TWO-DIMENSIONAL CFD SIMULATION APPROACH OF COMBUSTION IN A FURNACE USING REFINERY-OFF GASES WITH DIFFERENT COMPOSITION

Sergio Andrés Morales Restrepo [a]
Daniel Ricardo Barragan Noriega[b]
Viatcheslav Kafarov [c]

Research Center for Sustainable Development in Industry and Energy (CIDES)
Universidad Industrial de Santander (UIS)
Bucaramanga, Colombia

[a] sergiom80@hotmail.com
[b] engineer.barragan@gmail.com
[c] kafarov@uis.edu.co

Abstract *This work analyses the effect of the variability of the chemical composition and calorific power of refinery gases in the combustion process of furnaces using Computational Fluid Dynamics (CFD), which is a technique that has been proven useful to obtain detailed information about the operation of furnaces.*

Keywords: *Furnace, Contours, Combustion, Gases, Calorific Power*

INTRODUCTION

In the different processes carried out in the oil refining industry, mixtures of waste gases are produced commonly known as "refinery gases". Most of the time, these gases are used as fuel in different equipment such as furnaces, boilers or industrial burners, with the intention of saving energy and avoiding additional storage costs. Nevertheless, the chemical composition of refinery gases varies depending on the process from which they are produced and their calorific power can present a variation as much as 1000 Btu/ft^3, which causes important changes in the combustion process. This effect can be studied using CFD simulations.

FORMULATION OF THE PROBLEM

The mathematical models used to calculate the flow inside the furnace are based on the Reynolds-Average-Navier-Stokes (RANS) steady-state equations that contain mass, energy, chemical species and momentum. It was necessary to select mathematical models to solve turbulence, chemical reaction (combustion) and heat transfer(radiation) equations. In these simulations, the model selected to account for turbulence was the two-equation standard k-ε, which is the most used turbulence model in CFD, the model selected to solve combustion reaction was the PDF-mixture fraction, which is used for non-premixed combustion cases and the model selected to account for radiation was the Discrete Ordinate

model, the most robust and widely used radiation model in CFD. For more information about the mathematical models, please consult [1], [2], [3].

RESEARCH ANALYSIS

The configuration of the furnace in this study is typical box type with floor burner. Since the furnace and the burner are symmetric, a Two-dimensional approach can be used in order to reduce computational costs.

Comparative Two dimensional CFD simulation cases are carried out with fuels of different composition. The first case with natural gas (100% methane), second case a mixture with moderate propane content (>16%) and the third case a mixture with high propane content (>%35), obtaining temperature and CO mass fraction contours inside the furnace, which are compared to stablish the effect on temperature distribution and complete combustion.

CONCLUSIONS

The simulation of the first case showed the highest maximum temperature inside the furnace and the lowest CO mass fraction, that is that the combustion was complete. On the other hand, in the second and third case the combustion is incomplete and more CO is produced. This indicates that the combustion process is not efficient and needs to readjustment. The results are very important to stablish a starting point in the improvement of furnaces efficiency when using mixture gases.

REFERENCES

1. *F.A. Díaz Mateus and J.A. Castro-Gualdrón*, mathematical model for refinery furnaces simulation, CTF – Cienc. Tecnol. Futuro, vol. 4, pp. 89–99, 2010.
2. *A. J. M. Oprins y G. J. Heynderickx*, Calculation of three-dimensional flow and pressure fields in cracking furnaces, Int. Symp. Math. Chem. Kinet. Eng., vol. 58, núm. 21, pp. 4883–4893, nov. 2003.
3. *G. D. Stefanidis, B. Merci, G. J. Heynderickx, and G. B. Marin*, CFD simulations of steam cracking furnaces using detailed combustion mechanisms, Comput. Chem. Eng., vol. 30, núm. 4, pp. 635–649, feb. 2006.