COMPUTATIONAL FLUID DYNAMIC 3D SIMULATION USING BLENDED GASES BY IGNITING FURNACES WITH PARAMETERS CONFIGURATION

Engr. Nnadikwe Johnson[1], Ikputu Woyengikuro Hilary[2], ODIKi Esther E[3], Ibe Raymond Obinna[4] , Engr Ewelike Asterius Dozie [5]

[1] H.O.D in Department of Petroleum and Gas Engineering, Imo State University, Nigeria

[2] Lecturer in department of Petroleum and Gas Engineering, Nigeria, Nigeria Maritime University, Okerenkoko, Delta State, Nigeria

[3] Lecturer in department of Petroleum and Gas Engineering, Nigeria, Nigeria Maritime University, Okerenkoko, Delta State, Nigeria

[4] Lecturer in department of Chemical Engineering, Imo State Polytechnic, Nigeria

[5] H.O.D in Agricultural Engineering, Imo State University, Nigeria

ABSTRACT

Furnaces are important pieces of equipment in the petroleum refining sector because they supply the necessary heat for many operations. These furnaces are designed to run on natural gas, but in most cases, they run on a mixture of waste gases known as refinery gases (RG), whose molar composition varies depending on the process in which they are produced, but in most cases contains high levels of propane, hydrogen, and propylene, among other things. Most of the time, this is done to save energy and reduce storage costs. This shift in molar composition, however, results in a change in calorific power of up to 1,200 Btu/ft3, causing changes in the combustion process, lowering efficiency and raising pollutant emissions. The Computational Fluid Dynamics (CFD) technique is utilized in this study to simulate the combustion process in a representative portion of a typical refinery furnace utilizing RG as the fuel and three different molar compositions. The influence of molar composition fluctuation on the temperature and chemical species profiles inside the furnace is investigated using comparative CFD 3 D simulation instances. The results reveal that adding more gases to the furnace, such as propane, propylene, or hydrogen, increases the calorific power and peak temperature, but the temperature profile distribution is less uniform. When employing combination gases with low CH 4 content, the chemical species profiles inside the furnace show an increase in CO, indicating that a more extensive analysis of air excess and flow is required. The findings are significant because they can be used as a tool for determining whether it is more convenient to use refinery waste gases as a fuel in furnaces or not, as well as a starting point for a more detailed investigation into furnace operation and process safety.

Keywords: *CFD, Refining Gas, Dynamics, Configuration, Computational*

---

1. INTRODUCTION

To reduce storage costs and preserve natural gas, many refining methods are carried out in which waste gases, known as refinery gases, are created as by-products and used as fuel in furnaces and boilers. The molar composition of these refinery gases, on the other hand, varies substantially depending on the unit in which they are produced, resulting in variations in their calorific power.

The combustion process involves several disciplines that are closely related, including as thermodynamics, kinetics, and transport phenomena. The fundamental equations of mass, momentum, and energy conservation can be used to express these events quantitatively (Gentile et al., 2016). However, when there is turbulent flow, these equations, when paired with the supplementary equations, can only be solved analytically in extremely simple cases. As evidenced by several studies published in the literature on modeling and simulation of combustion processes in typical industrial furnaces using CFD, the application of the Computational Fluid Dynamics (CFD) technique has proven to be a very useful tool for providing valuable information about furnace operating processes.

Stefanidis et al. (2006) used computational fluid dynamics (CFD) to simulate the combustion process in a cracking furnace and evaluate the implications of several combustion models. Flow, temperature, and concentration profiles of chemical species were calculated in the combustion chamber of an industrial cracking furnace using two combustion models: The Eddy Dissipation Concept with full reaction kinetics and The Eddy Break Up Model with simplified reaction kinetics. They argue that in cracking furnaces, more advanced combustion models, such as the Eddy Dissipation Concept with exact reaction kinetics, should be used.

Aminian et al. (2010) used CFD to evaluate temperature and flow characteristics in an alternative industrial cracking furnace design. In the study, they used the RNG model k- to model turbulence, the P1 and DO models to model radiation, and the Eddy Dissipation Model to model combustion. The P1 and DO models for solving the radiation transfer equation and the Eddy Dissipation Model for combustion in industrial cracking furnaces were found to be suitable. A different design was also simulated to see if it could improve combustion gas recirculation and temperature distribution within the furnace.

Others used CFD to model and simulate combustion processes in furnaces, getting flow profiles, temperature, and species concentrations using known-composition fuels. For the CFD modeling of a refining vacuum furnace, Li et al. (2015) used a fixed composition of methane as fuel, although there are few studies in the literature that use fuels with variable calorific content and power. Using a three-dimensional CFD simulation, the goal of this simulation is to investigate the effects of using different composition and calorific power fuels in the combustion process in industrial furnaces. This may be performed by analyzing and comparing the temperature and CO mass fraction contours obtained from various simulated furnace scenarios. Cala et al. (2013) explored the effect of composition variability on combustion process characteristics such as the Wobbe Index, efficiency, and adiabatic flame temperature. The findings showed that a variety of chemical compositions of refinery gases might be identified to increase efficiency.

However, other factors such as temperature behavior inside the furnace must be investigated in order to reduce or even avoid overheating and corrosion issues. Temperature profiles derived from a CFD simulation, as well as pollution emissions, can be used to investigate this issue. As a result, this study looks into the effect of changing chemical composition on temperature and CO mass fraction profiles. Because the purpose at this point in the study is to use a simplified model (i.e., simplified geometry and burner) as a starting point for the entire investigation, the model is confined. In their studies, several authors in the literature use simplified geometry models. This is demonstrated in the study by (Iancu et al., 2017), which uses a simplified 2D geometry and burner design for a CFD simulation of the combustion process in a furnace. In the next stage of this project, a more complex geometry and burner will be investigated. The goal of this paper is to present preliminary interesting results that can be used as a starting point for future studies on improving furnace efficiency and lowering pollutant emissions.

2. FURNACE CHARACTERISTICS

The furnace under investigation is a non-premixed cylindrical kind with a burner in the floor, which means the fuel and air inlets are separated and it is adiabatic. Because of the furnace's symmetry, only half of the whole geometry needs to be modeled. Figure 1 shows the geometrical dimensions of the simulation, which was run in the commercial code ANSYS FLUENT. The air and gas entry fluxes, which are 1.682 kg/s and 0.0974 kg/s, respectively, as well as the proportions of the gas mixtures mentioned in Table 1, are determined by the boundary conditions. Because the furnace is cylindrical and symmetric, only half of it needs to be simulated, lowering computational costs dramatically. The air and fuel temperatures are 300 K at both inlets, the Fuel Stream Rich Flammability Limit is 0.06, and the air surplus is 10% in all cases. For pressure-velocity coupling, the SIMPLE method was employed, and for equation discretization, the first order UPWIND method was used. There are 800,000 elements in the computational mesh.

Figure 2 shows a zoomed-in part of the burner, which has two simple circular air and gas inlets. The air intake measures 1.6 meters in diameter, while the gas inlet measures 0.08 meters. Due to the demand for air flow in this type of combustion, the gas inlet is situated in the center of the air inlet, which is larger in size.

3. MATHEMATICAL MODELS

The mathematical models required to perform the simulation are based on the basic equations of mass, motion, energy, and chemical species conservation. This means that a turbulence model, a combustion model, and a radiation model must all be chosen before constructing a furnace simulation using CFD. There are a variety of CFD models available, and the simulation case object of study determines which one to use. For further information on the mathematical model, see (DazMateus and Castro-Gualdrón., 2011).

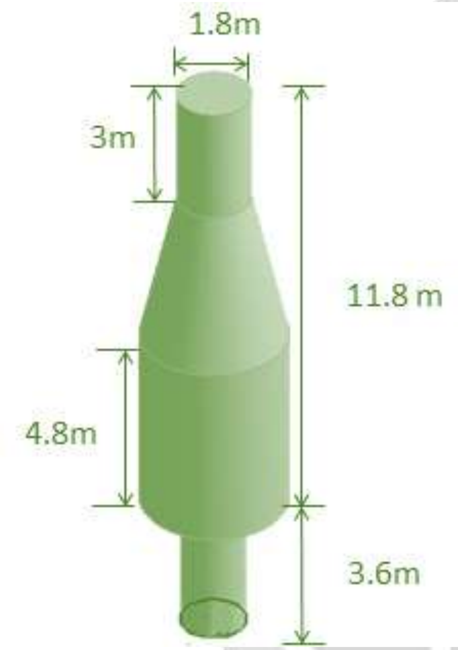

Figure 1: Geometry of the furnace object of study

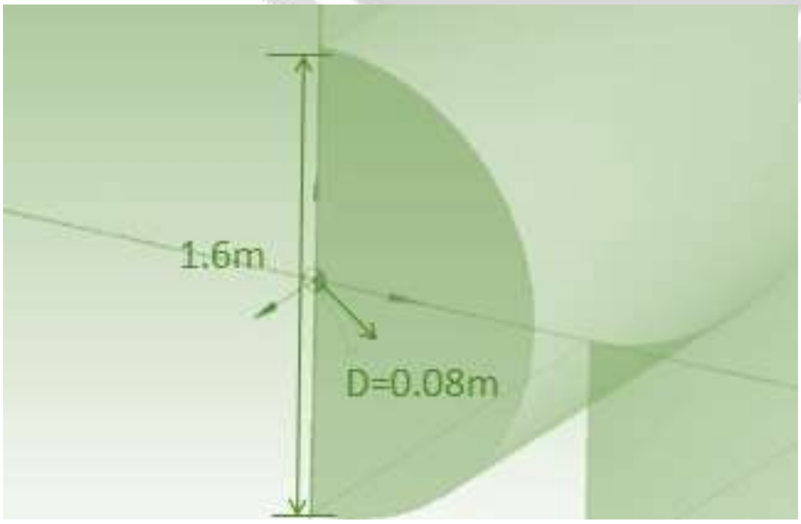

Figure 2: Burner zoom of the furnace object of study

3.1 Turbulence model

The turbulence model utilized in this study is the conventional k-model, which has two equations: one for k (kinetic energy) and another for k (kinetic energy) (dissipation of kinetic-turbulent energy). Because it is adaptable to a wide range of applications, this is the most often used turbulence model.

3.2 Combustion model

In this case, the "PDF Mixture Fraction" model is the most usually used. By combining two equations, one for the average mixing fraction and the other for the variation of the average mixing fraction, this model depicts species mobility.

3.3 Radiation model

Radiation contributes for more than 80% of heat transfer in the radiant zone of the furnace (Li et al., 2015). To depict this phenomenon, a radiation model must be chosen. Because it is the most robust model available and covers the widest range of optical thickness, the Discrete Ordinate Model (DOM) was chosen in this case.

In addition, the effect of flue gases on radiation, particularly water vapor and CO2, must be taken into account. The emissivity of various gases is calculated using the weighted sum of gray gases model (WSGGM).

4. RESULTS

The effect of employing gases of varying composition for combustion in industrial furnaces was investigated in this work using a three-dimensional simulation using CFD, for which three instances were simulated using gas mixtures of varying composition, as indicated in Table 1.

The temperature profiles and CO mass fractions generated by the industrial furnace CFD simulation were examined to learn and conclude significant aspects about the combustion process utilizing the three gas combinations listed in Table 1.

Table 1: Compositions of the mixture gases used in the 3 simulation cases

Component	Natural gas (%)	Mixture 1 (%)	Mixture 2 (%)
CH_4	100	55	25
C2H6	0	10	8
C3H8	0	0	25
C4H10	0	4	10
C2H4	0	5	10
C3H6	0	2	5
H_2S	0	4	2
H_2	0	20	15
Inferior			

Calorific Power (Btu/ft^3)	983	955	1530

Figure 3 shows the temperature profiles within the furnace as a result of the three simulation scenarios using natural gas, mixture 1, and mixture 2. The simulation involving natural gas serves as the foundation or ideal example for examining the impact of chemical composition changes. From the center to the chimney, a consistent pattern of heat dispersion can be seen. In the second simulation, the temperature profile created by mixture 1 differs from the profile created by natural gas.

In this circumstance, the radiating zone (half) of the furnace has a higher temperature uniformity, implying that the furnace's central zone heats up faster than the chimney. This is due to the presence of additional gas mixture components such as ethane, propane, and hydrogen. In the third case, where there are more heated zones, the same thing happens. More hot spots form inside the furnace when the methane concentration of the fuel decreases and the calorific value rises due to the presence of other components.

Figure 4 shows the trends of CO content in the furnace when using natural gas, mixtures 1 and 2. The molar concentration of CO inside the furnace is concentrated at the burner's exit, according to this data, and expands as the molar proportion of CH4 decreases. This means that if the fuel contains a higher concentration of CH4, the molar proportion of CO decreases because natural gas combustion is more efficient, i.e. complete combustion. The second and third case simulations demonstrate a higher CO content within the furnace, indicating that the combustion is incomplete and that a more detailed analysis with further air and flow adjustments is needed to improve the process' efficiency. The next step in this research will be to conduct a mesh analysis to establish the suitable size and alter the air surplus to improve the efficiency of combustion using gas mixes. It will also look into the effect of employing these gases on the efficiency of the furnace.

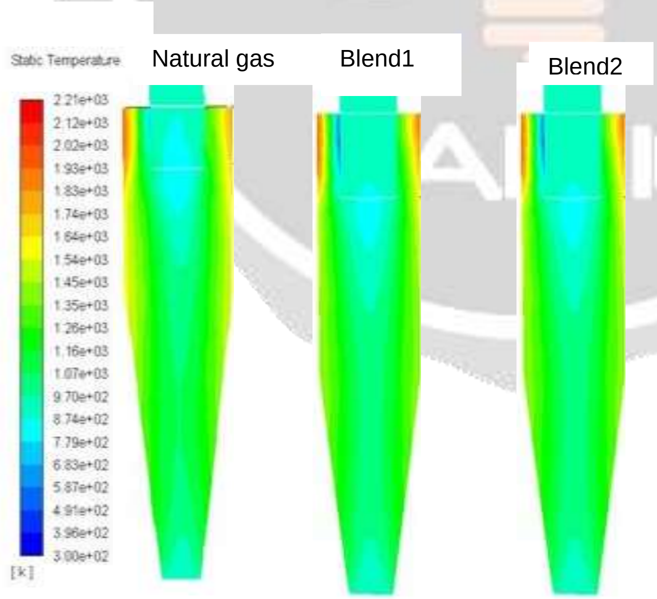

Figure 3: Temperature profiles inside the furnace obtained from 3 simulation cases

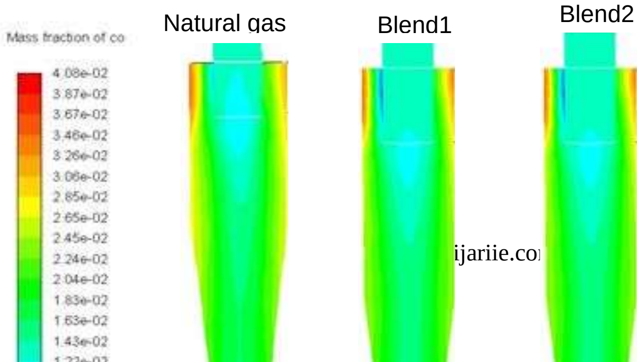

Figure 4: Profile of CO mass fraction inside the furnace obtained from 3 simulation cases

5. CONCLUSIONS

Utilizing three-dimensional simulation in CFD, the effect of using a mixture of gases with variable composition in the combustion process of industrial furnaces was investigated. Temperature and CO mass fraction profiles were taken inside the furnace. The simulation with less natural gas shows less heat dispersion, i.e. fewer hot zones in the furnace's central zone, and the CO mass fraction distribution also reduces, indicating that combustion is complete. However, with mixes 1 and 2, the combustion is incomplete, resulting in the production of additional CO; the CO mass fraction increased from 1.02e-02 to 2.85e-02, showing that the combustion is inefficient and has to be modified. As a result, when using mixtures of gases with varying compositions, a more thorough assessment of air excess and fuel flow is required. The next step in this investigation will be to test a more complex geometry (cabintype furnace), a grid independence research, NOx emissions analysis (post-process), and data validation to see which ranges of refinery gas calorific power work best in furnaces.

ACKNOWLEDGEMENT

The authors would like to acknowledge the centre for energy resources and refining technology at Imo State University, Owerri, Nigeria for their technical support, especially providing access to their own software application.

REFERENCES

Aminian J., Shahhosseini SH., Bayat M., 2010, Investigation of temperature and flow fields in an alternative design of industrial cracking furnaces using CFD. Iranian Journal of Chemical Engineering, 7(3), 61–73.

Cala, O. M., Meriño, L., Kafarov, V., Saavedra, J., 2013, Effect of the composition of the refinery gas on the characteristics of the combustion process. Revista Ingenierías Universidad de Medellín; 12, (23), 101-112.

Cala Parra, OM., 2014, Determination of an energy efficiency index for combustion processes of gas mixtures in refinery, M.sc Thesis. Universidad Industrial de Santander, Bucaramanga, Colombia. (In Spanish).

Díaz-Mateus, F.A., Castro-Gualdrón, J.A., 2011, CFD Technique to Calculate Tube Skin Peak Temperatures in Refinery Furnaces. CT&F - Ciencia, Tecnología y Futuro, 4, 73–88.

Gentile, G., Debiagi, P., Cuoci, A., Frassoldati, A., Faravelli, T., Ranzi, E., 2016, A CFD model for biomass flame-combustion analysis, Chemical Engineering Transactions. 50, 49-54

Li, X., Zhang, L., Sun, Y., Jiang, B., Li, X., Wang, J., 2015, Numerical simulation of the flue gas side of refining vacuum furnace using CFD. Chemical Engineering Science, 123, 70–80.

Stefanidis, G. D., Merci, B., Heynderickx, G. J., Marin, G. B., 2006, CFD Simulations of Steam Cracking Furnaces Using Detailed Combustion Mechanisms. Computers & Chemical Engineering, 30(4), 635–649.

Yang, J., Tai, N., Wang, L., Xiao, J., Yang, C., 2012, Numerical Simulation of the Flue Gas and Process Side of Coking Furnaces. Industrial & Engineering Chemistry Research, 51(47), 15440–15447.