NOVEL BURNER DESIGN SUPPORTED BY CFD TO MINIMIZE DEPOSITS INSIDE COMBUSTION CHAMBERS OF SAMARCO PELLETIZING FURNACES

Maycon Athayde²
Sergio Fernando Nunes³
Guilherme Araujo Lima da Silva⁴
Francisco Domingues Alves de Sousa⁵
Marcos Noboru Arima⁴

²Eng., Samarco Mineração
³MSc., Samarco Mineração
⁴PhD., ATS4i Aero-Thermal Solutions for Industry
⁵Eng., ATS4i Aero-Thermal Solutions for Industry
INTRODUCTION

- In order to reduce greenhouse gases emitted and operational cost, Samarco choose to enhance the thermal matrix converting their 3 pelletizing furnaces from heavy oil to natural gas. This action improved combustion system safety and the process control.

- Nevertheless, a significant increase in the deposited dust rate had been detected on the bottom of the combustion chamber as well as an over-deflected flames.
INTRODUCTION

The downcomer air caused deflection and pushed the flame zone towards the bottom. Consequently, the high temperatures and CO concentrations cause sintering of deposited particles, and then a high frequency of cleaning was necessary to do not chemically wear the refractory lining.

55.2% Fe₂O₃  41.3% FeO  1.1% SiO₂  1.1% Al₂O₃  0.9% CaO
INTRODUCTION

Three main operational issues:

- **High material deposition rate** at lower surface of the combustion chamber with no more melted but then **sintered**, that could lead an chamber obstruction if it does not clean out frequently;

- **Large flame mean diameter** touching the refractory walls at lower surface, increasing bottom temperatures of chamber refractory lining;

- Higher **%FeO level** in sintered material inside the combustion chamber when using gas than when using heavy oil as fuel;
OVERVIEW OF THE COMBUSTION CHAMBER

Mean center flame distance to lower refractory wall:

\[ Y = 2640 \]
OBJECTIVES

Develop a novel burner design to minimize the operational difficulties caused by deposits inside pelletizing furnaces chambers.

METHODOLOGY

• To solve the problem, instead to develop based on “Cut-and-Try” methodology, Samarco used a methodology based on analytical methods and together with field experience supported by a Computational Fluid Dynamics tools (CFD) held by ATS4i.

• In these study were used CFD approach based on finite volume to model the flow pattern inside the chamber
ROOT CAUSES AND SOLUTION PROPOSAL

• High momentum unbalance among streams the flame center line was deflected and its boundaries touched the lower wall.

• Higher upstream pressure of air when using liquid fuel than the natural gas upstream regulated pressure;

• Heavy oil sprays mass density higher than natural gas density.

• As the momentum depends on the upstream pressure, discharge velocity, density and area, the ratio between the oil fuel spray and downcomer air were higher than the same ratio when using natural gas.
According to Lefebvre\textsuperscript{[3]}, the deflection of the jet depends on the ratio of momentum flux between the jet and mainstream cross flow, which is defined as $J$:

\[
\frac{Y}{d_{jet}} = 0.82 \cdot J^{0.5} \left( \frac{X}{d_{jet}} \right)^{0.33}
\]

In the case of the free jet, the jet trajectory can be described by a semi-empirical one-dimensional correlation:
Analytical Models Developed

- Fluidodynamics calculations were initially used also to check the root cause influences and to study the sensitivity to design parameters variation.
  - Natural Gas thermo-physical properties;
  - One-dimensional compressible flow for each air and natural gas nozzle to estimate mass flow, discharge velocity and momentum flux with varying nozzle upstream pressure;
  - Adiabatic and pseudo-adiabatic flame temperature for liquid fuel oil and natural gas for furnace units 1, 2 and 3;
  - Refractory wall mean temperature for liquid fuel oil and natural gas for furnace units 1, 2 and 3;

- The CFD simulation took much less time. It simplified the geometry and reduced the number of cases and establish coherent boundary conditions.

- After the CFD runs, those models served as a sanity check to debug errors and to improve setup.
Results

Nozzle Configurations
The dimensions and operational parameters for configurations A, B and C chosen. The parameters shown on table below was calculated on the previous model.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>24</td>
<td>348.5</td>
<td>230</td>
<td>236</td>
<td>42970</td>
</tr>
<tr>
<td>B</td>
<td>18.8</td>
<td>348.5</td>
<td>556</td>
<td>341</td>
<td>94325</td>
</tr>
<tr>
<td>C</td>
<td>27.0</td>
<td>348.5</td>
<td>149</td>
<td>194</td>
<td>28600</td>
</tr>
</tbody>
</table>
Trade-off CFD Results

Methane mass fractions
Symmetry Plane
No combustion – Isothermal jet

For a qualitative reference, at 20°C and atmospheric pressure, the stoichiometric limit for CH$_4$ combustion is 5.5%, the first and second limits of flammability are 3% and 11.2% in mass fraction.
CFD Results with Combustion

Nozzle B was elected as the best candidate to solve the flame deflection issue. A one simple step combustion reaction approach was applied.

\[ CH_4 + 2 \cdot O_2 \rightarrow 2 \cdot H_2O + CO_2 \]
CFD Results with Combustion

Nozzle B was elected as the best candidate to solve the flame deflection issue. An one simple step combustion reaction approach was applied in order to evaluated the streamline pattern inside the chamber.

\[ CH_4 + 2 \cdot O_2 \rightarrow 2 \cdot H_2O + CO_2 \]
A modified nozzle was evaluated in order to reduce even more the inflection of the flame. This device was, then, assembly in the chamber to see how its behave.
OPERATIONAL RESULTS

Before

- Flame unstable and high diameter
- Touching the chamber bottom and increasing lining temperature
- High dust deposition rate
OPERATIONAL RESULTS

- Less dust accumulated in the bottom surface because the particles followed the streamlines.
- Centralized flame
OPERATIONAL RESULTS

Combustion chamber after 2 weeks of operation with and new burner.

original burner

configuration B Modified
CONCLUSIONS

• The novel burner was tested successfully in the pelletizing furnaces and presented a significant minimization on the deposited dust rate and less temperature on the lining.

  • Therefore, the excellent operational performance validated the methodology developed and confirmed CFD as an alternative to develop innovative designs.

  • However, it need to be applied with criteria and requires also the field and professional experience for adequate tools selection, problem setup and analysis of the results.