation of the model is required to verify whether the model properly captures the underlying physics controlling the combustion process. Table 2 shows a comparison among the field and simulated values of heat fluxes absorbed at each tube ID in the convection box.

Tube ID	Field Data	Fluent	OpenFOAM
10	58.96	47.58	48.43
20	52.96	51.85	50.77
25	30.57	24.5	24.24
30	54.05	20.85	20.61
40	29.78	17.79	15.62
50	13.31	16.19	12.06

Table 2: Comparison of the heat fluxes obtained at each tube ID yielded by Fluent and OpenFOAM with the field data [kW/m^2].

From Table 2, it can be observed that a good agreement between the OpenFOAM results with those of Fluent and field data for ID's 10, 20, 25 and 50. However, both Fluent and OpenFOAM underestimate the fluxes at ID's 30, 40 compared to the field data. This can be explained by the fact that the fins were not taken into account, and consequently, less heat is absorbed at those sets of tubes.

The temperature of the flue gas leaving the radiant section was also compared with field and Fluent data. Table 3 summarizes the results.

Field Data	Fluent	OpenFOAM
1417.15	1485.33	1296.61

Table 3: Comparison of the mean flue gas temperature leaving the radiant section predicted by OpenFOAM with the field and Fluent data [K].

The deviations observed in Table 3 can be explained due the fact that a five-step reaction mechanism was used in Fluent, while the one-step global reaction was applied in the present OpenFOAM model. In this context, detailed mechanisms involve more reactions that can affect the temperature distribution. However, the values presented are still in good agreement.

Conclusions

In this work, the performance of a once-through steam generator has been investigated using computational fluid dynamics. First, the computational domain was created to capture the main geometric features of burner, radiant and convection sections, and steam tubes. Then, a hexa-dominant computation grid was generated with local refinement regions to properly resolve the main flow and flame characteristics. The combustion reaction is assumed to take place based on a one-step reaction mechanism, in which the rate-controlling step is governed by the turbulent mixing of reactants. The steady-state heat fluxes for each tube section were compared with values taken from available field data and previous Fluent simulation results, in which good agreement was observed. Finally, the mean temperature of the flue gas leaving the radiant section was also compared. Although, the results were comparable, small discrepancies were observed due to the different reaction mechanism applied in this work.

The analysis and results presented in this work show that CFD modeling has a good potential to be applied to simulate the operation of OTSGs. However, the overall accuracy of the model can still be affected by the discrepancies related to sub-models selected to capture the effects of turbulence-chemistry interaction and radiation. In future work, the impact of the selected reaction mechanism is going to be further studied. Additionally, a NOx solver is going to be implemented to predict the NOx emissions from this unit.

Acknowledgements

We would like to thank the International research activities by Small and Medium-sized Enterprises (IraSME) for the financial support provided to conduct this research.

References

[1] E. Askari, E. Turco Neto, R. Lozowy, A. Chatel, and M. Forcinito. 3d cfd model development and validation for once-through steam generator (otsg): Coupling combustion, heat transfer, and steam generation. In *Proceedings of the 2020 American Flame Research Council (AFRC) Symposium*, 2020.

[2] B F Magnussen and Bjørn Hjertager. On mathematically modeling with special emphasis on soot formation and combustion. *Symposium (International) on Combustion*, 16:719–728, aug 1977.

[3] Florian Menter, M Kuntz, and R B Langtry. Ten years of industrial experience with the SST turbulence model. *Heat and Mass Transfer*, 4, jan 2003.

[4] Ali Hussain Kadar. *Modeling Turbulent Non-Premixed Combustion in Industrial Furnaces*. PhD thesis, Delft University of Technology, 2015.

[5] H K Versteeg and W Malalasekera. *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. Pearson Education Limited, 2007.