

2Dimensional CFD Simulation of Gas Fired Furnace during Heat Treatment Process

Vivekanand D. Shikalgar¹, Umesh C.Rajmane², Sanjay S. Kumbhar³, Rupesh R.bidkar⁴,
Sagar M. Pattar⁵, Ashwini P. Patil⁶.

Assistant Professor, Department of Mechanical Engineering, Sanjay Bhokare Group of Institute, Miraj-India.

ABSTRACT: Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. A computational fluid dynamics extensively used for modeling the gas flow in industrial furnaces. In this paper we are presenting a two dimensional computational fluid dynamics (CFD) simulation for gas fired furnace during heat treatment process. Steady state condition and different numerical models are used in this simulation. Temperature contours, velocity vectors plotted for detailing the simulation of gas fired furnace during heat treatment process. In CFD analysis computer aided design (CAD) software CREO is used for create geometry. Special meshing or grid generation software HYPERMESH is used. This stage is the key element in the CFD finite volume numerical simulation and it also contributes to the accuracy of the final results.

KEYWORDS: Gas fired Furnace, CFD, and Meshing.

I. INTRODUCTION

A) GENERAL

Heat treatment is process of heating and cooling the component to change mechanical properties of component like strength, hardness, ductility. Sometimes heat treatment processes also could be done to reduce the residual stresses in the component. These results of heat treatment processes directly related to wear resistance, heat resistance and life of component. Heat treatment processes requires the temperature control over the entire period of heating and cooling of component and soaking or keeping the component for some particular temperature to get uniform temperature though out entire across section of component. Furnace under this study is mainly used for heat treatment processes like Annealing, Normalizing, Hardening, Tempering, stress relieving. Furnace is gas fired furnace, fuel used in this furnace is mixture of air and natural gas.

B) PROBLEM STATEMENT

The heat treatment furnace is shown in figure.1. A small nozzle in the centre of burner introduces natural gas at 80mbar pressure and ambient air pressure is 600 mm of water column enters in to burner coaxially. The high speed natural gas jet mixes with air. The reaction of the mixture is given following $CH_4+2O_2=CO_2+2H_2O$, furnace is working at temperature of 1100°C. Furnace is not working satisfactorily because of following parameters are difficult to measure during the working of furnace:

Non uniformity in Temperature distribution inside the furnace.

C) AIM AND OBJECTIVES

If we can predict the above parameters, it is easy to adjust the input parameters like natural gas and air pressure to achieve the satisfactory working of furnace hence this can help to reduce the cost required in adjustment of input parameters unnecessarily. A Mathematical model of fluid flow in heat treatment furnace, commercial software CFD (Computational Fluid Dynamics) FLUENT is used. Turbulence in the combustion chamber is represented by standard k-ε model [1] [2] [6], species transport species model is considered for combustion. Radiation heat transfer is

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

represented by P1 model, P1 model is considered because it represents correct temperature during the combustion of natural gas [4].

II. COMPUTATIONAL MODEL AND GOVERNING EQUATIONS

The heat treatment furnace is showing with dimensions in following figure.1. Convection heat transfer is modeled using turbulence wall function in which near wall temperature profile follows universal profile through viscous sub layer and logarithmic region [5]. For radiation P1 model is considered because P1 model correctly represents the flame temperature during combustion of natural gas [4].

Standard k- ε turbulence model combined with standard wall function is considered for modeling gas flow in combustion chamber. Turbulence in combustion chamber is represented by standard k- ε turbulence model [1] [5]. k- ε turbulence model determine turbulence viscosity by calculating turbulence kinetic energy equation and its rate of dissipation equation. The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). In the derivation of the k-ε model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k-ε model is therefore valid only for fully turbulent flows. The turbulent viscosity computed by combining both k and ε as shown below [3].

$$\mu_t = \frac{C_\mu \rho k^2}{\epsilon} \tag{1}$$

The turbulence kinetic energy and its dissipation rate are obtained from the following

A) TRANSPORT EQUATION:

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon \tag{2}$$

$$\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C1_\epsilon \frac{\epsilon}{k} (G_k + C3_\epsilon G_b) - C2_\epsilon \rho \frac{\epsilon^2}{k} \tag{3}$$

$$G_k = \left[\mu_t \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial u_l}{\partial x_l} \right) \delta_{ij} \right] \frac{\partial u_j}{\partial x_i} \tag{4}$$

$$G_b = \beta g_i \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i} \tag{5}$$

C1 and C2 are constants σ_k and σ_ϵ are the turbulent prandtl number for k and ε. Default values of above model are shown in table. [1]

Table.II.I. Default values of standard k- ε turbulence model constants:

C_μ	C1	C2	σ_k	σ_ϵ
0.09	1.44	1.92	1	1.3

Table no.II.I

For combustion reaction is considered by using solver of mixing limited and reaction limited rates. For mixing limited rates Eddy Dissipation model is used with the rate given by.

$$R_{EDM} = A_{EDM} \left(\frac{\epsilon}{k} \right) \min \left([Fuel], \frac{[oxidant]}{C_{stoich}} \right) \tag{6}$$

Where,

$A_{EDM} = 4$ are coefficient, k and ε from the turbulence model. [Fuel] And [oxidant] are fuel and oxygen concentration respectively. C_{stoich} Is coefficient based on stoichiometric reaction.

For all flows, ANSYS FLUENT solves conservation equations for mass and momentum. For flows involving heat transfer or compressibility, an additional equation for energy Conservation is solved. Additional transport equations are also solved when the flow is turbulent.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

B) NAVIER- STOKES EQUATION:

The governing equation for steady state gas flow is the Navier Stokes equations, for compressible fluid and fluid continuity equation are follows:

$$\text{Continuity equation: } \frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (7)$$

$$\text{X-Momentum: } \frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial \rho}{\partial x} + \frac{1}{Re_r} \left[\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \right] \quad (8)$$

$$\text{Y-Momentum: } \frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial \rho}{\partial y} + \frac{1}{Re_r} \left[\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \right] \quad (9)$$

$$\text{Z-Momentum: } \frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uw)}{\partial x} + \frac{\partial(\rho vw)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial \rho}{\partial z} + \frac{1}{Re_r} \left[\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \right] \quad (10)$$

Energy:

$$\frac{\partial(E_r)}{\partial t} + \frac{\partial(uE_r)}{\partial x} + \frac{\partial(vE_r)}{\partial y} + \frac{\partial(wE_r)}{\partial z} = -\frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y} - \frac{\partial(wp)}{\partial z} - \frac{1}{Re_r Pr_r} \left[\frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} + \frac{\partial q_z}{\partial z} \right] + \frac{1}{Re_r} \left[\frac{\partial}{\partial x} (u\tau_{xx} + v\tau_{xy} + w\tau_{xz}) + \frac{\partial}{\partial y} (u\tau_{xy} + v\tau_{yy} + w\tau_{yz}) + \frac{\partial}{\partial z} (u\tau_{xz} + v\tau_{yz} + w\tau_{zz}) \right] \quad (11)$$

C) CFD SIMULATION

To solve mathematical model of gas flow and heat transfer in the heat treatment furnace, CFD software, FLUENT is used. Governing equations combined with initial and boundary conditions are discretized in implicit form using the control volume method and solved using SIMPLE based approach. The combustion chamber turbulence was represented by the k-ε model [1] [2] [6]. Radiation heat transfer is represented by P1 model, P1 model is considered because it represents correct temperature during the combustion of natural gas [4]. Pressure based solver is used for simulation, pressure-based approach was developed for low-speed incompressible flows, while the density-based approach was mainly used for high-speed compressible flows. In pressure based solver pressure-based approach, the pressure field is extracted by solving a pressure or pressure correction equation which is obtained by manipulating continuity and momentum equations and the velocity field is obtained from the momentum equations. Since the governing equations are nonlinear and coupled to one another, the solution process involves iterations wherein the entire set of governing equations is solved repeatedly until the solution converges. Steady state two dimensional region is considered for gas flow and heat transfer simulation. The discretization schemes were first order for all the equations, except for the pressure that used standard.

III.GEOMETRY

Geometry of heat treatment furnace is shown in figure, it having single burner configuration at left hand side wall of furnace as shown. The dimensions of geometry are given following. The three dimensional model and assembly of component is created by using the CREO modeling software.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

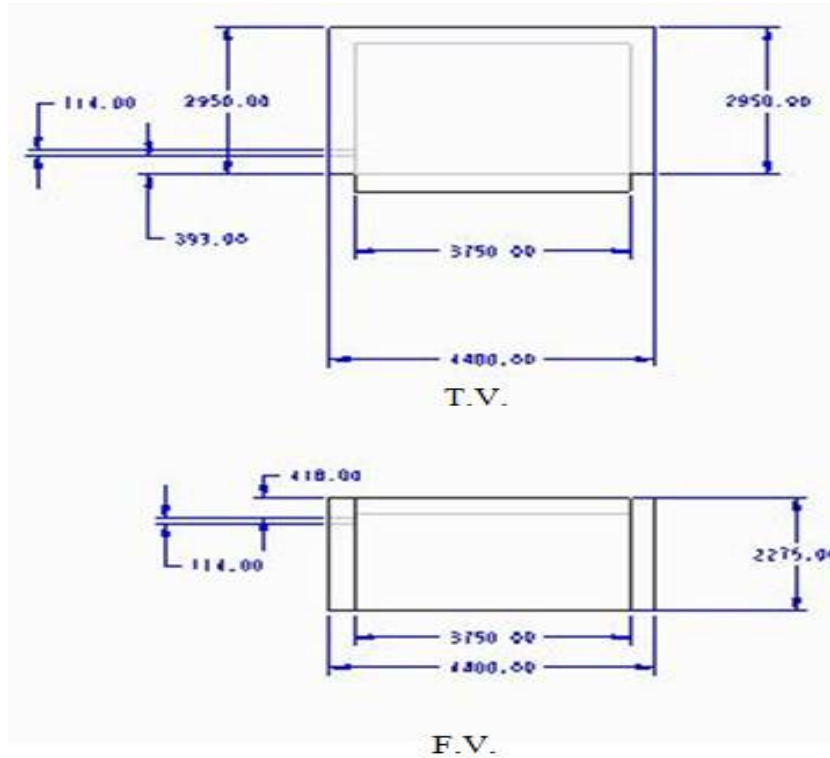


Figure. III.I. Two dimensional sketch of furnace

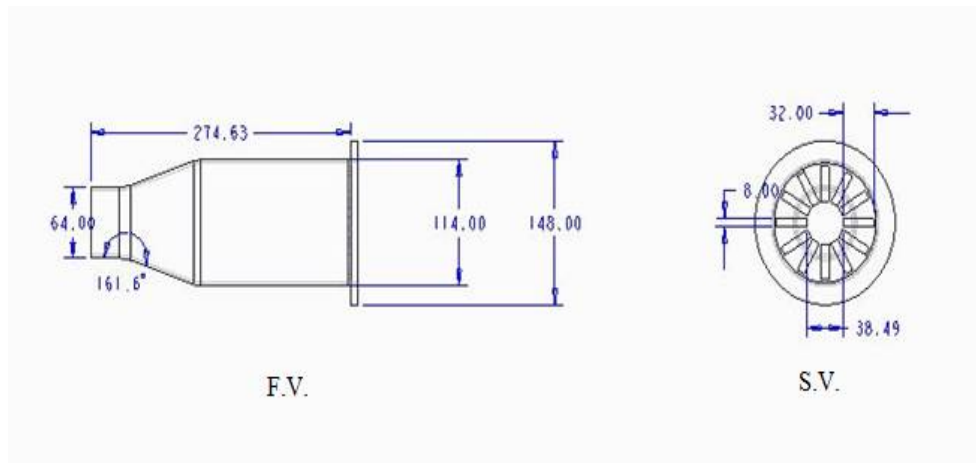


Figure.III.II. Two dimensional sketch of burner with diffuser ring

IV. MESHING

A Two dimensional geometry is created in Hypermesh software for meshing. Geometry is created by first crating node then lines are joined to those nodes. Surface is created by using surface command, once surface has been created it meshed by auto mesh command. Type of element taken is mixed it contain both the quadrilateral and triangles. Total number of element created is 18972 as shown below. After completion of meshing the element quality has been checked this shows element is created of good quality.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

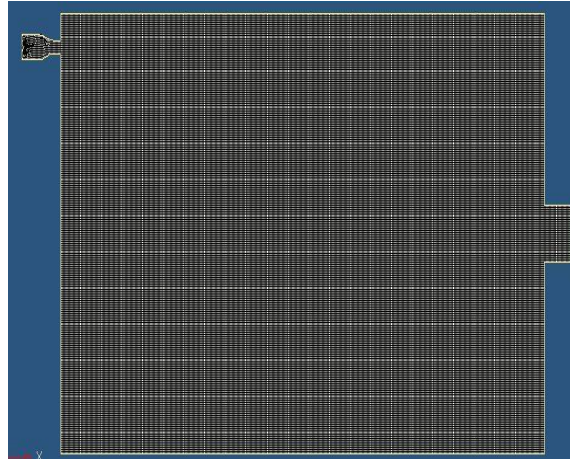


Figure.IV.II. Surface meshing

V. BOUNDARY CONDITIONS

Modeled equation	Momentum	Turbulence	Energy	Specie Mass fraction
Air inlet	P1 = 600 mm of water column	K= const ,ε= const	T inlet=323 k	O ₂ = 0.23
Fuel inlet	P2 = 80 mbar		T inlet=300 k	Ch4=1
Outlet	outflow	Extrapolate from the interior of domain	Extrapolate from the interior of domain	
Wall	No slip	Wall function	q = const	

Table.V.I. Initial boundary conditions are given in table below:

Wall is of insulating material made up of zirconia ceramic, it consist fibers of zirconia, ceramic, glass wool .zirconia cerawool material is created by FLUENT user defined function and thickness is 300mm given as shown. Properties of zirconia ceramic material are given in table.

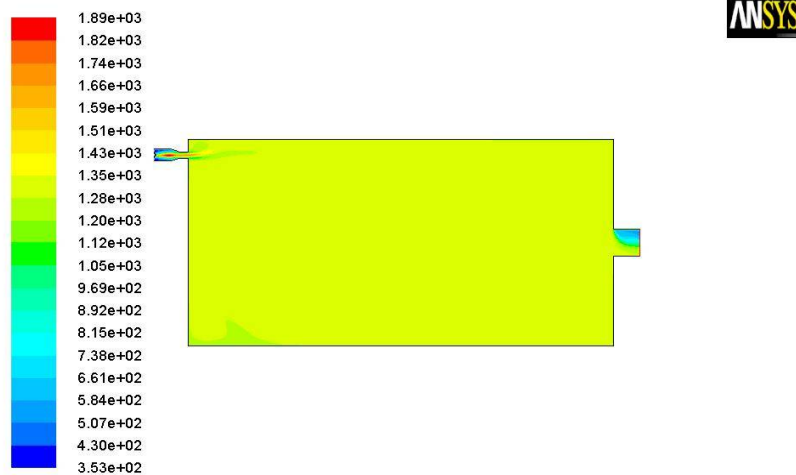
Thermal conductivity(k)	0.12 W/mk
Density(ρ)	164 kg/m^3
Specific heat (cp)	418.68 $J/kg k$ at room temperature
Electrical resistivity	71.04 Ωm
Refractive index	1

Table.V.II. Zirconia ceramic properties

VI. RESULTS

A). TEMPERATURE FIELD IN FURNACE:

Gas temperature distribution in 2 dimensional plane is shown following, the minimum temperature is 353 k and maximum temperature is 1890 k is shown at flame , slowly it decreases in x-direction. Temperature in middle portion shown by light yellow is of 1350 k. Temperature near the outlet is shown by blue color (661 k).

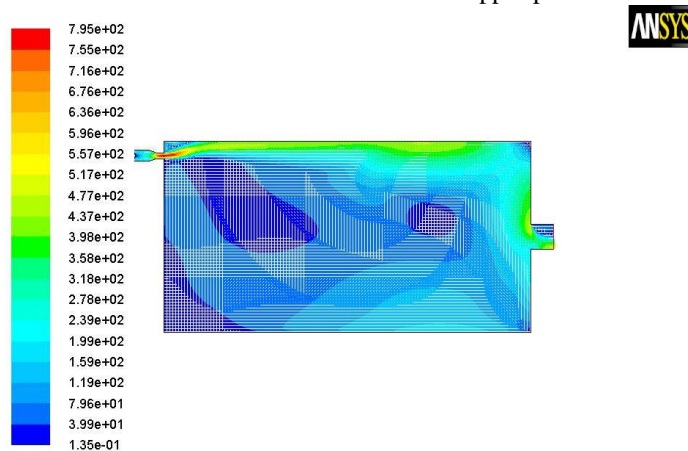


Contours of Static Temperature (k) ANSYS FLUENT 12.0 (2d, dp, pbns, spe, ske) Jan 22, 2015

Figure.VI.I. Temperature distribution in furnace

B). VELOCITY FIELD IN FURNACE:

Velocity vector in 2 dimensional planes is shown below, velocity is ranging from 0.135 m/s to 795 m/s . Maximum velocity is shown at flame combustion, after the combustion energy is released hence velocity is increases. Maximum velocity is at flame and it goes on decreasing towards outlet. At burner mixture velocity is max and when it enters in to chamber, turbulence is created at furnace corner as well as at upper portion.



Velocity Vectors Colored By Velocity Magnitude (m/s) ANSYS FLUENT 12.0 (2d, dp, pbns, spe, ske) Jan 22, 2015

Figure.VI.II. Velocity distribution in furnace

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 5, May 2016

VII. CONCLUSION

The CFD simulation of heat treatment furnace is successfully done by using ANSYS 12. In this two dimensional simulation species transport combustion model is considered with Natural gas as Fuel. The simulation gives the satisfactory values of temperature. Meshing is important factor in CFD which influences convergence or divergence of solution or error results. Mesh quality is checked with maximum skewness not to be exceeded more than 0.98. The solution method and solution monitors suitable for model to be solved. The residual monitor is very important to monitor iteration steps whether solution converge or diverge. The contour of temperature can be plot with ANSYS FLUENT.

REFERENCES

- [1] Ying Liu, Yiping Liu , Shuming Tao, Xunliang Liu, Zhi Wen, "Three-dimensional analysis of gas flow and heat transfer in a regenerator with alumina balls", Applied Thermal Engineering 69 (2014) 113-122.
- [2] Zhen Wang, You Fub, Ninghui Wang, Lin Feng, "3D numerical simulation of electrical arc furnaces for the MgO production", Journal of Materials Processing Technology 214 (2014) 2284-2291.
- [3] S.F. Zhang, L.Y. Wen, C.G. Bai, D.F. Chen, Z.J. Long, "Analyses on 3-D gas flow and heat transfer in ladle furnace lid", Applied Mathematical Modeling 33 (2009) 2646-2662.
- [4] Oscar Diaz-Ibarra, Pablo Abad, Alejandro Molina," Design of a day tank glass furnace using a transient model and steady-state computation fluid dynamics", Applied Thermal Engineering 52 (2013) 555-565.
- [5] L.C.B.S. Reis, J.A. Carvalho Jr, M.A.R. Nascimento, L.O. Rodrigues, F.L.G.Dias, P.M. Sobrinho, "Numerical modeling of flow through an industrial burner Orifice", Applied Thermal Engineering 67 (2014) 201-213.
- [6] Patrick G. Boisvert, Allan Runstedtler, "Fuel sparing: Control of industrial furnaces using process gas as supplemental fuel", Applied Thermal Engineering 65 (2014) 293-298