Numerical Simulation and Experimental Analysis of an Oxygen-Enriched Combustion Fiberglass Furnace

Zongming Liu, Guangbin Duan, Liang Li, Weixiang Wu, Xiaobin Liu
School of materials science and engineering,University of Jinan, Jinan, China
Ost_liuzm@ujn.edu.cn

Abstract: This paper established a three-dimensional mathematical model of oxygen-fuel combustion space and glass flowing, according to the practical situation of a fiberglass furnace whose production was up to 20,000 ton per year, and obtained simulation results by using UDF program to combine combustion space and glass flowing through the one-way couple. Based on the comparison between the simulation results and the measured data obtained, it can be suggested that the mathematical model can objectively reflect the real distribution of the temperature field and velocity field of the oxygen-enriched combustion fiberglass furnace. Results indicated that the maximum relative error was less than 2%. The study here was really useful for understanding the furnace, improving the working conditions, reducing the risks, and optimizing the designs of glass furnace.

Keywords: numerical simulation; oxygen-enriched combustion; glass furnace; glass flowing; one-way couple

1 Introduction

Simulation of industrial combustion system using computational fluid dynamics (CFD) modeling was still a challenging domain [1,2]. Besides, the research on fundamentals to understand the processes occurring in reactive flow systems, much work had been carried out to develop computational methods suitable for coupling the many important aspects of chemistry and physics in a way that is efficient enough for solving industrial problems.

Introduction of advanced processors and computational algorithms had enabled the three dimensional simulation of the glass furnace with more accurate treating of physical and chemical phenomena. The natural convection vortices of the glass tank, gas-fired and the turbulence and the chemical reactions of the combustion space were all simulated in recent studies [3,4].

Generally, the furnace was divided into the combustion space, glass tank and batch blanket. In order to simple model, this study assumed that there were only steady melting glass metal in the glass tank. However the combustion space was simulated separately. The main contribution of this work was the introduction of a new method that was used to couple the combustion space to the glass following model by using UDF program.

In this study the average temperature distribution of glass metal surface were obtained through the simulation results of combustion space firstly, and then the results were regarded as the boundary conditions of glass flowing model by UDF program. Finally, the simulation results of glass fluid flow and heat transfer, completed the one-way couple between combustion space and glass flowing were achieved.

2 Models

2.1 Combustion Space Model

2.1.1 Turbulence Model

The turbulent nature of the combustion gases was simulated by applying standard k-ε model. This model composed of mass conservation equation, momentum conservation equation, and energy conservation equation.

2.1.2 Combustion Model

In this work, by using premixed methane-air combustion reaction [5] and adjustment the air species mass fractions in which the oxygen accounts for 90%, the nitrogen accounts for 10%. The density of the mixture was defined as ideal gas law. The viscosity, the thermal conductivity

Financed by Natural Science Foundation of Shandong Province (NO. ZR2010EM064)

and the specific heat of mixture were defined as a mass fraction average of pure species.

2.1.3 Radiation model

The radiation exchange of hot gases was modeled using the discrete ordinates radiation model (DORM). In this model imaginary spheres were considered around each cell.

2.2 Glass Metal Model

The conservative equations included mass, momentum and energy equations. The molten flow was dominated by strong free convection cells created by severe temperature differences on the melt surface. The back flow in the glass tank was due to the low velocity and high viscosity of melt. The effect of radiation was considered by an effective thermal conductivity. In this work, the radiation was modeled using the discrete ordinates radiation model too.

2.3 Thermal Boundary Model of the Glass Furnace

2.3.1 Thermal boundary calculation model of the furnace wall

The breast wall and the crown were defined as the wall, and thermal boundary conditions were defined as fixed heat flux. The heat flux formula of the furnace wall was:

$$Q = \alpha_1(t_1 - t_2) + \varepsilon \cdot C_0\left[(\frac{T_1}{100})^4 - (\frac{T_2}{100})^4\right] = \alpha_2 \cdot (t_1 - t_2) \quad (1)$$

$$\alpha_2 = \alpha_1 + \frac{\varepsilon \cdot C_0}{t_1 - t_2}\cdot\left[(\frac{T_1}{100})^4 - (\frac{T_2}{100})^4\right] = 2.2\sqrt[4]{t_1 - t_2} + \frac{\varepsilon \cdot C_0}{t_1 - t_2}\cdot\left[(\frac{T_1}{100})^4 - (\frac{T_2}{100})^4\right] \quad (2)$$

In the formula, α_1—the outside wall and the ambient air convection heat transfer coefficient. ε—the outside wall of blackness. T_1—the outside wall temperature. T_2—the ambient air temperature. α_2—the equivalent heat transfer coefficient between the outside wall and the ambient air.

This process was conducted outside the wall and the air was the natural convection heat transfer, so: $\alpha_1 = 2.2\sqrt[4]{t_1 - t_2}$ The heat flux of crown was 1.5 times more than the furnace wall.

Among them, the modification of crown, sides of the parapet, and the outer surface of glass tank wall on both sides of the average temperature and ambient temperature were given in Table1.

Table1 Temperature of outside wall and ambient

position	Temperature of outside wall (t_1)	Ambient temperature (t_2)
Crown	453K	300 K
Breast wall	411 K	
Wall of glass tank	400 K	
Bottom of glass tank	389 K	

2.3.2 Thermal boundary calculation model of the glass metal surface

In this study, the glass metal surface was defined as the wall and thermal boundary conditions were defined as radiation and convection combined. Glass surface to receive the total energy include: Q_{gm}^R—the flue gas radiative heat transfer to the glass surface. Q_{gm}^R—the inside wall radiative heat transfer to the glass surface. Q_{gm}^K—the flue gas to convection heat transfer to the glass surface.

$$Q_m = Q_{gm}^R + Q_{gm}^K + Q_{wm}^R \quad (3)$$

In the furnace wall on the one hand to accept the heat, on the other hand to release the heat, played the role of heat transfer medium. Using this heat balance of furnace wall, eliminate the inside wall temperature T_w. Unit area on the glass surface by heat was calculated as:

$$Q_m = \varepsilon_{gwm} C_0\left[(\frac{T_g}{100})^4 - (\frac{T_m}{100})^4\right] + \alpha_{gm}^K(T_g - T_m) \quad (4)$$

In the formula, ε_{gwm}—the total relative blackness of the flue gas, furnace wall and the glass surface. C_0—the black body radiation coefficient. T_g—the average flue

978-1-935068-37-2 © 2010 SciRes.

gas temperature. φ—angular coefficient. T_m—the average temperature of the glass surface. α_{gm}^{K}—the convection heat transfer coefficient between the flue gas and glass surface.

For solving the model, using the Fluent software provides a user interface UDF (User-Defined Function), by VC + + program, the program completed and the UDF into Fluent module internal data exchange. UDF can be addressed include: the boundary conditions, modify the source item; the definition of material properties, and so on. This work dealt with the UDF function of boundary conditions.

3 Results and Discussion

3.1 Furnace and Operating Conditions

The fiberglass furnace with a capacity of 20,000 ton/year was simulated. The glass tank was 14 m long, 4 m wide and 1 m deep. The combustion space had an arc shape crown with the heights of 0.6 m. It was 2.25 m between the first pair of gun and back-wall, and 2.5 m between the third and the forth pair, other gun spacing for 1.65 m. The structure of this furnace size was shown in Figure 1 and Figure 2.

Based on the actual size of furnace, the geometric model and mesh creation of glass furnace was set up by using the software GAMBIT in former course of treatment. Using the grid increase technology in the spray gun, it can led the simulation result to be more accurate. The map of three-dimensional mesh encryption in spray gun was shown in Figure 3.

At fuel ports, the gas velocity was given such that the flow rate of ports 1–6 was 2.23, 4.81, 5, 5, 3.61 and 3.33 m/s, respectively. The temperature of premixed gas was taken 650 K and the backflow total temperature was taken 2000 K.

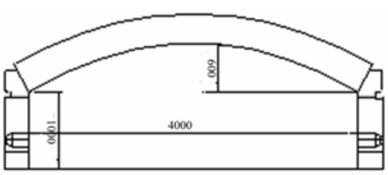

Figure.1 Profile of combustion space

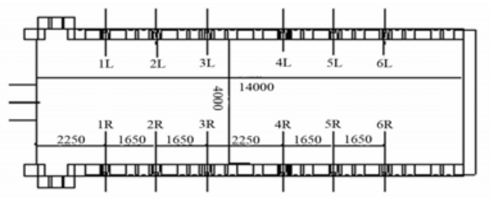

Figure.2 Vertical view of combustion space

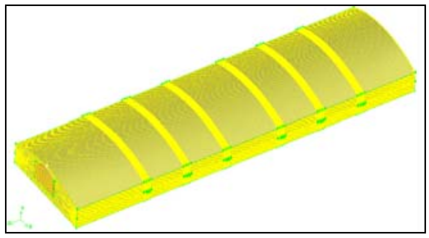

Figure.3 Map of three-dimensional mesh encryption in spray gun

3.2 Simulation Results

3.2.1 Analysis of Simulated and Measured Result

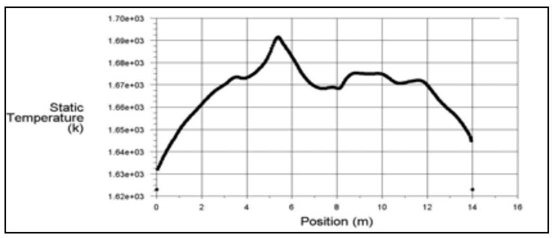

Figure.4 Simulated temperature distribution of center line in the crown.

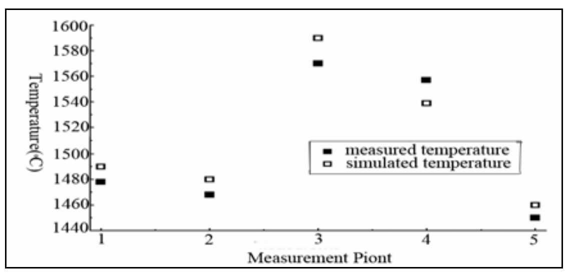

Figure.5 Relationship between measured and simulated value of crown

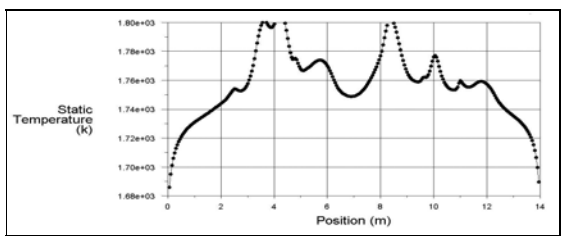

Figure.6 Simulated temperature distribution of center line in glass surface

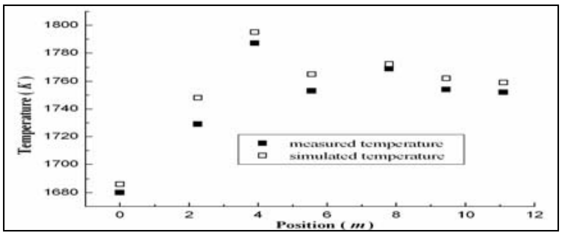

Figure.7 Relationship between measured and simulated value of glass surface

In the actual production of fiberglass furnace, the most important technological parameter was temperature. The temperature of the crown and glass surface was measured by the furnace thermocouples embedded in its walls. The simulated results of the crown and glass surface in Figure 4 and Figure 6. The simulated temperatures and the measured temperatures of crown and glass surface were compared in Figure 5 and Figure 7.

Through the simulated and measured compared result, it can be found that although some assumptions were made in simulation process, but the tendency was consistent. Results indicated that the maximum relative error was less than 2%, which suggested that the mathematical model can objectively reflected the real distribution of the temperature field and velocity field of the fiberglass furnace.

3.2.2 The Velocity Field and Temperature Field in Combustion Space

Figure 8 showed the velocity distribution of x-z section distance 0.5m from glass surface, obviously, the flame from ports at the both sides. The fume as the reaction production removes from the flue gas. This was the fume mainstream direction in combustion space, and suitable to the actual situation. But because 3#, 4# port's gas flow rate was too big to cause the flowing in combustion space unstably, and impact against the crown.

The temperature contours of combustion gases at the middle planes of ports 1–6 were shown in Figure 9. The maximum temperature was 2730 K and related to the flames of ports 3, 4 and 5. These ports received the maximum gas flow rate to ensure that the batch blanket sinks until the middle of glass tank. But in analyzes of the temperature field distribution in 3#, 4# port planes indicated that the temperature was low where approach to the side wall, and the temperature was not full of the entire space, but only concentrated in the spray gun ambient temperature distribution.

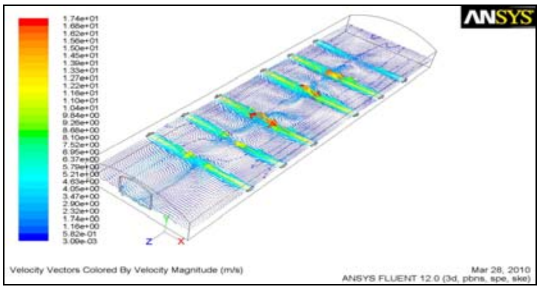

Figure.8 Velocity distribution of *xz* cross-section distance 0.5m from the glass surface

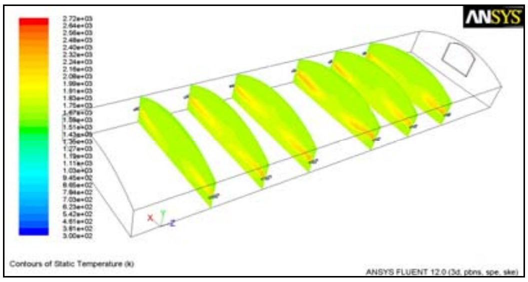

Figure.9 The temperature contours at the middle planes of ports1-6

In summary, existing combustion technology was unreasonable, and needed to modify the technology to achieve the energy the effective use.

3.2.3 Analysis of Modify Simulated Result

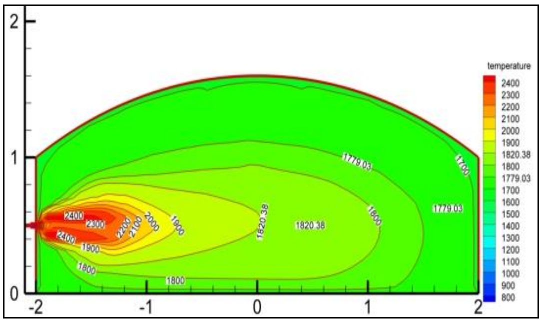

Figure.10 Improved temperature distribution in the center of the 3# port

978-1-935068-37-2 © 2010 SciRes.

Improvement of temperature distribution results were shown in Figure 10. Under the new combustion technology the entire temperature distribution was evener than before. The high temperature averaged distributes in the combustion space bottom and transmitted the thermal centralism to the glass surface. The temperature of the crown went too low, which was advantageous to reduce the crown heat loading.

3.2.4 Analysis of Simulated Result in Glass Tank

The method to couple the combustion space and the glass tank models was applied by one-way couple. In this method the heat flux of the glass tank and the batch blanket free surfaces was estimated by thermal boundary condition.

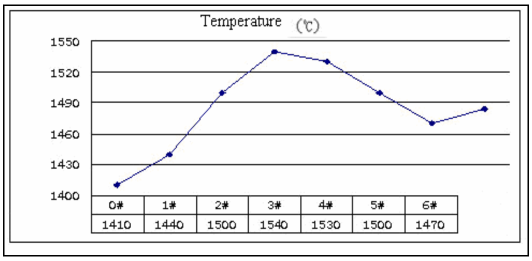

Figure.11 The average temperature distribution of glass surface

Applying this boundary condition, the combustion space was simulated and the average temperature over the free surface was calculated, which was shown in Figure 11. Input this temperature distribution to the glass tank by using UDF- temperature code. The temperature distribution was shown in Figure 12. Using this temperature distribution, the glass tank and the batch blanket were simulated and the heat flux of free surface was calculated. The calculated and the guessed heat flux were compared. If difference was significant, the above operation was repeated until convergence was achieved.

The temperature contours of the glass tank at its longitudinal middle plane were shown in Figure 13. The maximum temperature was 1800 K and was related to the free surface at the middle regions of the glass tank.

The streamlines of the glass tank at its longitudinal middle plane were shown in Figure 14.

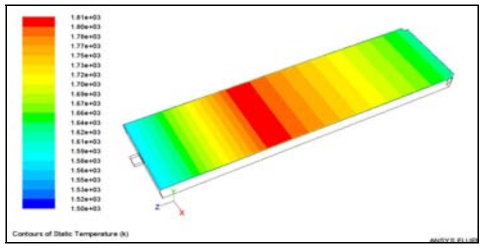

Figure.12 The temperature distribution of glass surface in glass tank

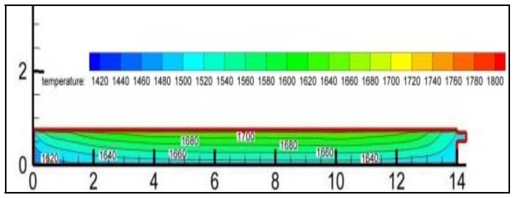

Figure.13 Temperature distribution of glass tank along it middle plane

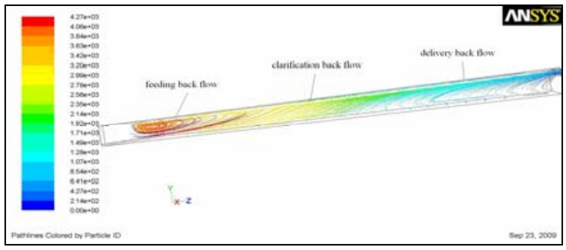

Figure.14 Streamlines distribution of glass tank along it middle plane

The molten flow was dictated by three free convection back flow. The delivery back was flow from the middle regions of the glass tank to the back wall. This vortex played an important role of heating the batch blanket lower boundary, although some of its heat was lost through the bottom and the side refractory of glass tank.

4 Conclusions

(1) Based on the goal date acquisition for the actual glass furnace, the work established hot boundary condition computation model based on the numerical simulation software. It can reduce to the numerical simulation boundary condition supposition, and the simulation result can more reflect the actual situation in the glass furnace than before.

(2) Comparing with measured data, the total relative error lower than 2% which explained the simulation re-

sult had the very high accuracy.

(3) On the basis of research results, by adjusting the layout of the port on the combustion space, it can obtain reasonable results to achieve the energy the effective use.

(4) This paper established a method to couple the combustion space and the glass flowing models by using UDF program.

References

[1] Omar S Asfour, Mohamed B Gadi. Using to investigate ventilation characteristics of vaults as wind-inducing devices in buildings [J]. Applied energy, 2008(25):120-134.

[2] S L Chang, C Q Zhou, B Golchert. Eulerian approach for multiphase flow simulation in a glass melter [J]. applied thermal engineering, 2005(25):3083-3103.

[3] Petr Schill, Josef Chmelar. Use of computer flow dynamics in glass technology [J]. Journal of NON-crystalline solids, 2004(345):771-776.

[4] M Falcitelli, S Pasini, L Tognotti. Modelling practical combustion systems and predicting NOx emissions with an integrated CFD based approach [J]. Computers&Chemical Engineering, 2002(26):1171-1183.

[5] Evatt R Hawkes, Jacqueline H Chen. Direct numerical simulation of hydrogen-enriched lean premixed methane-enriched[J]. Combustion and flame, 2004(138):242-258.

978-1-935068-37-2 © 2010 SciRes.