CFD Simulation of Glass Melting Furnace

Prawin C. Patel[1] V.K.Matawala[2]
[2] Professor
[1,2] Shree S'ad Vidya Mandal Institute of Technology, Bharuch

Abstract--- Melting glass requires large amounts of energy, representing 15% of manufacturing cost for industrial producers. Melting glass furnaces' primary source of energy is natural gas. Inefficiencies in furnaces cause energy consumption per ton of glass to range from 3.8 to 20 mm Btu. The majority, 70%, of the energy used to produce glass is for smelting and refining of the glass. The high inefficiencies cause 60% of the energy consumed for smelting and refining to be lost to the atmosphere. This represents a large amount of waste, 6.3% of total manufacturing cost. The rising cost of natural gas is expected to increase the fuel cost proportion of total manufacturing cost and decrees the competitiveness of glass producers. Thus there is a need for minimizing this amount of energy that is wasted. The development and verification of an innovative modeling technique for industrial gas fired glass melting furnaces will be discussed. Rapid and accurate modeling of industrial furnaces is needed for the development of advanced furnace control systems. Although, computational fluid dynamics modeling of industrial furnaces has been shown to be accurate, they are computational expensive. The CFD simulation match with information obtained via CCD camera for calculation heat exchange within the furnace. The calculated heat flux to the glass using turbulence and species model is in close agreement with the CFD model. It is also observed wall heat loss during combustion.

I. INRODUCTION

The guidelines of the huge R&D efforts made around glass furnaces have changed at the 80-90 turn. First dedicated to optimization of the still very poor system efficiency (50%) and reducing of energy costs, priorities since one decade are more dictated by the expanding environmental regulations. With earth global warming in background, more and more constraints on gas and particulates emissions (NOx, SOx, CO2, and soot) are applied to industries. The key areas are therefore improvements in emissions control, recycling methods and solid waste management [1]. The additional costs to meet environmental compliance in form of gas treatment installations for example as the widely disparate regulations in the different regions of the world constitute a tremendous challenge for the glass industry in the next few decades.Glass makers have to step forward on a narrower and narrower way, as the productivity of furnaces can still be increased interestingly [2]. An important area of interest is the combustion chamber, with study of staged and oxy-fuel combustion, re-burning, low-NOx burners. The focus is also set on a better understanding of heat transfer mechanisms in the furnace, and on finding better ways to minimize losses. More than never, simulation remains the best way to investigate new designs and operating conditions. In the applications just mentioned, Computational Fluid Dynamic (CFD) simulation implemented on fine mesh grids allow to get very precise prediction at the cost of heavy computation load.

Motivated by the need of such model sand the good results of zone method, we investigate the validity of a first principles model based on a coarse spatial decomposition of combustion chamber, bath and walls. We present here first the glass furnace and the modeling approach we conclude with applications and perspectives.

II. THE NEED TO MODEL FURNACES

A commonly overlooked factor in energy efficiency is scheduling and loading of the furnace. "Loading" refers to the amount of material processed through the furnace in a given period of time. It can have a significant effect on the furnace's energy consumption when measured as energy used per unit of production (Btu/lb). Certain furnace losses (wall, storage, conveyor and radiation) are essentially constant regardless of production volume; therefore, at reduced throughputs, each unit of production has to carry a higher burden of these fixed losses. Flue gas losses, on the other hand, are variable and tend to increase gradually with production volume. If the furnace is pushed past its design rating, flue gas losses increase more rapidly, because the furnace must be operated at a higher temperature than normal to keep up with production. Total energy consumption per unit of production will follow the curve in Fig. 2.6, which shows the lowest at 100% of furnace capacity and progressively higher the farther throughputs deviate from 100%. Furnace efficiency varies inversely with the total energy consumption. The lesson here is that furnace operating schedules and load sizes should be selected to keep the furnace operating as near to 100% capacity as possible. Idle and partially loaded furnaces are less efficient In order to achieve maximum efficiency it is required to design the furnace part load to the maximum furnace capacity. A numerical tool with a furnace model helps to achieve this goal. Also another important quality metric in the heat treatment processes is soak time (amount of time a load stays at a given temperature) a furnace model with work piece thermal profile prediction can accurately predict the time required to reach the process temperature and the soak time. So the process can be accurately designed eliminating guesses and removing conservative recipes.

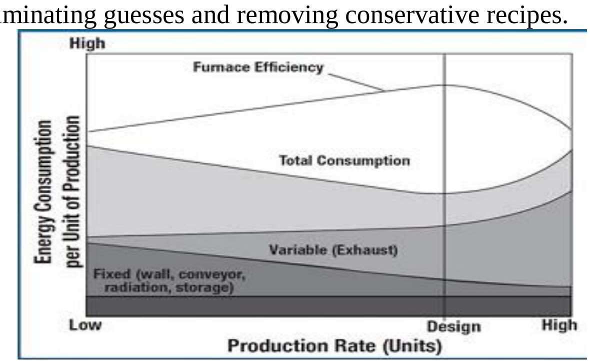

Fig. 1: Impact of production rate on energy

consumption per unit of production [3]

III. EXPERIMENTAL SET-UP DESIGN

The proposed method employs data from a CCD camera and utilizes established heat transfer calculation methods. The CCD camera provides information such as furnace wall temperatures, flame dimensions, and flame temperatures. This information is then used to calculate the radiative heat flux to the glass batch and melt. Convection with in most glass furnaces only accounts for a few percent of the total heat flux to the glass surface [4]. The convective heat transfer and its impact is not considered in this study. Hence, fluid flow equations do not need to be solved. In turn this decreases the computational expense needed for effective heat transfer calculations, because the fluid flow equations do not need to be solved. The data from the CCD camera is incorporated into the two rapid models for radiative heat transfer calculations.

IV. COMPUTATIONAL FLUID DYNAMICS METHODOLOGY

CFD – FLUENT is used for the modeling and simulation in this project. CFD – FLUENT is computer software that allows modeling and simulation of flow of fluid and heat and mass transfer in complex geometries. It is capable to complete meshing flexibility, solving flow problems with unstructured meshes that can be generated through the complex geometries. The program structure is shown in Figure 4.1.

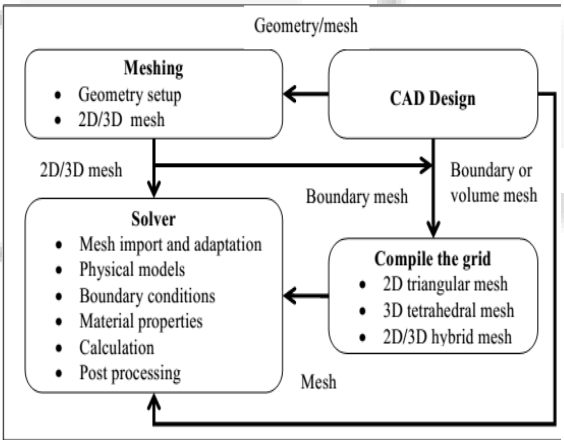

Fig. 2: .Program Structure.[5]

The computational domain is established in Fluent preprocessor. Dimensions from architectural drawings of the industrial glass furnace were used to draw the furnace within solid-works. The melter area of the furnace is the area of interest for modeling. The rest of the furnace including port necks, regenerators, and working area of the furnace will not affect the Rapid model. The exterior of the computational domain is shown in Figure 4.2

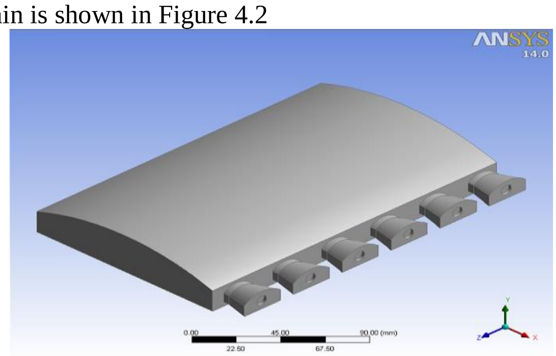

Fig. 3: Geometry Modeling

Meshing is a part of modeling. After complete geometry mesh will be applying on geometry. Meshing was required element size, element type and element connectivity in terms of skeewness which can be set in Ansys mesh modeler. Complete step of mesh generation not down element number and node number which can use for grid independency study. Mesh of above figure 4.2 is shown in figure number 4.3.

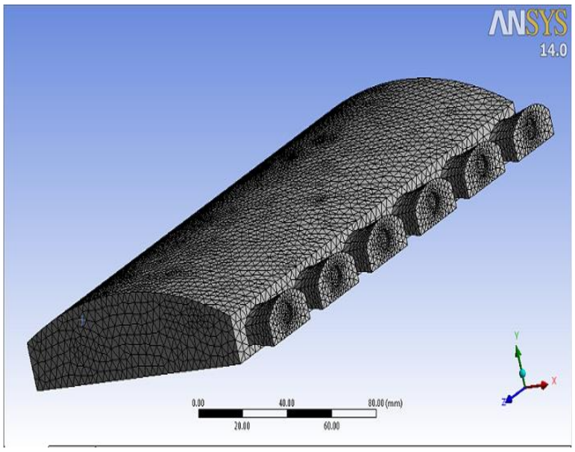

Fig. 4: Meshing of geometry

Boundary conditions specify the flow and thermal variables on the boundaries of your physical model. They are, therefore, a critical component of your **FLUENT** simulations and it is important that they are specified appropriately. As shown in figure 4.4.

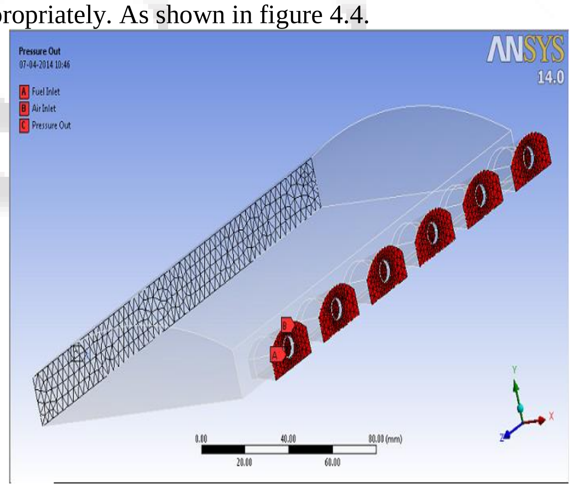

Fig. 5: Boundary define to mesh model

In this simulation model set as time steady and applying solver-pressure coupling method. Material- Liquid-Air at 600 K and Fuel inject at temperature 1900 K and Physical Model set k-epsilon turbulence with energy model along radiation with p1 model.

V. RESULT AND DISCUSSION

After finished simulation task in FLUENT. Its generating data this data use for further evaluation purpose and further improvement in results. From the contour of temperature was shown in figure 5.1 the temperature value at the symmetric plan. This contour generated from pressure 10000 Pascal and fuel temperature 1900K.

CFD simulations predict the minimum temperature 600 K and maximum temperature 1930 k after combustion. This is very close to practical temperature. The flame was propagated in linear direction.

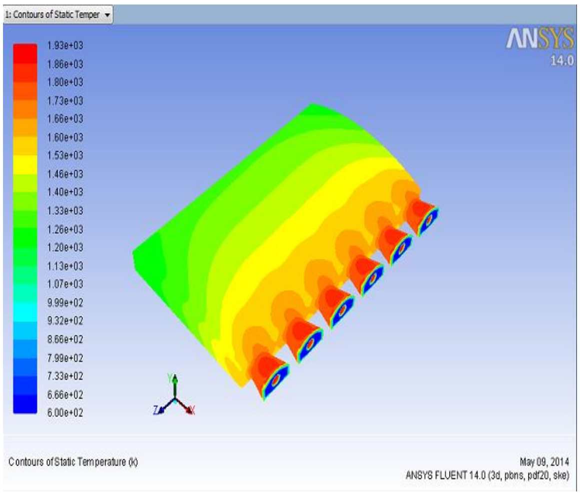

Fig. 6: Temperature Contour

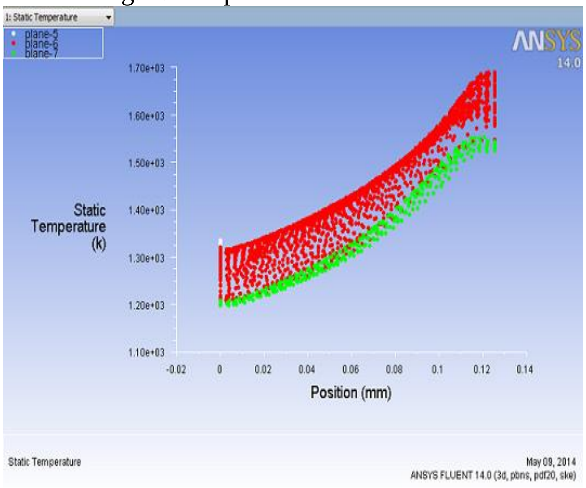

Fig. 7: Temperature Plot

Above plot was shown in figure 5.2 of three plane value of temperature it is observed that the value very close to furnace temperature value which is taken from the company. Furnace features and the glass surface are colored to indicate the total heat flux at the glass surface boundary was shown in figure 5.3. A negative value indicates that the glass surface is absorbing more heat than it is emitting.

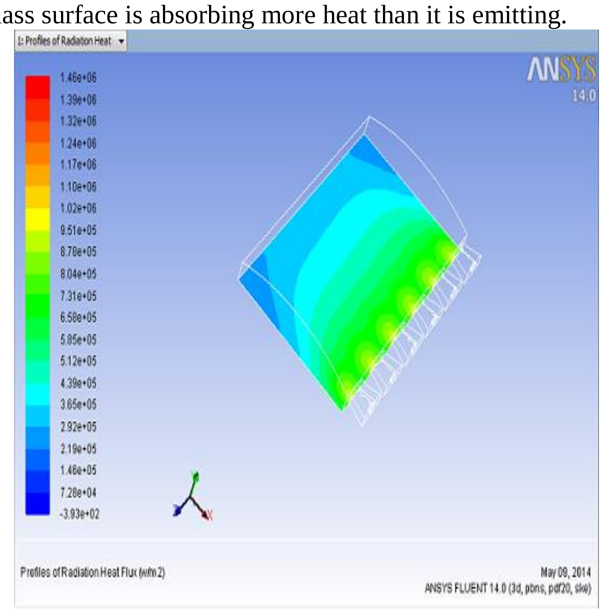

Fig. 8: Heat flux at glass surface

VI. CONCLUSION

The function of a glass furnace is to melt the raw materials and cutlet into molten glass, which is then shaped into different products. The efficiency of a furnace can be determined by the amount of energy required to produce a certain amount of glass. The more effective a furnace is at transferring the energy from the fuel source to the glass, the higher the furnaces efficiency. The effects of convective heat transfer in the combustion space of the CFD model also caused higher amounts of heat flux to the glass surface in the regions near the flames which is practically accepted.

REFERENCES

[1]. Glass: A clear vision for a bright future. Glass Manufacturing Industry Council .Expertise report of the Office of industrial technology (USA). 1996.
[2]. Industry Identified Combustion Research Needs for the Glass Industry. Expertise report of the Office of industrial technology (USA). 1997
[3]. Improving Process Heating System Performance: A Sourcebook for Industry is a development of the Best Practices initiative under the U.S. Department of Energy (DOE) Industrial Technologies Program (ITP) and the Industrial Heating Equipment.
[4]. Paarhuis, B.D., Lankhorst, A.M., Riepen, M., Velthuis, J.F.M., "A Rapid Simulation Model of The Combustion Space of Glass Melting Furances" TNO Institute of Applied Physics, Systems and Processes Divion Stieltjesweg 1, P.O. Box 155 2600 AD Delft, The Netherlands.
[5]. Fluent 6.2 user's guide, Ansys Corporation. 2002.