# CFD-modeling of gas combustion process in the industrial cyclone-calciner furnace

Roman Benko[1], Roman Havryliv[2], Volodymyr Maystruk[3]

[1]Department of chemical engineering, Lviv Polytechnic National University, UKRAINE, Lviv, St. Yura square, 9, E-mail: 96tej96@gmail.com

[2]Department of chemical engineering, Lviv Polytechnic National University, UKRAINE, Lviv, St. Yura square, 9, E-mail:romanhavriliv@gmail.com

[3]Department of design and operatin of machines, National University, UKRAINE, Lviv, Profesorska str., 1, E-mail:vmaistruk@gmail.com

***Abstract* – The computer model that enables to investigate the work of cyclone-calciner furnace for the heating up regime has been developed. Simulation carried out in ANSYS FLUENT 15.0. The temperature field and species concentration were obtained. The work results provide important data to further study the efficiency of the limestone thermal decomposition chemical reactions and optimizing furnace design and operation modes.**

Key words – CFD-modeling, cyclone-calciner furnace, heating up regime of furnace

## I. Introduction

High temperature processes are the basis for a number of industrial productions in chemical technology. This is especially true for limestone calcination processes to lime production. This process occurs in industrial furnaces of various designs. One type of these furnaces is the cyclone-calciner furnace. Furnaces of this type are universal and used in many industrial processes for processing and preparation of dispersed materials.

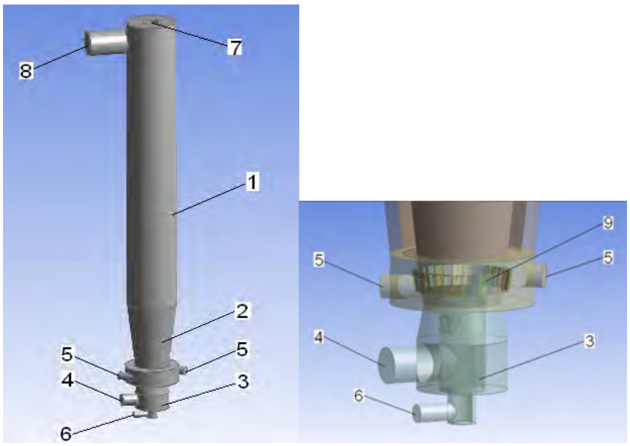
Exploitation of these manufacturing facilities are linked with the consumption of large amount of fuel for the thermal decomposition chemical reactions of limestone.

In addition, complex physical and chemical processes such as movement and heating of dispersed particles, combustion of fuel and interaction between particles and gas flows take place in the apparatus. Understanding these processes will make it possible to control the furnace and improve its efficiency, reduce fuel consumption and optimize modes of operation.

The most effective way to study the processes that take place in the cyclone calciner furnace and efficiency of work is the use of numerical simulations that can significantly reduce the costs of experimental and industrial research.

## II. Boundary conditions for numerical simulation

For numerical simulation of combustion process fuel in calciner furnace the industrial testing data were used. Geometric dimensions and design of the furnace were described in [1]. The modeled cyclone-calciner furnace is shown in Fig. 1, a, b. In this type of furnace, the burner "ГГВ-МГП-200" is used.




a b

Fig. 1. Cyclone-calciner furnace geometry:
a - general view of furnace; b - burner location
1 - cylindrical section; 2 - conical section; 3 - burner; 4 - air inlet pipe; 5 - additional air inlet pipe; 6 - fuel (methane) pipe; 7 - inlet solids pipe; 8 - outlet pipe; 9-blades

Operation modes of the furnace is presented in table 1. All the geometric data and the initial and boundary conditions were supplied by "Pustomyty lime plant", Ukraine.

TABLE 1

OPERATION MODES OF THE FURNACE

| Operation modes | Main regime | Additional air flow |
|---|---|---|
| Inlet air, m/s | 17-22 | 13-20 |
| Inlet gas, m/s | 7 | 7 |
| Inlet additional air, m/s | 0 | 5 |
| Temperature wall, K | 1000 | 1000 |

Numerical simulation of combustion was carried out using software complex Ansys Fluent 15. The computational mesh was built using mesh generation ANSYS Meshing CFD and consisted near the 2 mln. cell. The qualities of the created mesh had checked to avoid error from occur.

For the CFD modeling the Realizable k-e turbulence model, the Discrete Ordinates (DO) radiation model and Non-Premixed combustion model were used. In the numerical procedure the 3-D, steady-state, Navier–Stokes equations, energy equation and single mixture fraction probability function (PDF) were used.

The fuel composition in mole fractions of the species $CH_4$, $C_2H_6$, $C_3H_8$, $C_4H_{10}$, and $CO_2$ is presented in table 2. For all inlet flows temperature was set as 300K.

TABLE 2

THE FUEL COMPOSITION

| Species | Mole fraction |
|---|---|
| $CH_4$ | 0.965 |
| $C_2H_6$ | 0.017 |
| $C_3H_8$ | 0.001 |
| $C_4H_{10}$ | 0.001 |
| $CO_2$ | 0.003 |
| $N_2$ | 0.013 |

The oxidizer (air) consists of 21%$O_2$ and 79% $N_2$ by volume.

To setting up the calculation module used the following assumptions:

1. The working volume of the furnace is a mixture of gases (oxygen, nitrogen, fuel, waste products);
2. The heat exchange between the environment and the wall is missing (the walls are “adiabatic”)
3. The thermal radiation from the wall in the volume of gas simulated by the model of discrete ordinate at a constant temperature of 1000 K.

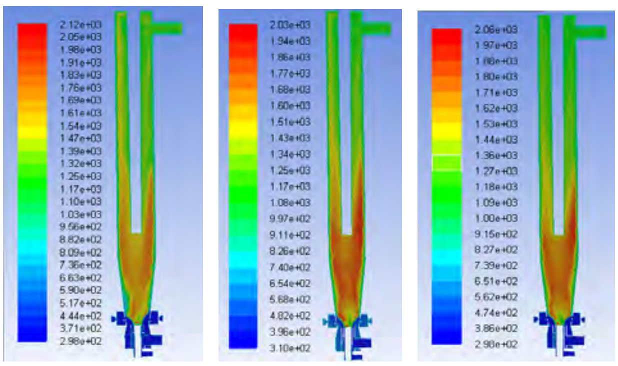
A finite volume method was chosen since most of fuel combustion simulations applied this type of method. SIMPLE-based approach was used for pressure-velocity coupling scheme. The solution was simulated until convergence is achieved. Approximately 3500 iterations was performed for each cases.

## III. Results modeling

Simulation of the furnace was carried out for various technological modes, which correspond to different ratio of the primary and secondary air flow and fuel (gas methane) and described in [1].

The thermal fields results from the numerical modeling are demonstrated in Figs. 2 and 3, respectively. Figures displays the thermal fields according to the two operating conditions: case 1 - with different ratio of main air flow and gas flow; case 2 - with different ratio of main and additional air flow and gas flow.

From Fig. 3, the maximum temperature ranges from 1700K to 2000K and the temperature color scale is the same for all three cases.



a b c

Fig. 2. The temperature field for a vertical plane (case 1)
a) inlet air 17 m/s; inlet gas 7 m/s;
b) inlet air 20 m/s; inlet gas 7 m/s;
c) inlet air 22 m/s; inlet gas 7 m/s

When the gas and air enter the furnace and mixed in burner, combustion was occurs due to high velocity.

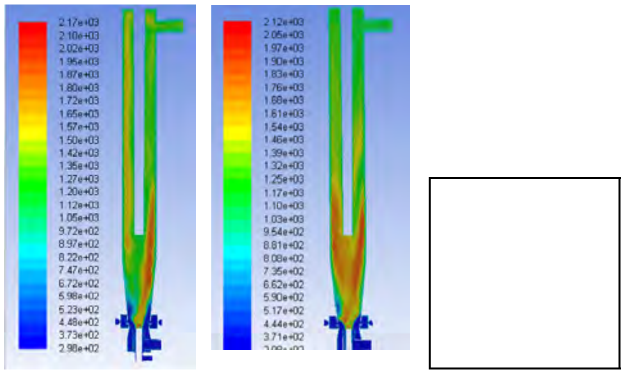
The central vortex created by the tangential blades of the burners is visible at the center of the conical section furnace.

One can notice that the highest temperature is located between the central pipe and wall of the furnace in conical section, were the stoichiometric condition is obtained. After that, the temperature was increased until reach maximum temperature.

As the flow moves to the outlet heat is exchanged with the furnace walls and central tube, creating the temperature gradient shown in the Figs. 2 and 3.

At the higher levels in cylindrical section of the furnace general temperature distribution shows lower values than at the lower levels. This is important since the particles calcination time decreases with non-uniform temperature distribution.

Fig.3 showing the effect of varying the amount of combustion air flowing to the burner according to case 2 (with additional air flow).



a b c

Fig. 3. The temperature field for a vertical plane (case 2)
a) inlet air 13 m/s; inlet gas 7 m/s; inlet additional air 5m/s;
b) inlet air 17 m/s; inlet gas 7 m/s; inlet additional air 5m/s;
c) inlet air 20 m/s; inlet gas 7 m/s; inlet additional air 5m/s

For case 2a (Fig.3.a) additional air flow velocity is low, so the peak flame outside of the conical section, meaning the combustion is poor in the furnace. In 2b and 2c cases (Figs.2b and 2c), the combustion is improved by using the additional air and the combustion flame is kept in the main combustion chamber.

One can conclude that the the high ratio of main air flow and additional air flow to the burner provides more uniform temperature field. For this cases the areas of local overheating are absent, and the maximum temperature in the working area is 2000K.

The flame is longer and characterised by a slightly higher temperature than that predicted for the air-fired case.

Probably the most efficient calcination of limestone can be arranged on the circuit with additional air flow. The answer to this question will be obtained after the particles calcination process simulation in the furnace according to cases 2b and 2c.

## Conclusion

The combustion modeling provides some promising results to better understand the complex processes occurring within the furnace and can be a baseline for future modeling of pollutant emissions and limestone calcination processes.

The results make it possible to evaluate 3-D field of temperatures in the reaction zone and in the whole system volume, concentration species distribution, determine the length of the torch and choose the best operation mode for heating regime of furnace.

The experimental data are in good agreement with the field tests results of the device.

## References

[1] R. Havryliv, V. Maystruk, V. Biliak, "Development of a numerical model for gas-solid flow in the industrial cyclone-caciner furnace, EEJET., vol. 3/8 (75), pp.14-21, 2015.