

CFD analysis of an **industrial burner** for a regeneration gas heater application

Burners are widely used to satisfy the request of thermal energy in many industries. The design of a burner and the related heat transfer equipment must fulfill severe safety requirements, in order to avoid issues during the operation stage. One of the risk is that the flame can impinge onto tubes or other part of the

equipment, with consequent safety issues. In the oil and gas industry, some fired equipment design practices dictate the maximum flame height with respect to the size of the radiant chamber. Flame length is usually determined by performing a dedicated firing test, but CFD can be conveniently used for this purpose. The present work describes a numerical analysis of a gas-fired burner in a vertical cylindrical fired heater. Starting from the 2D drawing, a 3D model of the burner and the radiant section of the heater was created and meshed with ICEM CFD. The solver ANSYS CFX was used to run the simulation. The analysis was developed in cooperation with EnginSoft, especially in the development of the 3D model and meshing stage. Different load conditions of the burner have been tested, in order to check the flame height at different conditions.



Figure 1 - Typical fired heater: cylindrical heater with vertical coil



Introduction

Burners are used to convert the chemical energy of a fuel into thermal energy through a combustion process. The aim of these devices is to mix the fuel and air in the right proportion, initiate and maintain ignition and ensure the stability of the flame. A burner shall also be operated safely: this means that the flame shall be stable and the flame pattern shall not damage the equipment, like refractory or heat transfer surfaces. It shall also meet the required turndown capacity and fulfill the local requirements in term of pollutants emissions. Burners are widely used in the Oil&Gas industry and are key components within API 560 direct fired heaters.

In direct fired heaters, the heat released by the burners is transferred to a process stream that can be used for many different purposes (e.g. feed for a reactor or a distillation column, regeneration gas for an absorption unit, hot oil heating, etc.). A typical



Figure 2 - Burner, (a) General arrangement, (b) swirler and burner tip

API 560 direct fired heater is divided into two sections: a radiant section, where the combustion process takes place, in which the heat is mainly transferred by radiation, and a convection section, where the flue gas cools down by convection heating the fluid inside the tubes (Figure 1). In API 560 design, the first three tube rows encountered in the convection section are made by plain tubes and are called shock tube rows: they shield the remaining part of the convective section, typically made by finned tubes, from the direct radiation of the flame. Part of the heat in the shock tube rows is transferred by direct radiation of the flame.

requirements regarding burner selection and

arrangement has to be fulfilled to ensure a safe operation of the heater. Some fired equipment design practices dictate the maximum flame height with respect to the size of the radiant chamber. The height of the flame is usually assessed with a firing test, except when the burner capacity exceeds the maximum allowable load of the test facility. In this case CFD can be conveniently used for the purpose of a preliminary assessment, since the only opportunity to perform a physical test would be given with the burner already installed at field. The development of an accurate CFD model could represent an economical and time saving alternative to both firing tests and field tests.

	Case 1	Case 2	Case 3
Burner heat release (MW)	15.52	12.37	8.7
Radiation duty absorbed by radiant tubes (MW)	8.57	7.38	5.81
Radiation duty absorbed by shock tubes (MW)	0.45	0.37	0.27
Air mass flow (kg/s)	5.873	4.689	3.406
Fuel mass flow (kg/s)	0.352	0.283	0.191

Table 1 - Analysis cases

During fired heater design stage, a number of Figure 3 - 3D models, (a) 180° model, (b) 60° model

Case Study

The case study is a regeneration gas fired heater designed and supplied for a project in Oman. Design is of the vertical cylindrical type according to API 560. The heater is equipped with a single burner and forced draft fan. The fuel is a gas constituted by a mixture of various hydrocarbon. The main components are methane, ethane and propane: those components accounts for the 90% of the total mixture. The radiant section is cylindrical, and the process tubes are vertical. The fluid is distributed in four parallel passes, with 12 tubes per pass (Total 48 tubes in the radiant section).

The burner is placed on the floor of the chamber. It's a staged fuel burner: a portion of the fuel and all the combustion air are mixed in a primary combustion zone, while the remaining part of the fuel is supplied through a series of nozzles around the perimeter of the burner. Before entering the combustion chamber, the air flows through a swirler, to ensure proper mixing in the primary combustion region. A picture of the burner is reported in Figure 2.

Aim of the analysis is to determine the flame length in three different operating conditions: Table 1 reports the burner heat release, duty absorbed and flow rates

for both air and fuel for the three cases considered. Many definitions for the flame surface are possible: for the purpose of this work the flame has been defined as the iso-surface at a temperature of 1400K.

CFD model

The model has been implemented by Enginsoft. Starting from the burner manufacturer drawings, a 3D model has been created directly using ICEM CFD, which was also used to mesh it.

Since the focus of the analysis is the flame structure, only the radiant chamber has been modeled. There is the possibility to analyze just a portion of the domain, given the symmetries that are present. Unfortunately those symmetries doesn't quite match between each other: there are 6 nozzle for the staged fuel, 10 for the primary fuel, 4 tube passes and 48 tubes. The only effective symmetry is a 180° portion of the domain, also because the outlet of the domain is rectangular, not circular.

Simulating half of the radiant chamber would be the better solution from an accuracy point of view. Anyway it would require a significant computational time. In order to understand the effect of the periodicity, two models have been created: one model has a periodicity of 180°, one of 60° (the periodicity of the staged fuel nozzles). The nozzle diameter of the burner for the second model has been opportunely scaled in order to obtain the correct exit velocity. Also the temperature of the process fluid has been

adapted to remove the correct process duty out of the domain. The aim is to understand the impact of the assumptions of the simplified model on the flame height. For this purpose, the two models have been solved for Case 1, and the results have been compared. The two models are depicted in figure 3.

The case has been solved with ANSYS CFX.

The following boundary conditions have been set:

- Inlets: Mass flow rate, composition and temperature have been specified for both air and fuel. For the 60° an inlet velocity profile has been set; it has been determined with an independent model of the swirler.
- Outlet: gas flow exit the fluid domain. A pressure has been set. An energy sink has been defined in order to consider the radiant heat transfer to the shock tubes. A pressure drop has been set to simulate the presence of the tubes of the convective section and set the correct air pressure at the inlet.
- Process tube walls: Heat transfer coefficient and bulk temperature have been set; temperature values are indicate in datasheet, wall heat transfer coefficient and tube emissivity have been set in order to extract by tubes the heat duty defined in table 1.
- External walls: the heat loss is simulated with a heat transfer coefficient and an outside temperature

The chemical reactions which take place in the system have been explicitly simulated in order to accurately predict the temperature field in the entire system. The following reaction mechanism has been included in the model:

$$CH_4 + \frac{3}{2}O_2 \longrightarrow CO + 2H_2O$$

$$C_2H_6 + \frac{5}{2}O_2 \longrightarrow 2CO + 3H_2O$$

$$C_3H_8 + \frac{7}{2}O_2 \longrightarrow 3CO + 4H_2O$$

$$CO + \frac{1}{2}O_2 \longrightarrow CO_2$$



Figure 4 - Velocity streamlines, Case 1, (a) 180° model, (b) 60° model,

SST (Shear Stress Transport) turbulence model has been applied. Buoyancy effect has been considered. Monte Carlo Radiation Model has been applied for the radiation heat transfer.

Results

The Velocity streamlines are reported in Figure 4 for the 180° model and the 60° model. It can be seen the typical flow pattern of a heater: fuel and air enters at the bottom, flue gas flows through the center of the chamber. Part of the cold flue gas flows downward along the tubes, at the periphery of the chamber. The streamlines patterns are quite similar for the two models.

Figure 5 reports the temperature profile at the symmetry plane for the two models: the points with highest temperature are located in the combustion region over the burner, where the chemical reactions take place. The coldest region is located in the lower part



Figure 5 - Temperature profile, Case 1, (a) 180° model, (b) 60° model



Figure 6 - Isosurface at 1400K, Case 1, (a) 180° model, (b) 60° model

of the chamber far from the burner: the recirculating flue gas flows downward and it's cooled by the process tubes. The difference in the temperature profiles between the two models are concentrated in the lowest part of the chamber.

The iso-surface at 1400K are depicted in Figure 6. There are some differences in the shape of the surfaces, especially in the bottom part. Figure 7 reports the maximum height of a iso-surface for many temperatures: at 1400K the difference between the values



Figure 7 - Maximum height for iso-surfaces at given temperatures

predicted by the two models is limited. Both the values are lower than the maximum flame height for this specific heater (9 m). The other cases have been runwith the 60° model. The streamlines for those cases are reported in Figure 8, while the iso-surfaces are depicted in Figure 9. The flame height are respectively 7.37m and 4.83m for case 2 and 3.

Conclusions

A CFD analysis of an industrial burner has been performed. Two models have been created and compared: in the first one half of the radiant chamber has been simulated, in the other one sixth.

The two models have been tested with the same conditions: velocity streamlines, temperature profile and flame surface have been compared. Although the smallest model doesn't fulfill all the symmetry requirements of the real geometry, its results are in good agreement with the biggest model calculation.

Other two cases have been analyzed, and the flame height has been determined.

The results proof that the flame height can be determined with reasonable accuracy with the 60° model, with a consequent save in terms of computational efforts.

The heater is planned to be started-up in 2018. Accurate observation of the flame will provide significant feedbacks to the outcomes from the CFD study. Positive feedbacks will provide strong support in confirming CFD models as a reliable alternative to firing tests.



Figure 8 -Velocity streamlines, 60° model, (a) Case 2, (b) Case 3



Figure 9 -Isosurface at 1400K, 60° model, (a) Case 2, (b) Case 3

D.Agazzi, T.Odry, M.Rottoli - Brembana & Rolle A. Troia - EnginSoft

Thermal Analysis with ANSYS

The effects of heat and thermal management of structures is more and more critical as performance limits are pushed further by the need to have lighter, smaller and more efficient designs. Convection, radiation and conduction loads are obvious, but the need to include the effect of power losses and thermal energy from external sources such as pipe flows means that analysts need to have more tools at their disposal to simulate thermal models accurately.

For more information: Anna Rosa Troia, EnginSoft a.troia@enginsoft.com

