Abstract

The heat dissipated by the frit furnace walls is little used. Considering that the outside temperature of the walls of a frit furnace generally ranges from 250-300 °C, it is possible to take advantage of some of this energy. CFD simulations are done for an air flow near the outer surfaces of the furnace (floor and sidewalls), through its own channels. The result of this article shows that it is possible to heat air up to 80°C, with a flow of 200 m$^3$/h near the furnace floor. The extracted heat rate, in this case, reaches 5.0 kW. In the case of the sidewalls, the rate reaches 16.0 kW to heat the air 80°C. The rates of extracted heat and the air outlet temperatures vary according to flow used in the process.

Keywords: CFD, waste heat recovery, frit furnace, energy efficiency.
INTRODUCTION

The main advantages of waste heat recovery in engineering applications are two: the saving in consumption of non-renewable sources in order to the environment preservation with the reduction of pollutants emission, and the reductions of capital costs in new installations. In some cases, such as industrial furnaces, efficiency improvements resulting from waste heat recovery can improve energy efficiency by 10% to as much as 50% [1]. Considering that, glass manufacturing accounted for 1% of total industrial energy use in U.S., and the main fuel source is the natural gas with a participation of 73% of the total fuel sources [2], the waste heat recovery in glass furnaces has a promising future.

The most of the works about waste heat recovery only focus in the flue gases energy due to their high temperature. The objective of this work is the use of the heat lost in the walls of a frit furnace. The wall heat losses were measured by [3] and the results showed that almost of 30% of the energy input in a frit furnace is lost by the walls. The heat recovery from the walls in large size furnaces, like rotary kilns, can be considerably high, due to the large heat transfer area in the external walls e.g. in a rotary kiln used for calcination of dolomite, 26.35% of the input energy is lost in the walls, in contrast with 18.95% of the energy lost in the flue gases. [4]

This work shows an approach using the computational fluid dynamics (CFD) to quantify the heat rejected by the refractory bricks in a frit furnace. This energy is used to heat an air stream. The use of the heat air is, basically, the drying of the frit after process. In addition, the temperature of the air after heating is calculated. Other aspect considered in this work is the cooling of the refractory bricks to rise the durability and performance of them.

METHODOLOGY

The methodology used in this work is the computational simulation using the finite volumes software ANSYS to resolve the conservation equations (energy, momentum, etc.) for obtain the heat extracted and temperature profiles in the bricks and the air respectively. The simulation was divided in the following steps, as follows:

Physical domains involved in the analysis

The first step in any computer simulation is the meshing of the different domains presented in the problem, in this case, the domains correspond to the several layers presents in
the furnace wall, the heated air, and the structural shapes in the floor. Figure 1 shows the general scheme of the furnace with the main parts and Figure 2 presents the distribution of the brick layers in the floor of the furnace.

![Figure 1](image1.png)

**Figure 1** – General scheme of a frit furnace.

![Figure 2](image2.png)

**Figure 2** – Bricks distribution in the furnace floor.

The first layer is composed of an electro-melted brick layer. This layer has the special function of resist the wear caused by abrasion due to the melted glass flowing. The fabrication process of this kind of bricks make possible a minimization of the surface at the interface melted glass-brick [5]. The refractory brick layer has a structural function; it is cheaper than electro-melted brick and is thicker than the other brick layers installed. Finally, the insulating layer has the lower thermal conductivity and guarantee a good insulation of the furnace. For the case of the sidewalls, only refractory and insulating brick are installed in the furnace. Table 1 presents the thermal properties of the bricks.
Table 1 – Properties of refractory bricks. Adapted from [6].

<table>
<thead>
<tr>
<th>Brick</th>
<th>Al$_2$O$_3$ %</th>
<th>SiO$_2$ %</th>
<th>ZrO$_2$ %</th>
<th>Density (kg/m$^3$)</th>
<th>Specific Heat (kJ/kg K)</th>
<th>Thermal conductivity (W/m K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Refractory</td>
<td>63.26</td>
<td>32.01</td>
<td>-</td>
<td>2350</td>
<td>1.05</td>
<td>1.15</td>
</tr>
<tr>
<td>AZS 41</td>
<td>45</td>
<td>13</td>
<td>41</td>
<td>2850</td>
<td>0.84</td>
<td>8.84</td>
</tr>
<tr>
<td>Insulated</td>
<td>37</td>
<td>58</td>
<td>-</td>
<td>2000</td>
<td>1.11</td>
<td>0.58</td>
</tr>
</tbody>
</table>

*AZS 41 – Electric melted refractory.

The structural profiles help to support the weight of the whole furnace. The air flows between the two profiles. The meshing process was made in the software ICEM CFD. All the elements of the mesh in all domains are structured and the quality is between 90% and 100% and the aspect ratio of the elements is <100 [7].

**Solving the model**

The software ANSYS CFX-PRE was used in this work to set the boundary conditions in the model. The software employs the finite volumes methods to solve the conservation equations. Details of the method are found in [8]–[10]. The boundary conditions at the model are listed:

- The temperature at the top of the electro-melted layer is 1500°C, according to inside temperature measured. In the case of sidewalls, the inner temperature was 1500°C too.

- All external surfaces in the model are considered adiabatic.

- The velocity at the inlet of the air is varied to 2 m/s to 20 m/s.

When the boundary conditions are set, the next step in the simulation is the solution of the equations. The residual is the convergence criteria adopted [11] and has a value of 0.0001.

**RESULTS**

After several simulations, the temperature profiles for the air and the bricks were obtained for the case of the floor and the sidewalls.
Floor

The temperature profiles are obtained using the software ANSYS CFX POST. Figure 3 shows the temperature distribution along the air under the bricks layer in the case of a velocity of the air of 1 m/s. The air inlet is in the left of the figure.

![Temperature Profile](image)

**Figure 3** – Air flow on the floor of the furnace (air velocity 1 m/s, 75.6 m$^3$/h).

Figure 4 shows the same configuration but the velocity is 9 m/s. Comparing Figures 3 and 4, the increase of the velocity of the air reveals a lower temperature at the outlet.

![Temperature Profile](image)

**Figure 4** – Air flow on the floor of the furnace (air velocity 9 m/s, 680 m$^3$/h).

Figure 5 is a cross section of the floor showing the temperature distribution in the refractory bricks, considering the temperature at the top of 1500 °C.
Figure 5– Cross section of the floor and the temperature distribution along the bricks.

Sidewalls

In the same manner, an air stream flows from the left to the right. In Figure 6, the velocity corresponds to 1 m/s. The high temperature is shown in the middle of the heat transfer area due to the flow pattern described by the air generating a stagnant flow in the middle of the area. The dimensions of the cross section and the position of the inlet and outlet were randomly assigned. With the same configuration, Figure 7 presents the same flow pattern with velocity of 9 m/s. The temperature decreases, the “hot” area decreases too. To avoid hot areas in the middle and modify the flow pattern, Figure 8 shows a multiple inlet configuration that eliminates the middle stagnant flow and let to the air the uniform cooling.

Figure 6– Sidewalls temperature profile (air velocity = 1 m/s, 144 m³/h).
Figure 7— Sidewalls temperature profile (air velocity = 9 m/s, 1300 m$^3$/h).

Figure 8— Sidewall multiple inlet configuration (air velocity = 5 m/s, 720 m$^3$/h).

**Heat extracted**

Figure 9 illustrates the extracted heat from the furnace floor where two cases were analyzed: the first case corresponds to a length equal to the furnace width and the second case with a three times the width (5.4 m). The results show that the increment of the velocity in the air is proportional to the extracted heat, although for velocities higher than 5 m/s, the gain in extracted is minimum and the air pressure drop is increased too. When the channel length is increased (5.4 m), the extracted heat and the outlet air temperature grow. For the velocity of 5 m/s and a channel length of 5.4 m, the extracted heat is 5 kW. The whole floor has 10 channels of 5.4 m, so the total extracted heat of the floor is approximately 50 kW. The temperature of the air at the exit is increased with the augmentation of the length too. In the case of 5 m/s and a length of 5.4 m, the temperature is approximately 60 °C, while in the single channel is 30 °C. In the case of the sidewalls, for a velocity of 5 m/s, the extracted heat is approximately 16 kW by one wall, so the total heat extracted from the sidewalls is approximately 32 kW and the air is heating to 120 °C.
CONCLUSIONS  

The total heat extracted from the floor and the two sidewalls of the furnace is approximately 80 kW for a velocity of air of 5 m/s with a mean air temperature at the outlet of 100 °C. For this reason, the use of the heating air is limited to the drying process of the frit. Velocities higher than 5 m/s are not recommended due to the lower temperatures reached. An additional benefit is the cooling of the bricks for preserve the life of the refractories and reduce the discomfort produce in the factory due to the thermal radiation emitted by the wall in high temperatures.
ACKNOWLEDGMENTS

The authors thank to “Programa de Recursos Humanos da ANP para o Setor de Petróleo e Gás Natural – PRH09-ANP-MME-MCT” for the economic support of E. Morsch Fº, to SCGÁS for financing this project and special acknowledge to Smalticeram Unicer do Brasil Ltda. for the access to the frit furnace.

REFERENCES