3D CFD Model Development and Validation for Once-Through Steam Generator (OTSG): Coupling Combustion, Heat Transfer and Steam Generation

Ehsan Askari, Sr. CFD Analyst, AP Dynamics, Calgary, AB

Eugenio Turco Neto, CFD Analyst, AP Dynamics, Calgary, AB

Richard Lozowy, Acoustic and Fluid Analyst, AP Dynamics, Calgary, AB

Adrian Chatel, CFD Analyst, GDTech, Liege, Belgium

Mario Forcinito*, Chief Technology Officer, AP Dynamics, Calgary, AB
mario.forcinito@ap-dynamics.net

Abstract:

The current research studies the coupled combustion inside the furnace and the steam generation inside the radiant and convection tubes through a typical Once-Through Steam Generator (OTSG). A 3D CFD model coupling combustion and two-phase flow was developed to model the entire system of OTSG. Once the combustion simulation was converged, the results were compared to field data showing a convincing agreement. The CFD analysis provides the detailed flow behaviour inside the combustion chamber and stack, the flow of steam inside the tubes in the radiant section and the two-phase flow steam generation process in the radiant and convective section. Flame shape and orientation, velocity, species and temperature distribution at various parts of the furnace as well as steam generation and steam distribution inside the pipes were correctly predicted.

Keywords: Combustion, Steam Generation, Once-Through Steam Generator (OTSG), Computational Fluid Dynamics (CFD)

Introduction

The oil extraction industry in Northern Alberta relies on the use of large Once Through Steam Generators (OTSGs). Efficient steam generation with as low as possible pollutant emissions is a high priority for economic and environmental reasons. OTSGs are commonly used in Steam-Assisted Gravity Drainage (SAGD) operations which is an advanced oil recovery technology for heavy crude oil and bitumen production. Generally speaking, OTSGs are usually composed of three main parts: the burner, the radiant section and the convective section. Natural gas is burned through staged diffusive flames stabilized by the burner. The heat generated by the combustion is transferred to the water flowing through the piping system in the radiant and convective sections. The steam produced within the pipes is then injected underground to reduce the oil viscosity and allow its being pumped to surface.

Until now the design and operation of large OTSGs have been relied on somewhat simplified models and calculations backed heavily by experience developed over the years on evolving design of increasing size and complexity, in our opinion, this approach has reached its limits. The availability of powerful numerical simulation techniques will certainly be of benefit to the field and, as it has proved in other industries (Charles and Baukal, (2003)). The progress towards increasing efficiency and reliability of the equipment can be achieved only through the use of more advanced numerical simulations (Charles and Baukal, (2012)).

Simulation and computer assisted engineering services are not sufficiently developed for satisfying the current needs of producers. Improvements in the modelling methodologies to make them faster, more reliable and efficient for solving large problems are necessary to establish the producers to adopt and deploy these tools widely. The development of such tools and approaches gives a competitive advantage in the marketplace, and will help producers meet production, financial and HSE goals on a more stringently regulated industry.

Computational Fluid Dynamics (CFD) has become an accepted tool to help in the design and operation of oil and gas industry equipment. More recently, CFD has also found increasing application in the analysis of combustion equipment, such as industrial burners. In particular, CFD models are valuable assets for OTSG designers to study the efficiency in steam generation (Charles and Baukal, (2012)).

In comparison with empirical techniques, CFD would substantially reduce the total cost and processing time. However, CFD model development for such a complex equipment is challenging and requires deep understanding of physics as it simultaneously deals with the fields of combustion, heat transfer and phase change. to perform the CFD analysis in a timeframe compatible with the design and engineering process on such large and complex models, powerful computers are an indispensable requirement.

Singha and Forcinito (2018) proposed a methodology to reduce emission from a staged combustion burner in a typical OTSG. They produced a characterization map of the combustion system which was useful as a guideline towards efficient optimization of the OTSG during a field testing. Singha and Forcinito (2018) reached the conclusion that the CFD can minimize the number of experiments you need to characterize a burner and in a particular case they succeeded in eliminating Flue-Gas Recirculation (FGR).

The present work outlines a three-dimensional (3D) full CFD model development of a pilot OTSG. Initially combustion CFD simulation was performed to predict flow and temperature field and following that a two-phase CFD model with phase change was carried out in attempt to follow the coupled CFD model. The present research intends to serve as a typical case and aims to provide the detailed flow behaviour inside the combustion chamber and stack, the flow of steam inside the tubes in the radiant section and the two-phase flow steam generation process in the radiant and convective section.

Problem Description

The OTSG which is chosen in this research is in the pilot scale. The schematic of the OTSG is shown in Figure 1. In this figure, the burner is at the left, where the fuel and air are mixed, reacted and combustion takes place. The first section of the OTSG is called radiant section, since the primary mechanism of heat transfer is radiation at this location. After the radiant section, the flue gas enters the second part of OTSG (convection section), where heat is transferred primarily through convection.

The OTSG operations specs aim to achieve wet steam of 80-90% steam quality (by mass). For this purpose, Boiler Feed Water (BFW) enters the horizontal convection section of the OTSG where heat is exchange from the flue gases from the burner located in the radiant section to the water running through the tubes. As illustrated in Figure 2., the BFW travels through a series of the finned tubes (Second Tubes in Fig. 2) prior to exiting the OTSG and jumping to the first row of smooth tubes (First Tubes in Fig. 2) closest to the OTSG flame. The BFW then run in co-current flow with the flue gases until it reaches the last row of smooth tubes and exits the convection section. The BFW then enters the radiant tube section located closest to the burner where the radiant heat from the burner flame further heats the water until the desired outlet steam quality is achieved.

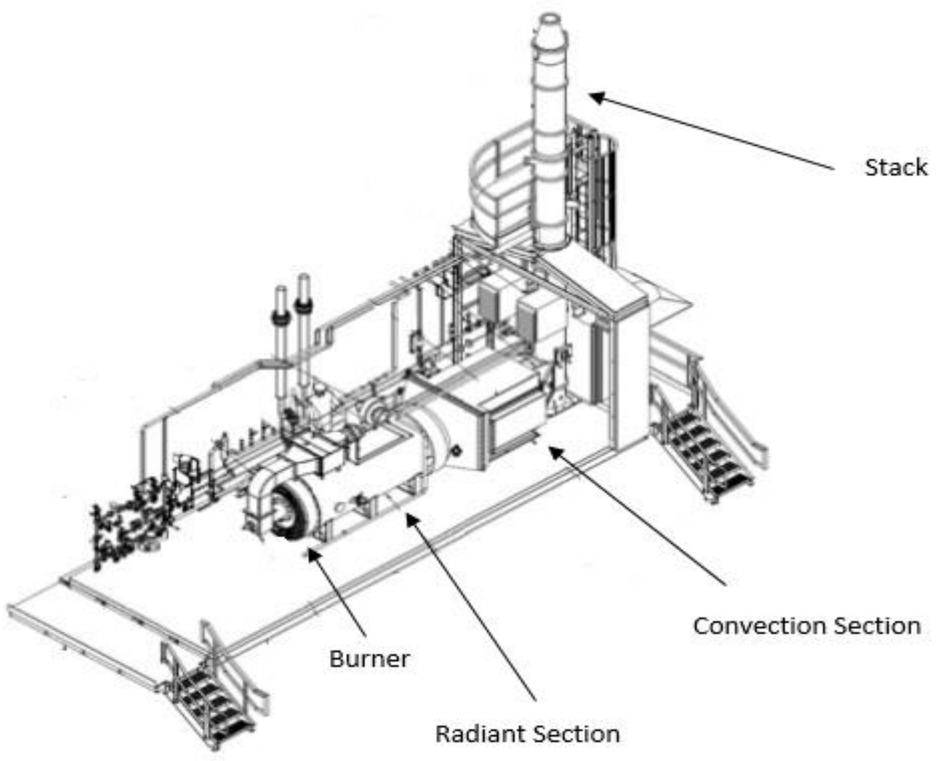


Figure 1: Schematic of the proposed OTSG showing corresponding sections

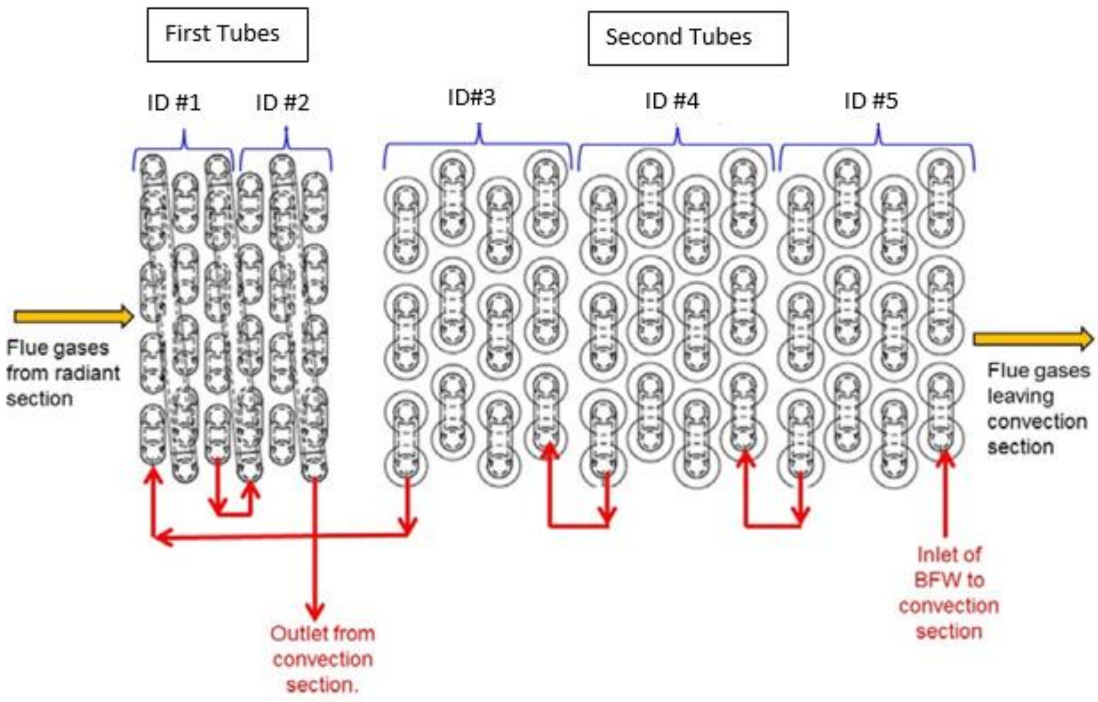


Figure 2: Tubes and flow arrangement in convection section

Table 1 summarizes the key characteristic parameters in the OTSG which is modelled in the current work.

Table 1: The OTSG Characteristics

Item	description
Size	Pilot Scale
Firing Rate	1.7 MW
Burner Type	Free-Jet
Target Steam Quality	80%

Mathematical Model

The present simulation was performed using commercial software ANSYS Fluent, using three-dimensional Reynolds Averaged Navier-Stokes (RANS) equations. The underlying physics related to turbulence, energy and species transport, radiation, two-phase flow, and boiling are also incorporated in the model.

Computational Fluid Dynamics Modelling

The following assumptions were adopted to create the CAD model:

1. The structural members (beams, etc.) inside the furnace were not modeled. They do not take part into the fluid flow simulation, and the effect of their presence is assumed to be negligible.
2. The inlet boundaries of the simulation are the air inlet duct to the windbox and the fuel pipe inlet. It is assumed that the flow is steady and homogeneous.
3. The burner geometry consists of the two burner tips. One half circle is type 1 burner tips and another half circle is the type 2 burner tips according to the burner manufacture drawings. The type 1 burner tips direct the fuel jet inward while the type 2 burner tips jet the gas parallel to centerline or slightly outward.
4. The outlet of the simulation was cut off after the end point of stack where the flue gas exits from the stack. Atmospheric pressure was applied at stack outlet.

5. All furnace surfaces were assumed to be adiabatic with a specific radiant emissivity.

Figure 3 to Figure 8 demonstrate the CAD model used in the OTSG CFD model development.

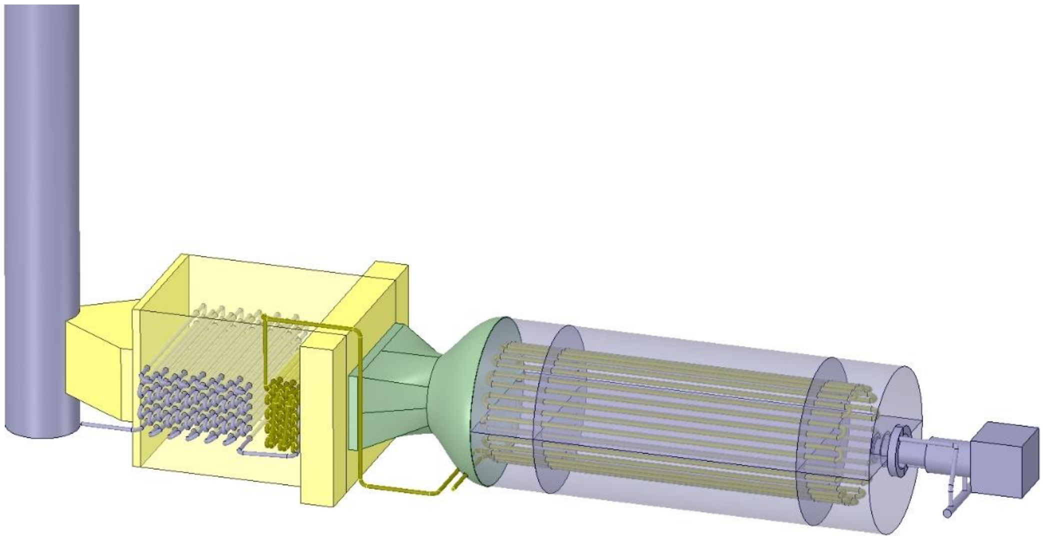


Figure 3: OTSG CAD Model including Fire Side and Water Side

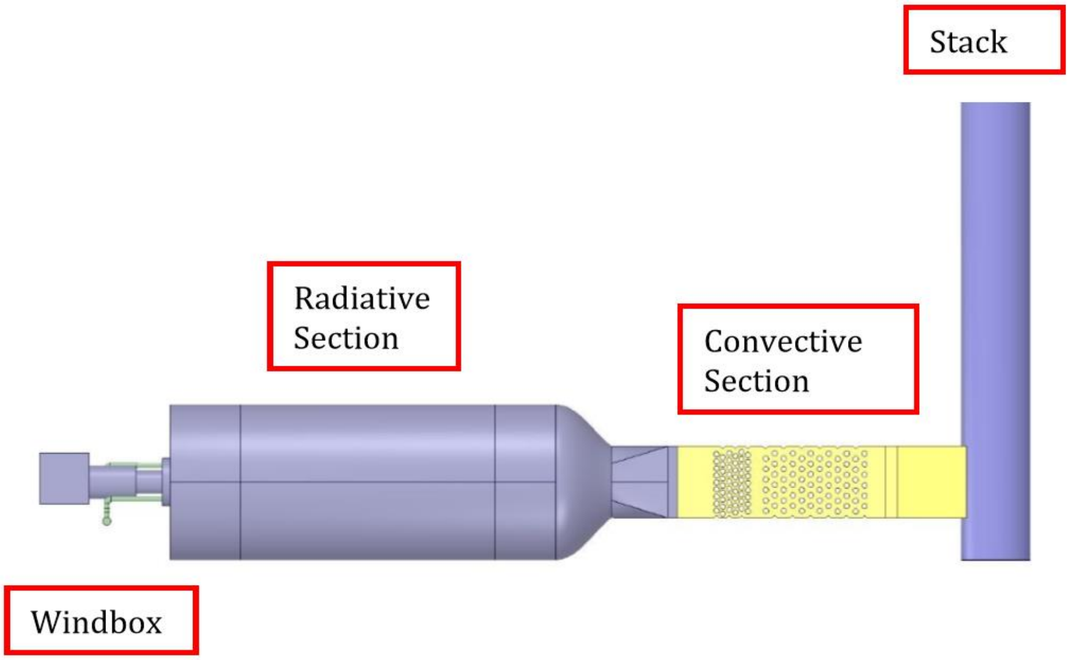


Figure 4: OTSG CAD Model for Fire Side

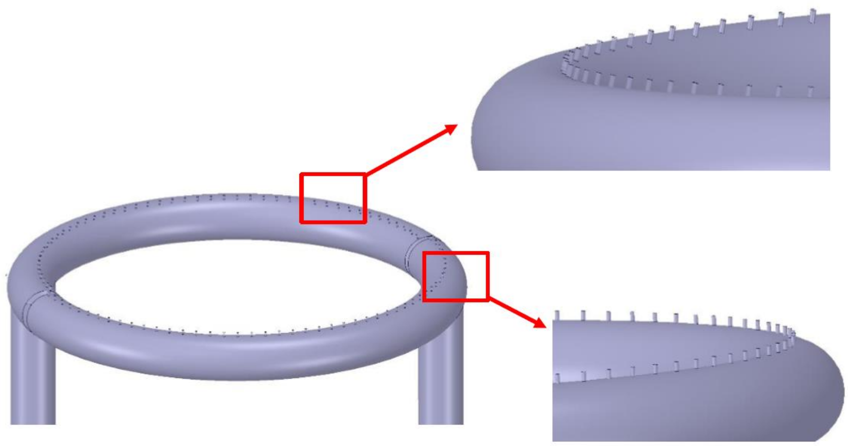


Figure 5: Burner solid model (Inset shows the close-up of the burner tips: (top=type 1, bottom=type 2)

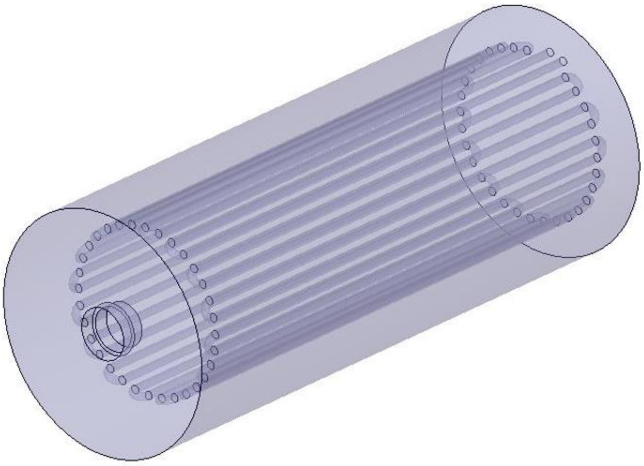


Figure 6: Radiant Section of OTSG – solid model [Transparent mode is shown to visualize the steam tubes inside the furnace]

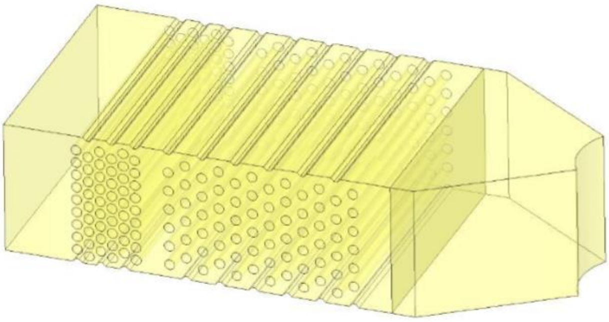


Figure 7: Convective Section of OTSG – solid model

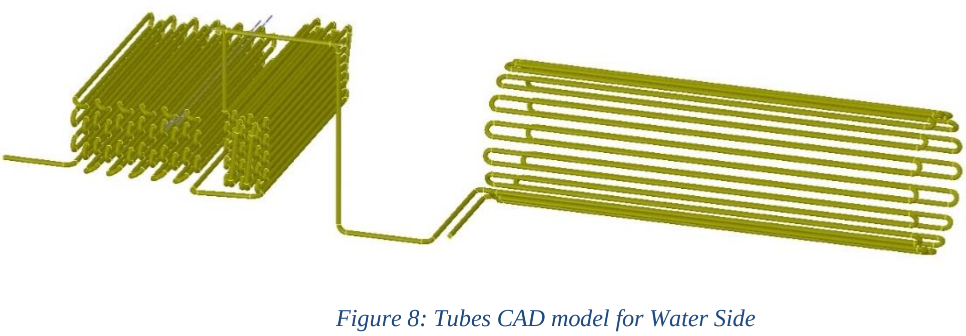


Figure 8: Tubes CAD model for Water Side

The discretization of the geometry utilized high order elements. The resulting mesh cell count is almost 40 million in the combustion side. The meshed geometry is shown in Figure 9 to Figure 14. The whole geometry was broken into two general assemblies for fire side and three assemblies for water side and mesh was generated separately for each part. The Burner-Radiant section consists of five parts including windbox, fuel gas piping (gas ring and burner tips), upper part, middle part and the lower part of radiant section. The interface between various meshes were handled through the creation of interpolating regions, which transfer the fluxes between non-conformal faces through conditional averaging of the over-lapping mesh faces.

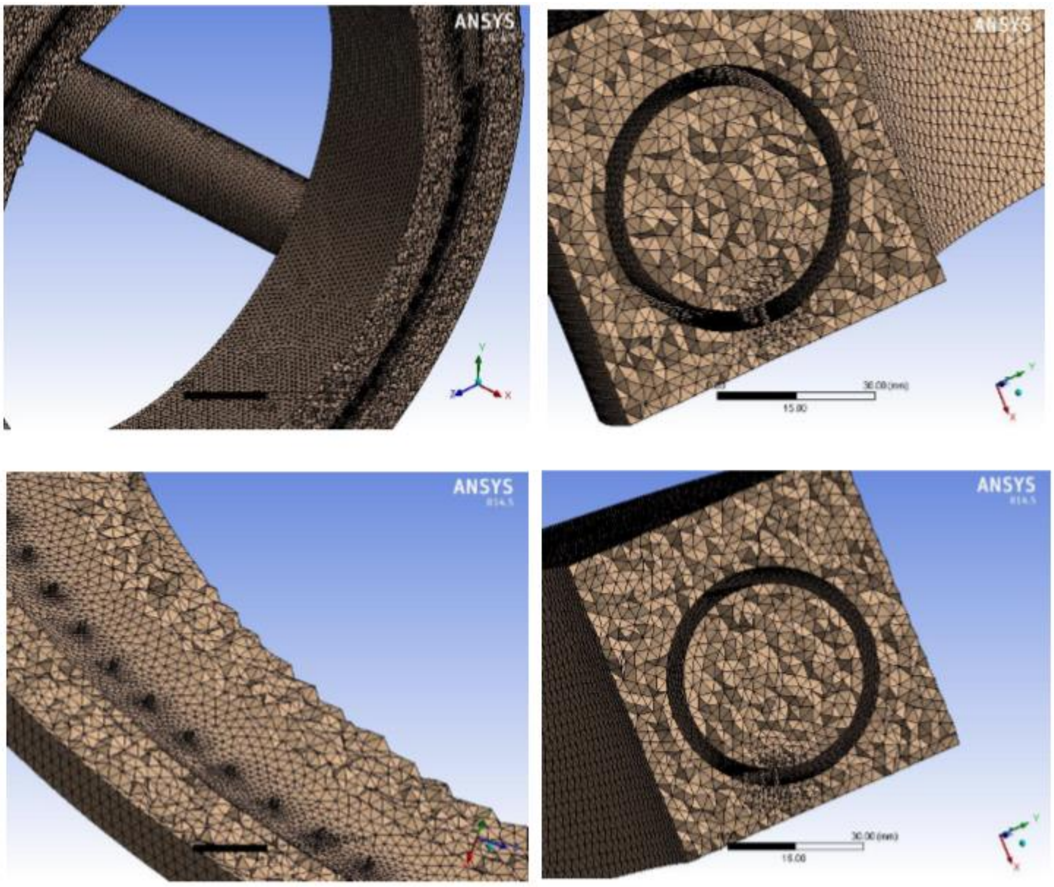


Figure 9: Mesh-Close up of the interior mesh of fuel gas ring showing the burner tips

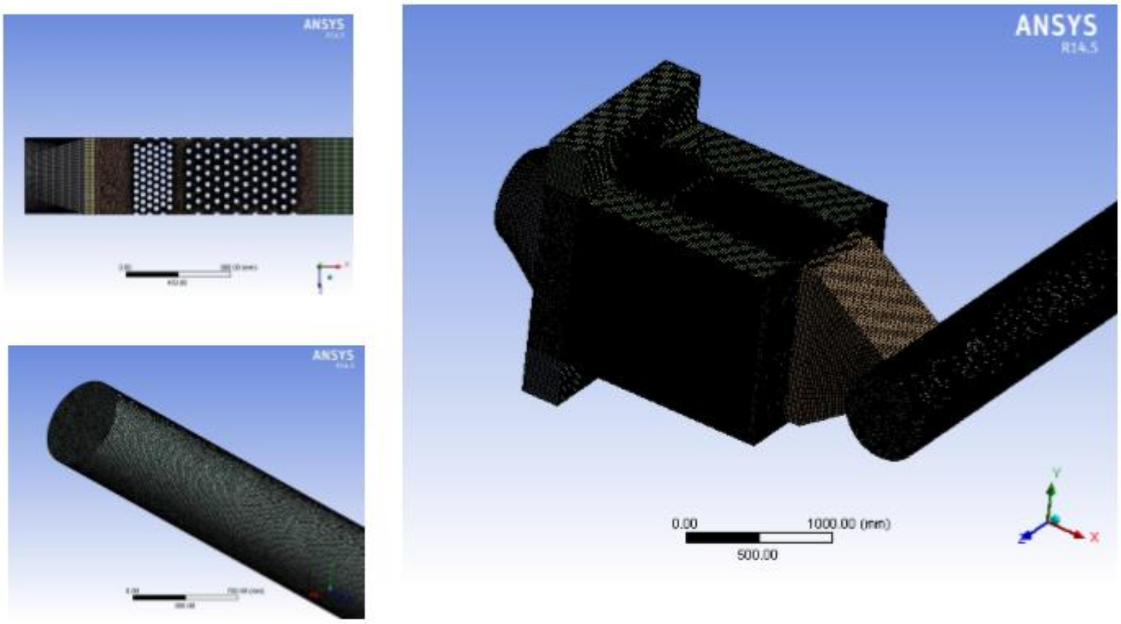


Figure 10: Fire Side Mesh Grid for Convection-Stack Section

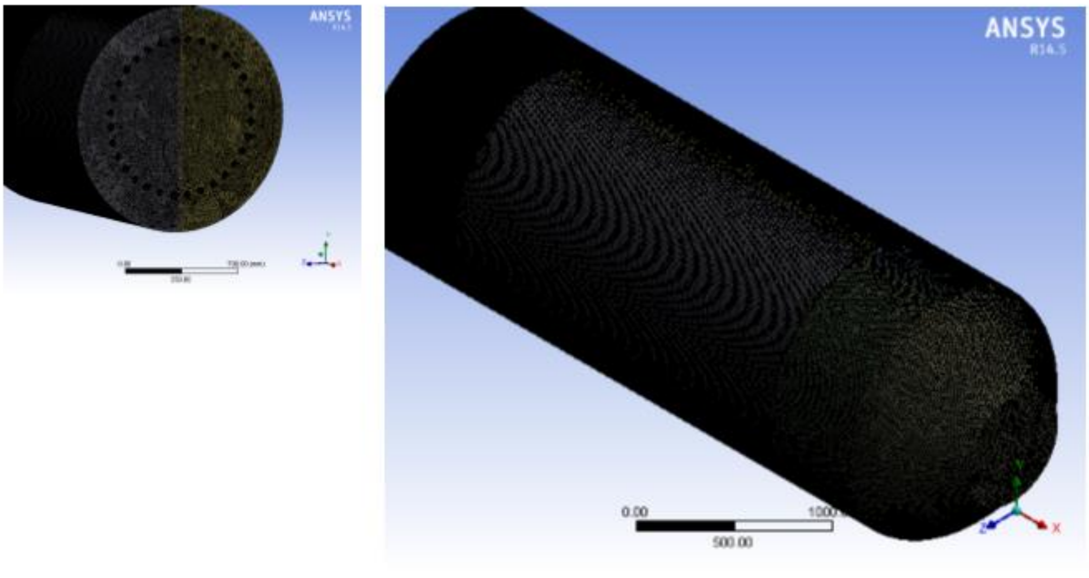


Figure 11: Fire Side Mesh for Radiant Section

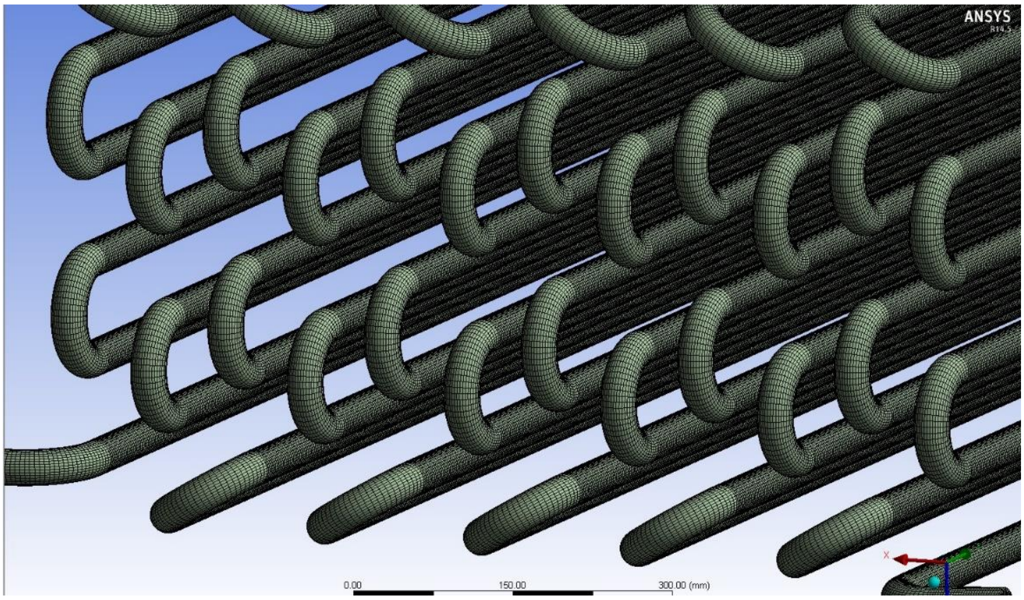


Figure 12: Mesh Grid for First Convection Tubes (Second Tubes)

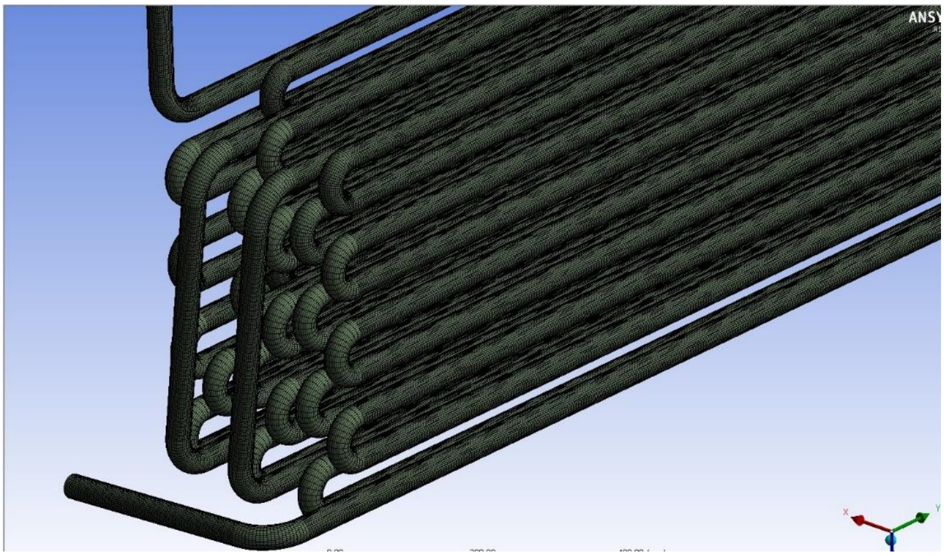


Figure 13: Mesh Grid for Second Convection Tubes (First Tubes)

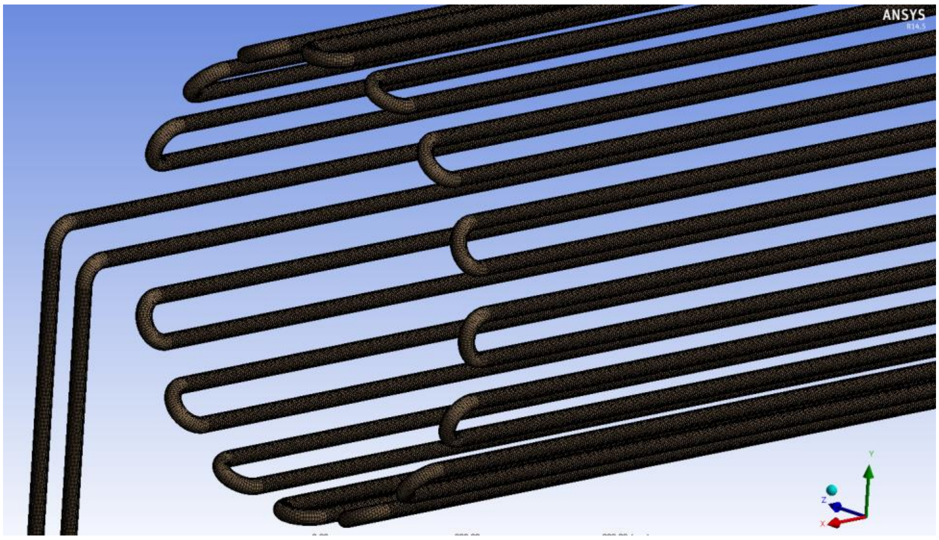


Figure 14: Mesh Grid for Radiant Tubes

Solution Strategy

A 16-core cluster of Intel® Zeon 3.2 GHz processor with 256 GB memory in each core was used to run the simulation. The interconnection between the cluster was achieved through Intel MPI architecture.

The simulation was initialized using the Hybrid-Initialization of Ansys Fluent and started without combustion reactions and radiation model. The simulation was run to capture a stable behaviour in terms of velocity and energy residuals. In a second step, the ignition process was performed to include the combustion reactions in the domain. To ignite, a spherical volume was defined next to burner tiles and given small values of combustion products with a high temperature (3000 °C). The simulation was restarted with a very low under-relaxation factor of density, and as solution progresses, the under-relaxation factors were slowly increased to ensure that the solution achieved a stable solution. In the last step, the radiation model was activated and simulation process was launched to reach the end point. The complete simulation required 160 hours of processor time to run. In addition to monitoring the velocity, energy and species residuals, the net mass and energy imbalance of the simulation were also monitored.

Once the combustion CFD model was converged, the heat fluxes on the tube surfaces were computed and passed to water side two-phase simulation as boundary conditions. In two-phase flow CFD simulation, the value of the vapour phase fraction at tube outlet was being monitored to confirm the simulation convergency.

Result and Discussion

- **Validation**

In the first validation step, the composition of combustion products was compared to field data. It was found that the CFD results is in good agreement as the pie chart in Figure 15 shows:

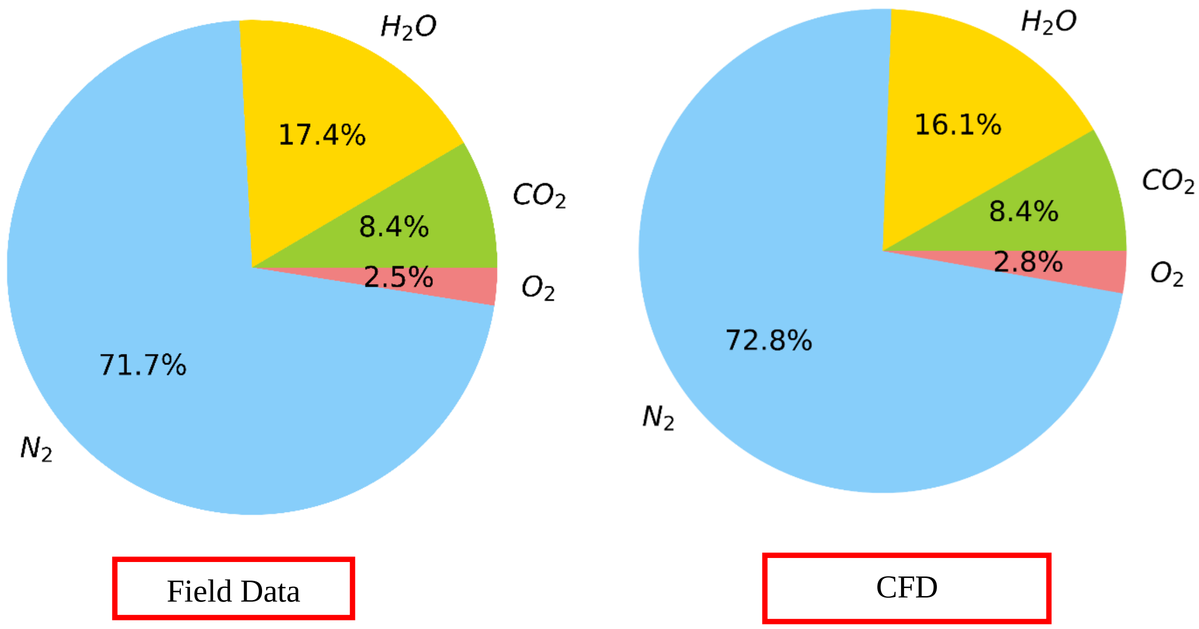


Figure 15 The composition of products of combustion

As Table 2 - Table 4 indicate, there is a fair agreement in terms of heat flux and flow temperature between field data-sheet and CFD model in Radiant section and first convection tubes.

Table 2: Comparison between field data-sheet and CFD results in Radiant section

Parameter	**Field Data**	**CFD**
Average Radiant Section Flux (kW/m2)	52.77	47.576
Flue Gas Temperature leaving Radiant Section, (°C)	1144	1152.18

Table 3: Comparison between Suncor data-sheet and CFD results in Bare tubes

Tube ID	**Heat Flux CFD** (kW/m2)	**Field Data** (kW/m2)
#1	51.85	52.96
#2	24.50	30.57

Table 4: Comparison between Suncor data-sheet and CFD results in Finned tubes

Tube ID	**Heat Flux CFD** (kW/m2)	**Field Data** (kW/m2)
#3	50.6	54.05
#4	28.61	29.78
#5	17.17	13.30

- **Result**

Figure 16 to Figure 18 display the temperature and velocity profile through vertical cuts inside OTSG domain. The flue gas passing the transition zone entering the convective section has a considerable high temperature, which is the consequence of combustion and the heat production in radiant section. The prediction of the blue zone in the middle of the convective section confirms the cooling effect imposed by BFW entering the tubes of the convective part. Then, the flue gas coming from the convective section enters the stack and moves upward with a fixed temperature due to the exposure of the stack adiabatic surface.

Figure 16 and Figure 18 illustrate the velocity contour and vector through the vertical cut in the domain. These figures represent the shape of the jet. The flue gas is transferred from the radiant part to the convective part by the transition section. The flue gas passes over the tubes and then reach the entrance of the stack. Then, the gas moves upward to exit at stack outlet.

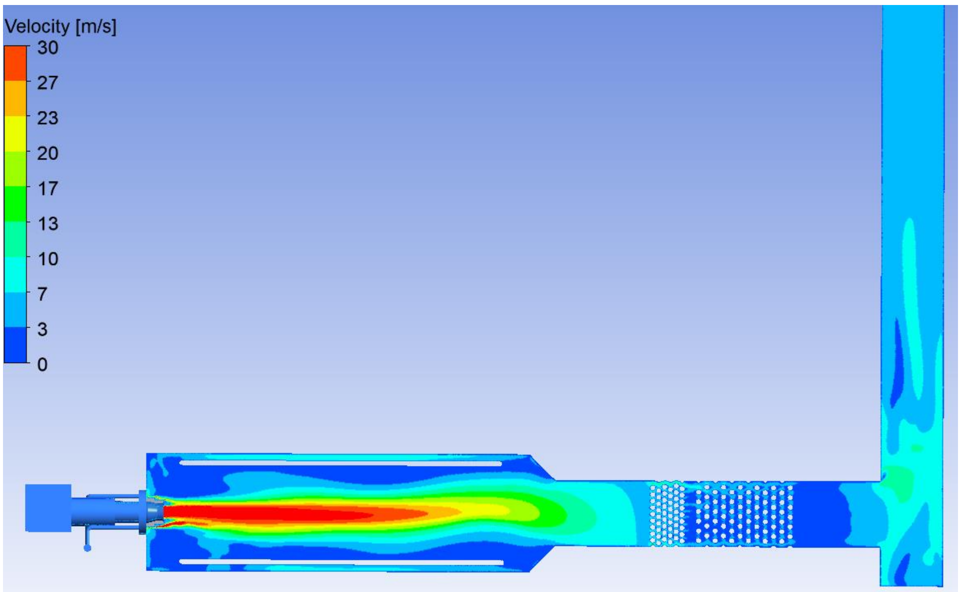


Figure 16: Velocity contours in vertical cut in the middle of OTSG

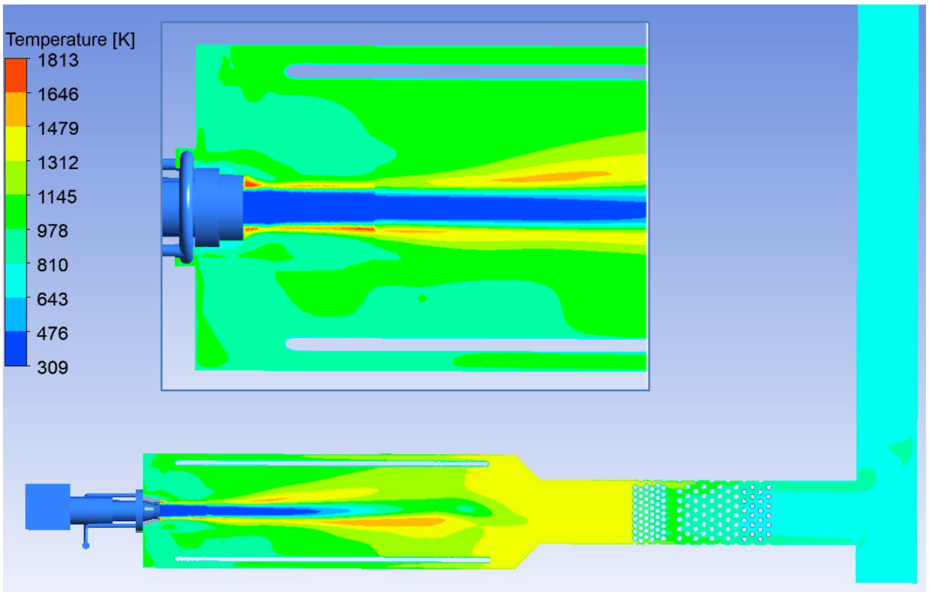


Figure 17: Temperature contours in vertical cut in the middle of OTSG

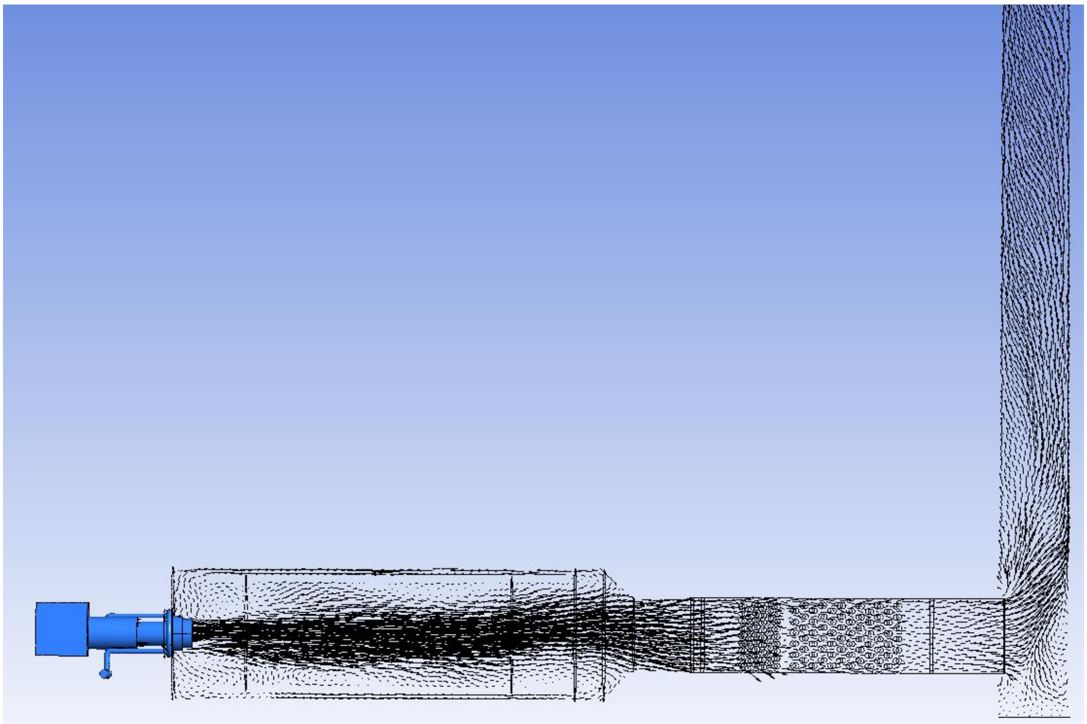
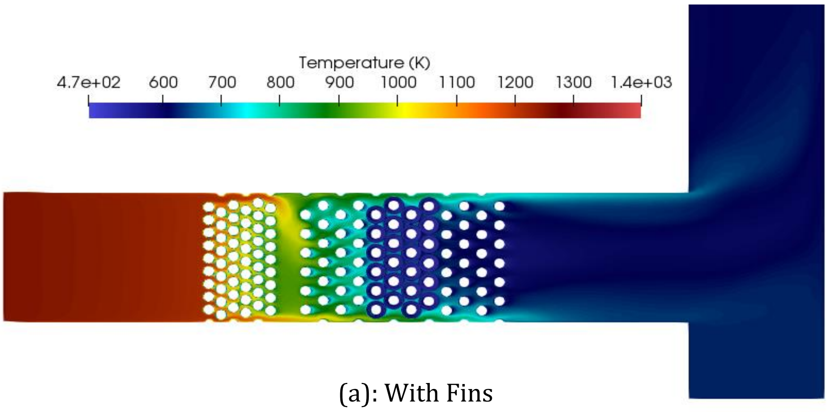
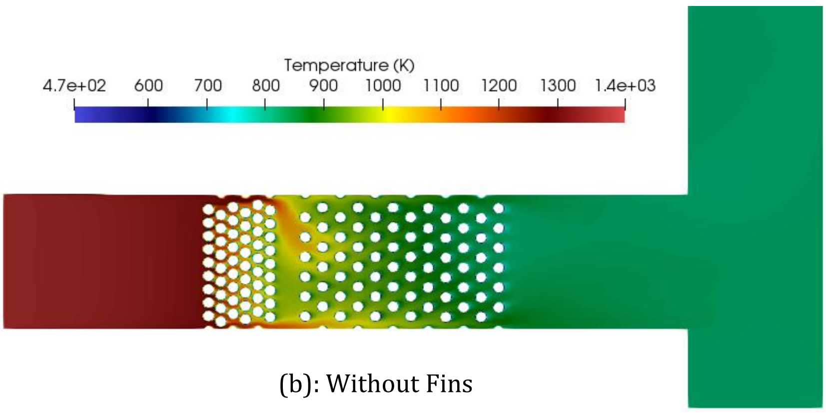


Figure 18: Vector vectors in vertical cut in the middle of OTSG

An analysis on the convection box with a more detailed shape of the fins was performed and it concludes that fins addition drops the stack temperature (Figure 19). The fins in this analysis are based on baffles definition. Baffle is recognized as zero-thickness walls. The amount of heat absorption is not entirely captured on the tubes without consideration of fins geometry. It plausible to attribute this issue to the absence of detailed fin geometry on the tubes in the model. The geometry of fins leads to additional disruption of the gas-side flow (Figure 20), and enhances the heat transfer characteristics of the fin. The trade-off for this enhanced performance, is increased pressure loss.

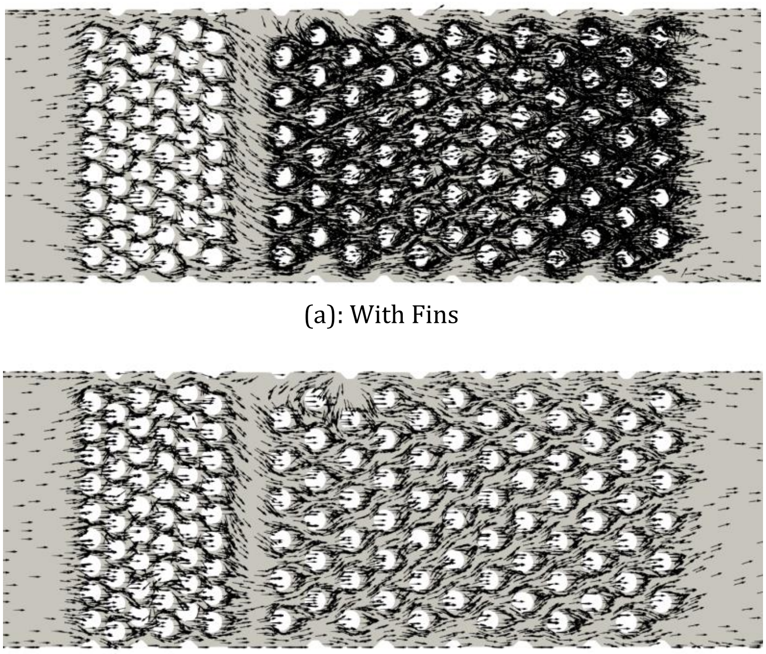


(a): With Fins



(b): Without Fins

Figure 19: Temperature distribution in convection box with and without fins on the first tubes



(a): With Fins

(b): Without Fins

Figure 20: Velocity contour profile on a convention box with and without fins on the first tubes

The Steam Generation through the pipes was modelled and then coupled with flue gas side using those heat fluxes predicted by the combustion CFD model on the tube surfaces. The results are shown in Figure 21 to Figure 23.

For convective part as shown in Figure 21 and Figure 22, at the elbows most of the vapour remains towards the intrados of the elbow, whereas the remaining liquid resides at the extrados of the elbow. The distribution is more drastic for second tubes due to presence of more liquid compared to radiant section (Figure 23).

While approaching the elbow most of the vapour occupies the upper portion of the pipe due to the buoyancy effect. Similarly, at the elbow, most of the vapour resides at the intrados of the elbow due to the centrifugal force acting on the vapour particles.

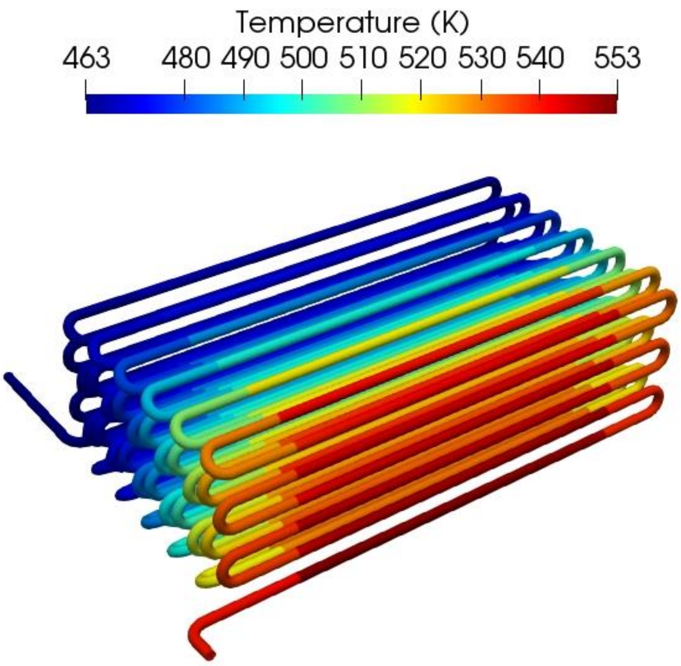


Figure 21: Temperature contours within first tubes in convention box– whole pipes

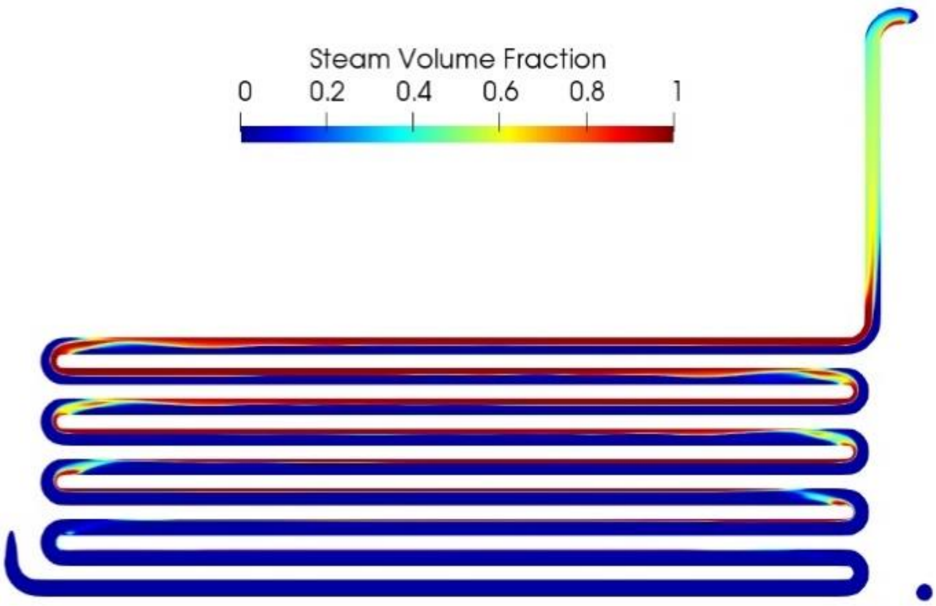


Figure 22: Vapor phase fraction contour within second convention tubes in convention box on a vertical cut

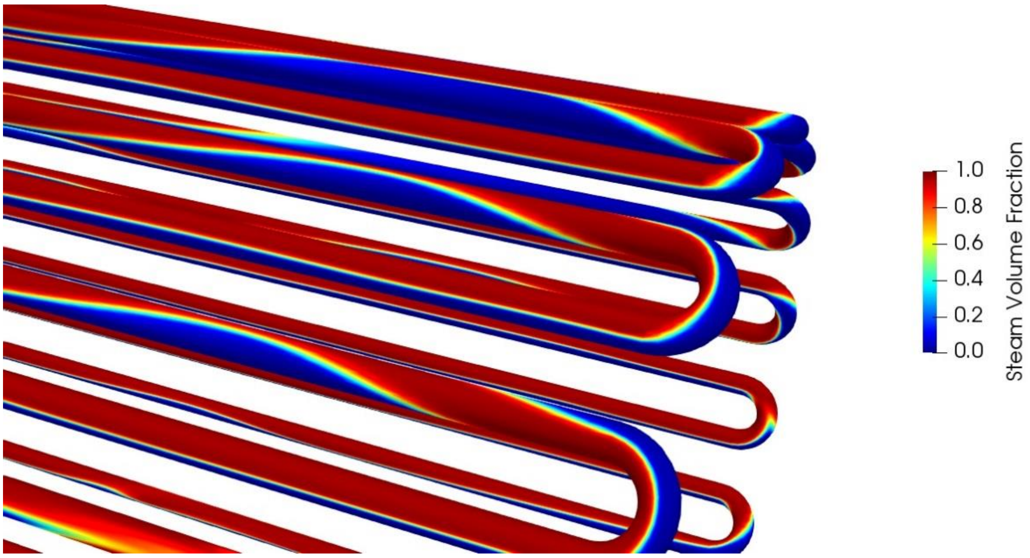


Figure 23: Vapor phase fraction contour within radiant tubes

Conclusion

A CFD simulation study of the combustion in the OTSG was performed. The results represent the shape of the jet and show that the flue gas goes through the transition part and passes over the convective tubes and then reach the entrance of the stack. Then, the gas moves upward to exit at stack outlet. The temperature profiles indicated that the flue gas passing the transition zone and entering the convective section has a considerable high temperature (1152.8 °C), which is the consequence of combustion and the heat production in radiant section.

The prediction of low temperature zone in the middle of the convective section confirms the cooling effect imposed by BFW entering the tubes of the convective part. Then, the flue gas coming from the convective section enters the stack and moves upward with a fixed temperature due to the exposure of the adiabatic surface of the stack.

An anlsysis with inlcuion of fins geometry on the second tubes of convebtiob box was performed and it indicated the fins addition on the second tubes improves the reults as it drops the flue gas temperature at stack outlet. The increased pressure loss caused by the fins in flue gas flow is the main reason for the temperature drop in the stack.

Results of the steam generation model show most of the vapour remains towards the intrados of the elbow, whereas the remaining liquid resides at the extrados of the elbow. On the other side, while approaching the elbow most of the vapour occupies the upper portion of the pipe due to the buoyancy effect. Similarly, at the elbow, most of the vapour resides at the intrados of the elbow due to the centrifugal force acting on the vapour particles.

We are studying the degree of coupling between combustion and steam generation necessary to accurately model the OTSG. This is proceeding using conjugate heat transfer which might be an essential consideration in finned-tubes. The inclusion of conjugate heat transfer (coupled convective-conductive heat transfer) may be necessary in order to determine the local heat transfer characteristics of the fin to enhance the current CFD results in this paper. We are also investigating another interesting aspect of the application of CFD together with additive manufacturing to create more efficient, low emissions burners.

References

E. Charles & Jr. Baukal, 2003*,* Industrial Burners Handbook, *CRC Press*

E. Charles & Jr. Baukal *2012, The John Zink Combustion Handbook, CRC Press*

A. Singha & M. Forcinito, 2016, Thermo-Mechanical Analysis of a Refractory - Root Cause Analysis, 2016 NAFEMS Americas Conference, Seattle, Washington.

A. Singha and M. Forcinito, 2018, Emission Characteristic Map and Optimization of NOx in 100 MW Staged Combustion Once-Through-Steam-Generator (OTSG), ARFC Industrial Combustion Symposium, Salt Lake City, USA

A. Singha and M. Forcinito, 2019, Modelling Different Aspects of Once Through Steam Generators, NAFEMS, Quebec City, Canada

Turns, S. B., 2012. Introduction to Combustion: Concepts and Applications, McGraw Hill.

Diez L, Cortes C, Artauzo I, Valero A, 2001. Combustion and Heat Transfer Monitoring in large Utility Boilers, *International Journal of Thermal Sciences*, 40:489-496.

Versteeg H. K., Malalasekera W. 1995. An Introduction to Computational Fluid Dynamics – The Finite Volume Approach, Longman Scientific and Technical.