CFD PARAMETRIC DESIGN SIMULATION OF COMBUSTION SEQUENCE IN GAS HEAT ELEMENT

Engr. Nnadikwe Johnson[1], Engr. Ewelike Asterius Dozie[2], Ibe Raymond Obinna[3], Okereke Godson[4]

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

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

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

[4] *Lecturer in department of Building, Federal University of Technology Owerri, Imo State, Nigeria*

ABSTRACT

Through an effective and reasonable design of the heat element structure, the combustion impact of natural gas-air mixed combustion is optimized, resulting in a reduction of atmospheric pollutants and energy savings and emission reductions. This article begins with a geometric model of the heat element and then utilizes Computational Fluid Dynamics (CFD) technology to numerically simulate the heat element's combustion sequence, which results in the heat element's combustion sequence after structural optimization. The blower guide vanes have a negligible guiding impact on the airflow, but the mounting body of the guide vanes has a substantial back pressure effect, which limits the primary air volume and prolongs the combustion history in the flame's center area; The temperature field is well spread, and the combustion is well dispersed; the CH_4 and CO_2 in the furnace chamber are essentially burnt away, and the NO concentration in the furnace chamber is around 800 ppm, which fulfills the emission requirement and has a good combustion effect. This article serves as a guide for future research on heat element structure.

Keywords: *CFD, CH_4, CO_2, Combustion effects, Simulate*

---

1 INTRODUCTION

In recent years, the usage of gas heat components as a supplement to mixed equipment has exploded. The term "gas heat element" refers to a collection of devices that allow natural gas and air to be expelled in a certain direction and combined for combustion[1]. The flue gas generated by the heat element's combustion includes the air pollutant NOx. Due to the stringent energy conservation and emission reduction policies, the requirements for NOx emission limitations in industrial flue gas are increasing. Thereby, by designing the heat element structure effectively and rationally, the proportionality and uniformity of the natural gas and air sequences are optimized to maximize the combustion impact of the heat element and thus minimize NOx emissions[2]. With the fast advancement of computer technology and numerical computing, an increasing number of academics are using Computational Fluid Dynamics (CFD) technology to perform numerical simulations and analytical research on the combustion sequence and combustion impacts of gas heat components. FLUENT is a widely used commercial CFD software package that includes extensive physical models, innovative numerical techniques, and robust pre- and post-processing capabilities. It is extensively used in the area of numerical modeling of combustors[3,4].

Sheng Chen et al.[5]present an enhanced low-NOx method based on a fuel-segmented natural gas heat element. The heat element's modeling results are analyzed using the commercial CFD program FLUENT, and it is found that the enhanced folded heat element can reliably lower the NOx content by a substantial amount. Antonio Andreini et al.[6]conducted a numerical study of a premixed heat element with a cross-flow jet structure in a premixed injection system in order to achieve homogenous fuel and air mixing. A.Andreini et al.[7] quantitatively analyzed the internal premixing process of the combustor using the Reynolds Averaged Navier-Stokes (RANS) and CFD methods, changing the fuel injection criteria and fuel splitting to minimize exhaust gas NOx emissions. Zakaria Mansouri et al.[8] used RANS calculations, Delayed-Detached Eddy Simulation (DDES), and experimental observations to investigate the non-premixed flame of the heat element cyclone quantitatively. Yichao Li et al.[9] investigated the combustion performance of slotted and unslotted cyclones in the heat element using CO_2 emission concentration measurements and a numerical analysis, concluding that slotted cyclones have the ability to increase the combustion chamber's combustion performance. Other authors have used computational fluid dynamics to simulate the flow within heat elements that are burning gaseous fuels, resulting in numerical assessments of the combustion consequences of various heat element configurations[10].

According to the findings of eminent experts, the construction of an efficient and suitable gas heat element plays a critical function in the combustion sequence and low NOx exhaust emissions[11]. This article presents a gas multi-level heat element and performs a numerical investigation of its combustion sequence and impact. The values of airflow sequence, fuel uniformity, and temperature field sequence are investigated using FLUENT software simulation to determine the influence of its combustion, and the study findings provide a reference value for the combustion sequence of gas heat elements.

2 NUMERICAL SIMULATION METHODS

2.1 Heat element model

A blower, a gas pipe, a flame holder, and a furnace chamber comprise the gas heat element. Figure 1 depicts a three-dimensional schematic of the heat element. The blower is equipped with unique deflector blades that impart an acceleration effect on the airflow; the blower's schematic design is depicted in Figure 2. The flame holder is composed of a two-stage cyclone blade that is mounted at a 30 degree angle, as seen in Figure 3. In the flame holder, primary air is supplied to the first stage cyclone blade, secondary air is supplied to the second stage cyclone blade, and the air flow outside the revolving flow outer ring is supplied with tertiary air.

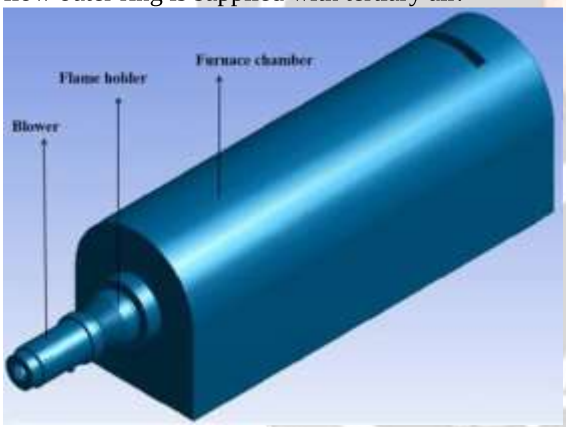

Figure 1. Three-dimensional diagram of the gas heat element

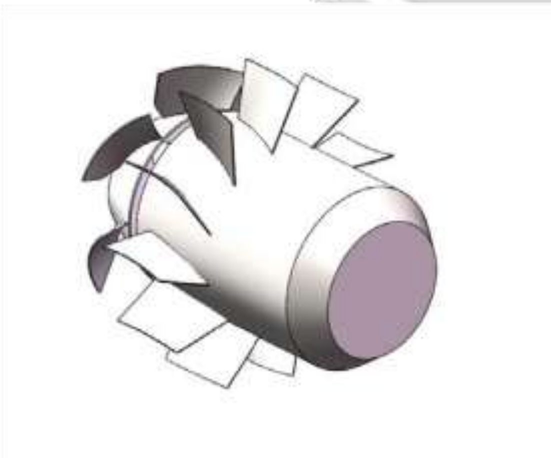

Figure 2. Structure of a blower guide vanes

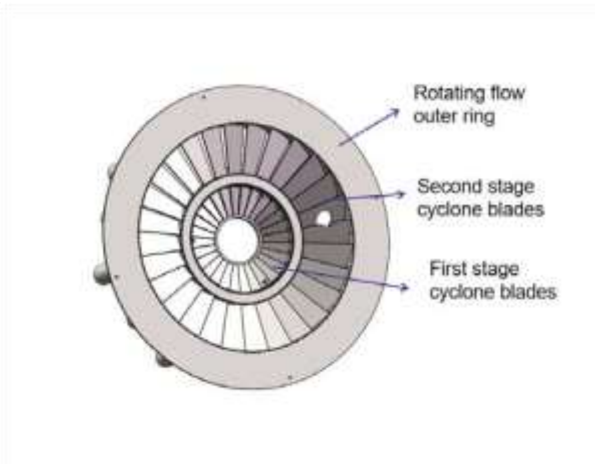

Figure 3. Structure of a flame holder

2.2 Calculation grids

Figure 4 shows the 9.04 million total mesh size, with a minimum mesh size of 1mm, Element Quality of grids better than 0.2, Skewness is less than 0.85, Orthogonal Quality is more than 0.2, and the Aspect Ratio is larger than 1.6, due to the geometric model being irregularly shaped. There are three elements to this numerical calculation: the hood air, the hood gas (CH4) and the furnace chamber (FurnacePart) sections.

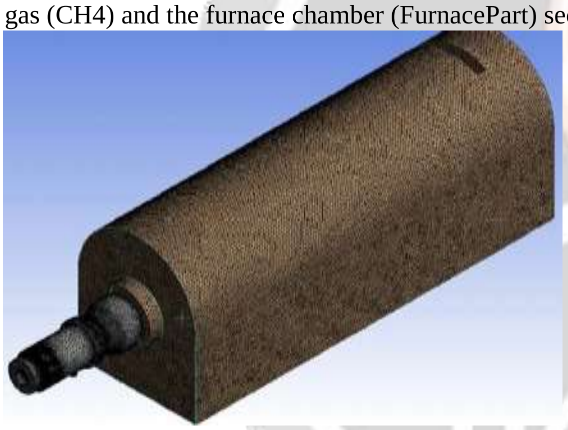

Figure 4. 3D mesh of the computational domain of the gas heat element

2.3 Boundary conditions

Pressure in the furnace chamber is 65 psi (negative pressure), with a heat element load of 50% and a gas and air flow rate of 1575 and 2165.25 m3/h, respectively. The fan speed is 1522 r/min. The input flow rate and cross-sectional area of the air inlet and methane inlet are used to compute the boundary conditions, with an air inlet velocity of 18.92 m/s and a methane inlet velocity of 28 m/s.

2.4 Calculation processes

The cold flow field in the fluid domain is calculated in the first step. A weighted average mass is used to calculate the outlet pressure field, which is then used in conjunction with the hot flow field, which is set to -65 Pa, to ensure that the volumetric reaction calculation for methane burns completely in the large blade area before the third part of the radiation field, which is based on a surface weighted average mass, is performed.

3. RESULT OF SIMULATION AND CONCLUSION

Using numerical simulations, we were able to explore the combustion temperature sequence and combustion impact of this heat element, as well as the influence of each portion of the heat element.

3.1 An Analysis of Blower Guiding Vanes and Airflow Sequence

The guiding vanes on the blower have a unique structure, and since they are equally distributed around the circle, the airflow created by the blower helps to promote combustion. It can be seen in Figure 5 that there is a significant influence on airflow sequence from the guide vanes of the pressure surface and suction surface pressure difference of about 150 Pa, which results in an acceleration effect on airflow in the suction surface. Figure 5 shows the blower pressure sequence diagram.

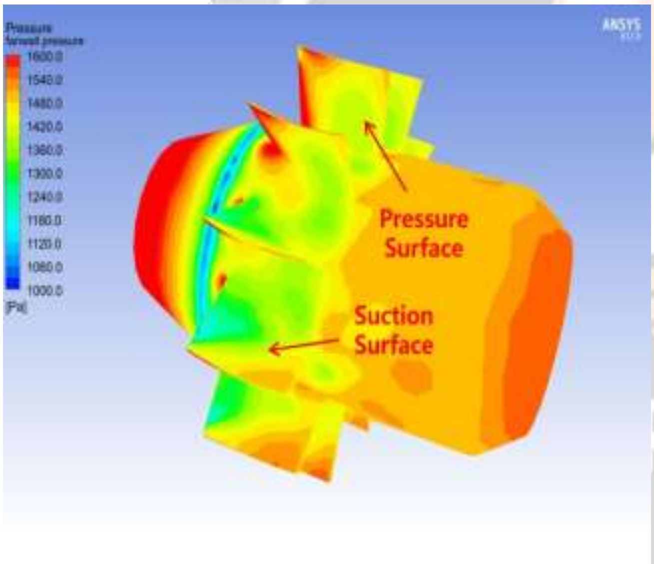

Figure 5. Blower pressure sequence

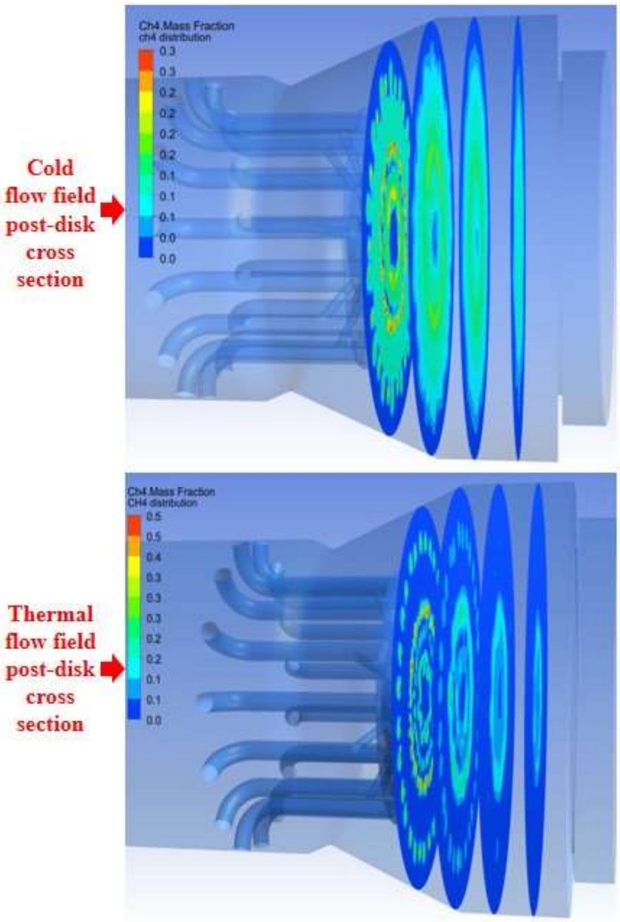

Figure 6. CH_4 sequence

3.2 The use of layered and uniformly spaced spray holes to measure the regularity of the fuel sequence

Using a layering and even sequence of injection holes, this heat element analyzes fuel sequence uniformity. The placement of the gas injection holes has an important impact on fuel sequence uniformity. Figures 6 and 7 show a cross-section behind the flame holder and a Y-section of the cold and hot flow fields, respectively, to compare the CH4 sequence analysis. The findings suggest that the cold flow methane is more evenly distributed.

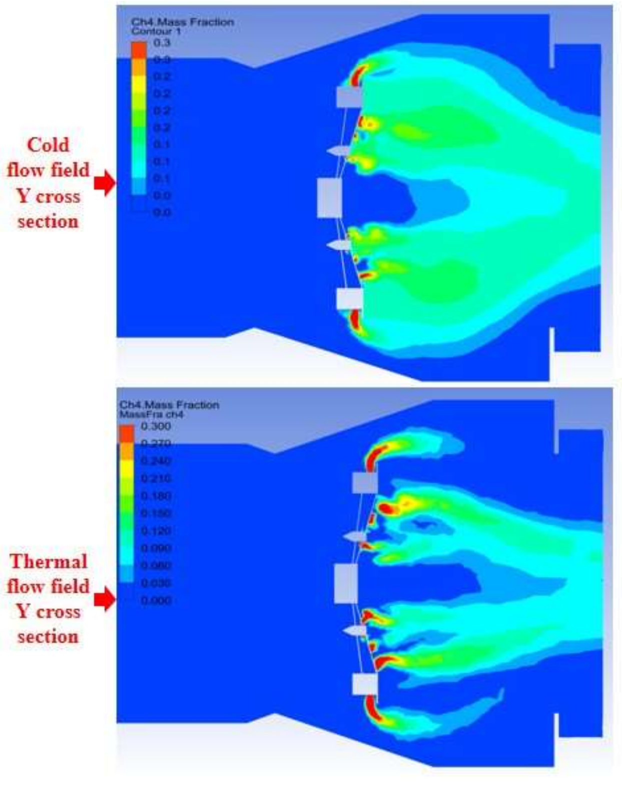

Figure 7. CH_4 sequence

3.3 cyclonic vane return flow strength analysis

Total air volume is estimated to be 17800 m3/h, with main air volume being 2867.8 m3/h, according to the simulation results. A total of 16.1 percent of the total air volume comes from primary air volume, which is illustrated in Figure 8 as a forward-moving stream of air with no return flow in the vicinity of the flame holder. The channel's constriction causes radial convection as the airflow diffuses away from the disc's surface in an axial direction.

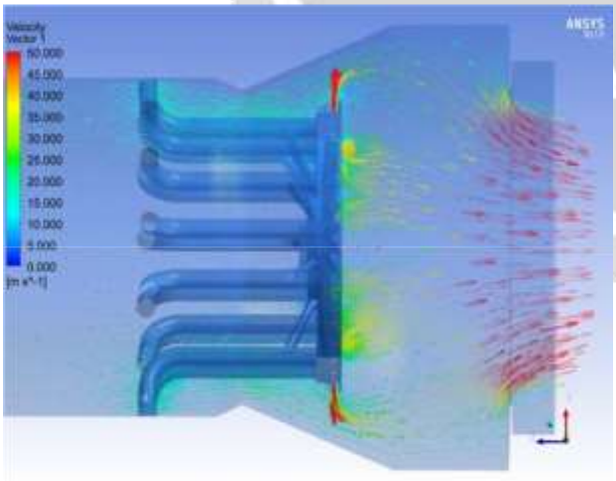

Figure 8. Rotational velocity vector

3.4 A study of the temperature distribution in the environment

If the temperature field is "high above and low below," "low above and high below," or any other uneven outcomes, the high temperature region will stimulate the generation of NOx, resulting in a large NOx emission concentration at the furnace's exit. An inefficient combustion process will arise from this, as well as a change in combustion and other issues. In the perspective view given in Figure 9, the temperature sequence around the whole flame holder is uniform, with the temperature range generally between 1500K and 1800K, and the combustion sequence is uniform, minimizing the NOx concentration in the high temperature zone.

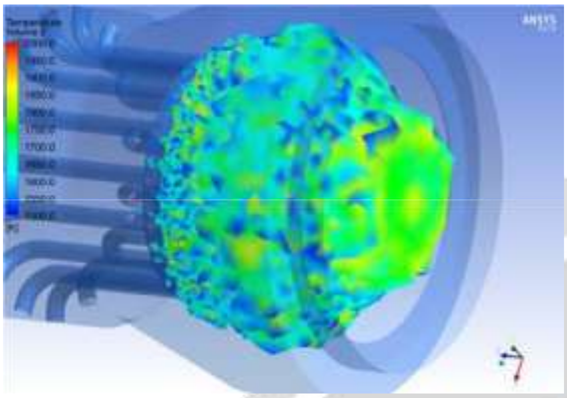

Figure 9. Perspective view of the sequence behind the flame holder

3.5 Combustion field products in order of appearance

Gas and air have little opportunity to mix properly, resulting in inefficient combustion that wastes energy and pollutes the environment while also producing a flame temperature that is too low for use in high-temperature industrial environments. An investigation into a heat element that is safe, burns completely, has a high combustion efficiency and a low NOx emission level is particularly crucial in light of these facts Because of this, we do numerical calculations to assess if the combustion is appropriate and effective in terms of residual fuel and the sequence of combustion products.. This is demonstrated in Figures 10 and 11, as well as Figure 12, which shows the distribution of NO in the heat element chamber, and the sequence of CH4 and CO as indicated in Figure 10 and Figure 11.

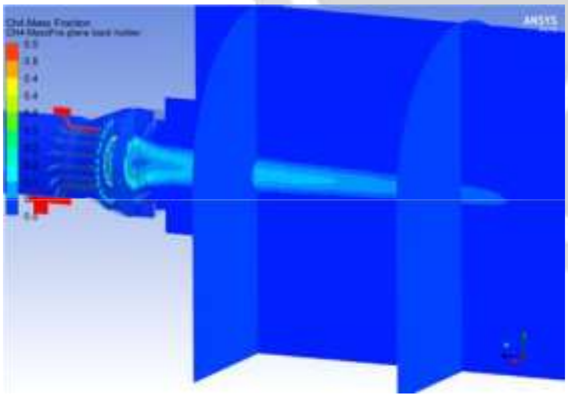

Figure 10. CH_4 sequence of cross sections in X=0

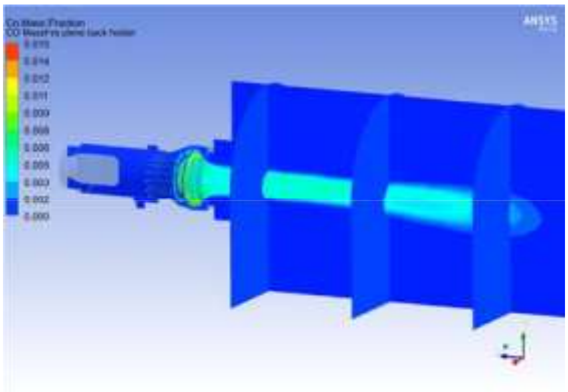

Figure 11. CO sequence of cross sections in X=0

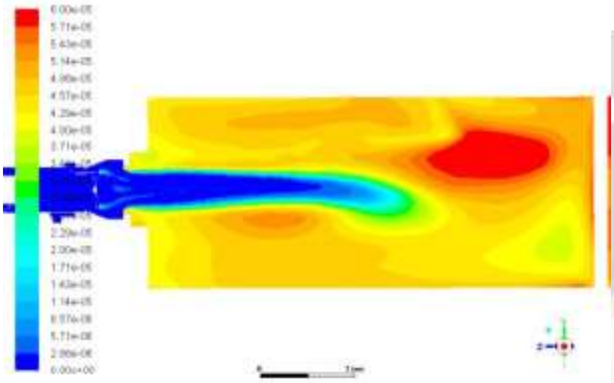

Figure 12. NO sequence of cross sections in X=0

4 CONCLUSION

A closer look at the impact analysis chart reveals that the blower guide vanes have a poor airflow guiding function, and the guide vanes put in the body back pressure have a considerable influence on the main air volume, reducing it by around 16.1 percent of the total air volume.

Aside from that, as can be seen from its structure (the flame holder), the small and large blades each have gas spray holes on both the inside and outside, and the gas diffuses in the axial direction. The primary wind speed is reduced, so that fuel mixing time in the central area of these small blades is extended, allowing for a longer combustion history in the flame's core.

A satisfactory combustion sequence may be deduced from the circumferential temperature sequence of the flame holder, which shows an equal temperature distribution over the flame holder's big blade and outer edge areas of around 12001600 degrees Celsius. The furnace chamber's CH_4 and CO have been completely burned off, and the NO concentration is about 800ppm, or 32mg/m3, which passes the emission requirement and has a favorable combustion effect.

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

1. Iguo Zhou, Junying Tang, et al. Design optimization and CFD evaluation of a volute swirl heat element with Steven R Mickey, Martin G Schönfelder, Joachim G Wünning. Direct-Fired Gas Heat elements for High Temperature Applications[J]. Industrial Heating, 2019, 87(10).

2. Murat Umut YANGAZ, Gökçen Alev ÇFTÇOLU, Mehmet Arif Neet KADIRGAN. Effects of Different Applications and Designs on Emission Reduction in a Gas Heat element. 2018, :218-226.

3. [1]Yunfei Jia, Wecentral gas supply[J]. Journal of the Brazilian Society of Mechanical Sciences and Engineering, 2020, 42(1–2).

4. [1]Minsung Choi, Yonmo Sung, Myungjun Won, et al. Effect of fuel sequence on turbulence and combustion characteristics of a micro gas turbine combustor[J]. Journal of Industrial and Engineering Chemistry, 2016.

5. Sheng Chen, Yuming Xing, Aojin Li. CFD investigation on Low-NOx strategy of folded flame pattern based on fuel-staging natural gas heat element[J]. Applied Thermal Engineering, 2017, 112.

6. Antonio Andreini, Bruno Facchini, et al. Numerical Analysis of a Low NOx Partially Premixed Heat element for Industrial Gas Turbine Applications[J]. Energy Procedia, 2014, 45.

7. A. Andreini, M. Cerutti, B. Facchini, A. Innocenti. CFD Analysis of NOx Emissions of a Natural Gas Lean Premixed Heat element for Heavy Duty Gas Turbine[J]. Energy Procedia, 2015, 81.

8. Zakaria Mansouri, Toufik Boushaki. Experimental and numerical investigation of turbulent isothermal and reacting flows in a non-premixed swirl heat element[J]. International Journal of Heat and Fluid Flow, 2018, 72.

9. Yichao Li, Renfu Li, Dinggen Li, et al. Combustion characteristics of a slotted swirl combustor: An experimental test and numerical validation[J]. International Communications in Heat and Mass Transfer, 2015, 66.

10. K. Khanafer, S.M. Aithal. Fluid-dynamic and NOx computation in swirl heat elements[J]. International Journal of Heat and Mass Transfer, 2011, 54(23).

11. Xuren Zhu, Renfu Li, Dinggen Li, et al. Experimental study and RANS calculation on velocity and temperature of a kerosene-fueled swirl laboratory combustor with and without centerbody air injection[J]. International Journal of Heat and Mass Transfer, 2015, 89.

12. Aleksandra Kiedrzyska, Robert Lewtak, et al. Numerical study of natural gas and low-calorific syngas co-firing in a pilot scale heat element[J]. Energy, 2020, 211.