

## Numerical Analysis of Combustion Furnace Performance by CFD

V. N. Chougule, O. F. Daud, G. R. Gupta, V. H. Shelke, R. V. Ugale  
 (Department of Mechanical Engineering, M. E. S. College of Engineering, India)

**ABSTRACT:** High temperature processing of raw materials often involves complex multi-phase fluid flow and heterogeneous chemical reactions at various scales. Due to the extreme temperature conditions existing within and around an industrial furnace, it is extremely difficult to measure the temperatures within a furnace at regular intervals during one operation cycle. Hence the industry and market need an accurate method to estimate the flow characteristics of a furnace operation. Computational fluid dynamics (CFD) has become a very useful simulation tool to improve process understanding and development. The paper discusses the benchmarking of the CFD analysis by performing the CFD analysis on a mobile heat treatment furnace.

**Keywords** - Benchmarking of CFD, furnace, heat transfer, time steps

### I. INTRODUCTION

A furnace is an equipment to melt metals for casting or heat materials for change of shape (rolling, forging etc.) or change of properties (heat treatment). Based on the method of generating heat, furnaces are broadly classified into two types namely *combustion type (using fuels)* and *electric type*. Combustion furnaces based on the kind of combustion, it can be broadly classified as *oil fired, coal fired or gas fired*.

In Indian Industries, furnaces are operating below efficiency limits, causing large expenses. Industries like Bharat Forge, ISPAT Group, Indian Seamless Metals Ltd., Chakan Forge, etc. in and around Pune are using furnaces for various operations such as Heat Treatment, Melting, etc., which is common process for Automobile Industries, Process Industries, etc. Performance Evaluation of a typical Furnace is measured by thermal efficiency of process heating equipment, such as furnaces, ovens, heaters, and kilns is the ratio of heat delivered to a material and heat supplied to the heating equipment. The purpose of a heating process is to introduce a certain amount of thermal energy into a product, raising it to a certain temperature to prepare it for additional processing or change its properties. To carry this out, the product is heated in a furnace.

As per Bureau of Energy Efficiency, India studies typical thermal efficiencies as per following table.

Table 1 - Thermal Efficiencies for Common Industrial Furnaces

Furnace Type	Typical Thermal Efficiency (%)
Low Temperature furnaces	
a. 540-980 °C (Batch Type)	20-30
b. 540-980 °C (Continuous Type)	15-25
c. Coil Anneal (Bell) radiant type	5-7
d. Strip Anneal Muffle	7-12
High Temperature furnaces	
a. Pusher, Rotary	7-15
b. Batch forge	5-10
Continuous Kiln	
a. Hoffman	25-90
b. Tunnel	20-80
Ovens	
a. Indirect fired (20-370 °C)	35-40
b. Direct fired (20-370 °C)	35-40

\*Ref.: Bureau of Energy Efficiency, India

This is due to energy losses in different areas and forms. For most heating equipment, a large amount of the heat supplied is wasted in the form of exhaust gases.

These furnace losses include:

\*\*\* This work is supported by BCUD, University of Pune

Second National Conference on Recent Developments in Mechanical Engineering  
 M.E. Society's College of Engineering, Pune, India

- Heat storage in the furnace structure
- Losses from the furnace outside walls or structure
- Heat transported out of the furnace by the load conveyors, fixtures, trays, etc.
- Radiation losses from openings, hot exposed parts, etc.
- Heat carried by the cold air infiltration into the furnace
- Heat carried by the excess air used in the burners.

To date, the forging industry lacks a full-proof method of completely estimating the flow within a furnace. Due to the extreme temperature conditions existing within and around an industrial furnace, it is extremely difficult to measure the temperatures within a furnace at regular intervals during one operation cycle. Hence the industry and market need an accurate method to estimate the flow characteristics of a furnace operation. Hence this research work aimed at using CFD as an accurate and reliable tool to analyze furnace performance. This paper shows the use of CFD to accurately determine furnace operation parameters.

## II. LITERATURE SURVEY

### 1. CFD Modeling Fundamentals

ANSYS ICEM CFD provides advanced geometry acquisition, mesh generation, mesh optimization, and post-processing tools to meet the requirement for integrated mesh generation and post processing tools for today's sophisticated analyses. Maintaining a close relationship with the geometry during mesh generation and post-processing, ANSYS ICEM CFD is used especially in engineering applications such as computational fluid dynamics and structural analysis. ANSYS ICEM CFD provides a direct link between geometry and analysis.

In ANSYS ICEM CFD, geometry can be input from just about any format, whether from a commercial CAD design package, third party universal database, scan data or point data. Beginning with a robust geometry module which supports the creation and modification of surfaces, curves and points, ANSYS ICEM CFD's open geometry database offers the flexibility to combine geometric information in various formats for mesh generation. The resulting structured or unstructured meshes, topology, inter-domain connectivity and boundary conditions are then stored in a database where they can easily be translated to input files formatted for a particular solver. The Set of equations that are solves by Ansys CFX are unsteady Navier Stokes equations in their conservative form. These equations describe the processes of momentum, heat and mass transfer.[2] These partial differential equations were derived in the early nineteenth century and have no known general analytical solution but can be discretized and solved numerically. These equations can be given as:

$$\rho X - \frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} + \frac{\partial^2 v_x}{\partial z^2} \right) = \rho \frac{Dv_x}{Dt} \quad (1)$$

$$\rho Y - \frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} + \frac{\partial^2 v_y}{\partial z^2} \right) = \rho \frac{Dv_y}{Dt} \quad (2)$$

$$\rho Z - \frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2} \right) = \rho \frac{Dv_z}{Dt} \quad (3)$$

where, X, Y and Z are the body forces in the x, y and z direction respectively

$\rho$  is the density of fluid flowing

$\mu$  is the co-efficient of dynamic viscosity

$v_x$ ,  $v_y$  and  $v_z$  are velocities of flow in x, y and z directions respectively.[6]

#### 1.1 Heat Transfer

The temperature change in a body is governed by three physical processes: Convection Heat transfer, Latent heat transfer associated with mass transfer and radiation heat transfer. The Convective heat transfer  $Q_c$  is given in terms of Thermal Conductivity of fluid ( $\lambda$ ), Temperatures of fluid and particle ( $T_G$  and  $T_P$ ) and Nusselt number( $N_u$ ) by equation-

$$Q_c = \pi d \lambda N_u (T_G - T_P) \quad (4)$$

Nusselt number given by equation:

$$Nu = 2 + 0.6 R_g^{0.5} \left( \frac{\mu C_P}{\lambda} \right)^{1/3} \quad (5)$$

where,  $C_p$ = Specific heat of fluid and  $\mu$ = Fluid dynamic Viscosity.

For heat transfer associated with mass transfer the CFX uses following equation:

$$Q_M = \Sigma \frac{dm_c}{dt} V \quad (6)$$

Where sum is taken over all components of the particle for which the mass transfer is taking place. The latent heat of evaporation  $V$  is temperature dependent and is obtained from MATERIALS section in CFX for the liquid in the particle and its vapor. For heat transfer by radiation for particle with diameter  $d_p$ , uniform temperature  $T_p$  and emissivity  $\epsilon_p$  the CFX uses equation:

$$Q_R = \epsilon_p \pi d_p^2 (I - \sigma n^2 T_p^2) \quad (7)$$

Where,  $I$  is the radiation intensity on the particle surface at the location of the particle,  $n$  is the refractive index of the fluid, and  $\sigma$  is the Stefan Boltzmann Constant.

The Rate of change of temperature is then obtained by the equation (8) as:

$$\Sigma (m_c C_p) \frac{dT}{dt} = Q_C + Q_M + Q_R \quad (8)$$

Where, the sum in the equation (8) is taken over all the components of the particle including those not affected by mass transfer. [2]

## 1.2 Momentum Transfer

Consider a discrete particle travelling in a continuous fluid medium. The forces acting on the particle that affect the particle are due to the difference in velocity between the particle and the fluid, as well as to the displacement of the fluid by the particle. The equation for it was derived by Basset, Boussinesq and Oseen for rotating frame. The Equation is:

$$m_p \frac{dU_p}{dt} = F_D + F_B + F_{VM} + F_P + F_{BA} \quad (9)$$

$m_p$  is mass of the particle

$\frac{dU_p}{dt}$  is rate of change of velocity of the particle

The equation (9) has following forces on the right side:

$F_D$ = Drag force acting on the particle

$F_B$ = Buoyancy force due to gravity

$F_{VM}$ = Virtual Mass force - This is the force to accelerate the virtual mass of fluid in the volume occupied by the particle. This term is important when the displaced fluid mass exceeds the particle mass, such as in bubbles.

$F_P$ = Pressure gradient force - This is the force applied on the particle due to the pressure gradient in the fluid surrounding the particle caused by fluid acceleration. It is only significant when the fluid density is comparable to or greater than the particle density.

$F_{BA}$  = Basset force or history term which accounts for the deviation in flow pattern from the steady state.

The left hand side of the equation is modified due to special form of mass term called virtual mass term (This force is caused by the fact that the particle has to accelerate some of the surrounding fluid, leading to an additional drag.) The modified equation is given by eqn. as:

$$\frac{dU_p}{dt} = \frac{1}{m_p + \frac{C_{VM}}{2} m_F} (F_D + F_B + F'_{VM} + F_P) + \frac{1}{m_p} F_R \quad (10)$$

Only a part of the virtual mass term,  $F'_{VM}$ , remains on the right hand side. The particle and fluid mass values are given by:

$$m_p = \frac{\pi}{6} d_p^3 \rho_p \quad \text{and} \quad m_F = \frac{\pi}{6} d_p^3 \rho_F \quad (11)$$

With the particle diameter  $d_p$  as well as the fluid and particle densities  $\rho_F$  and  $\rho_p$ . The ratio of the original particle mass and the effective particle mass (due to the virtual mass term correction) is stored in equation given as:

$$R_{VM} = \frac{m_p}{m_p + \frac{C_{VM}}{2} m_F} = \frac{\rho_p}{\rho_p + \frac{C_{VM}}{2} \rho_F} \quad (12)$$

$$1 - R_{VM} = \frac{\frac{C_{VM}}{2} \rho_F}{\rho_p + \frac{C_{VM}}{2} \rho_F} \quad (13)$$

Using  $R_{VM}$  the equation can be written as:

$$\frac{dU_p}{dt} = \frac{R_{VM}}{m_p} (F_D + F_B + F'_{VM} + F_P) + \frac{1}{m_p} F_R \quad (14)$$

### Numerical Analysis of Combustion Furnace Performance by CFD

Each term on the right hand side of equation (14) can potentially be linearized with respect to the particle velocity variable  $U_p$ , leading to the following equation for each term:

$$T = R + C_{lin}U_p \quad (15)$$

$C_{lin}$  is the linearization co-efficient

There are a number of different solution methods that are used in CFD codes. The most common and the one on which CFX is based, is known as the finite volume technique. In this technique, the region of interest is divided into small sub-regions, called control volumes. The equations are discretized and solved iteratively for each control volume. As a result, an approximation of the value of each variable at specific points throughout the domain can be obtained. In this way, one derives a full picture of the behavior of the flow. [2]

### III. BENCHMARKING

CFD is a process using a large number of complex equations. To solve these equations individually by hand is practically impossible for a large system like a furnace. Hence to simplify solving, a solver software is used which uses computing power to employ CFD. Nowadays, a large number of solvers are available commercially. The use of any particular solver does not necessarily mean a difference in results. But, with each solver comes a different and unique way of implementing the solution. Each solver in turn requires different variables to be input by the user. Furthermore, any CFD solver does not have the technique to ascertain whether the obtained solution is right or wrong. It will simply display the result based on the input that the user has given. Hence a solver follows the principle, "Garbage in  $\rightarrow$  Garbage out". So to make sure that our CFD analysis is accurate and that the research is following the right norms in accordance with generally accepted principles, a run was conducted on a pre-analyzed furnace. This furnace has been analyzed using known and tested CFD techniques in the past and is referred in a published and documented paper. This step is essential as it allows performing a run on our industrial with the confidence that the steps followed are accurate and will yield good results. This step is known as "Benchmarking".

Benchmarking involves the following vital steps:

- Selection of a tested furnace from a published paper and study it.
- Start CFD runs on the selected furnace in software package and using a selected method.
- Compare results obtained with the results published in the paper. If found matching, the selected method is acceptable.

CFD analysis for this paper has been done using ANSYS-CFX solver and ICEM-CFD as a meshing tool.

### IV. EXPERIMENTATION

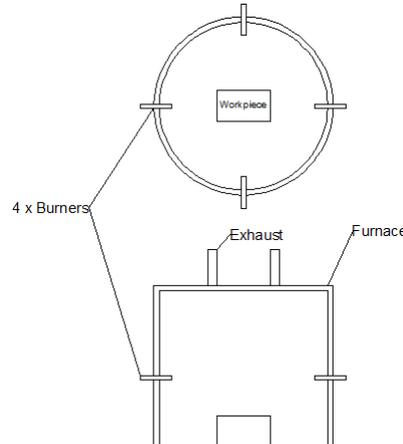


Fig 1: Furnace Design and Geometry

Fig. 1 shows a heat treatment furnace was simulated by using CFD to investigate thermal performance of the furnace and the heating process of the metal pieces. The furnace is used to heat treat dredging pumps and impellers to obtain the required microstructure and mechanical properties through stress relief, annealing, hardening and tempering. Since the temperature evolution inside the metal piece cannot be tracked in practice, CFD simulation provides a useful tool to predict the temperature evolution within the metal pieces during the heat treatment. The current CFD model consists of turbulent combustion, thermal radiation, and conjugate heat transfer. Heated air at 668°C is passed through burners over the impeller. The total time for simulation carried out is 5 hour in time steps of 20 minutes. The main purpose of this paper is to predict the temperature evolution within the metal pieces during the heat treatment and validate the results obtained with values from the paper.

**1. Furnace Design and Geometry**

- Furnace dimension: Diameter x Height - 3.76 m x 3.5 m (Volume = 156 m<sup>3</sup>)
- Burner diameter and location: 0.1 m in diameter, 1.5m from furnace bottom.
- Impeller dimensions: Outer Diameter x Inner Diameter x Height - 1.5 x 0.5 x 0.6

**2. Model Geometry**

The geometry created in ICEM is shown in the form of a wireframe model. The impeller is kept inside the furnace but is kept as a separate solid domain to enable the solver to interpret it as a separate entity. A simplified cad model of the impeller is used and the impeller is kept slightly above ground level in the furnace to facilitate use of supports in application.

**3. Mesh Parameters**

Meshing was done in ICEM-CFD. The fluid and solid domain were meshed separately. This was done to ensure that while applying boundary conditions, the solver can differentiate fluid and solid domains. Different mesh sizes were given to different parts based on their size. The mesh parameters are:

- Tetrahedral mesh is used.
- Mesh size is 0.1 for the furnace volume, 0.01 for the inlet and outlet pipes, and 0.05 for the impeller.
- A prism layer is kept around the impeller towards the fluid boundary. Prisms are of height 0.001 with a height ratio of 1.1. Three such layers are kept.

Prism layer helps in getting accurate reading around the impeller. Also keeping a good prism layer helps the solver to achieve a good y-plus value.

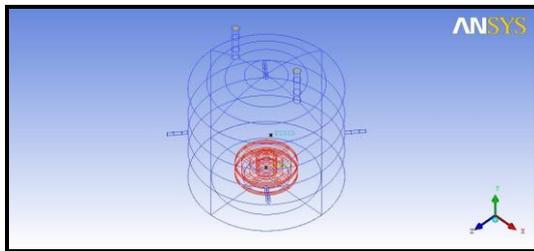


Fig 2: Model Geometry

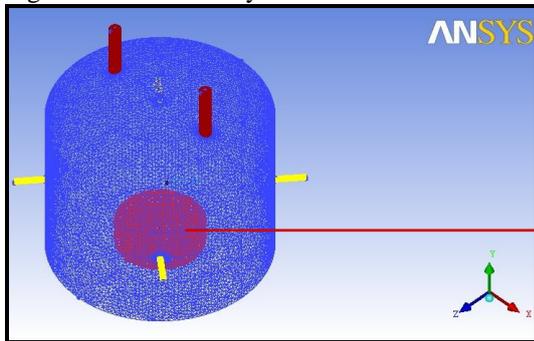
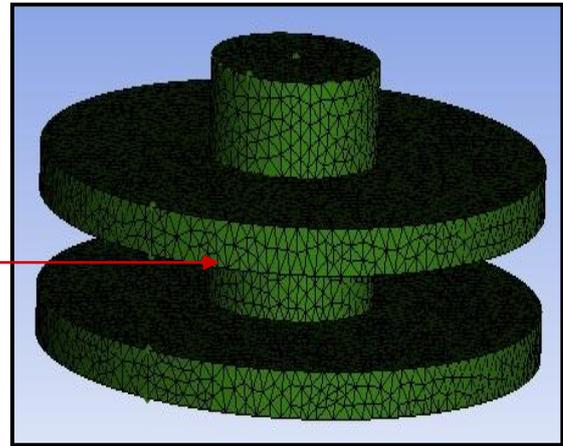


Fig 3: Mesh Parameters



**4. Assumptions:** The following assumptions have been taken while performing a solver run.

Table 2: Assumptions

No combustion	
Analysis type	: Transient
Working fluid	: Air at 668°C
Heat Transfer Model	: Thermal Energy
Turbulence Model	: K-Epsilon
Impeller is assumed to be continuous solid	
Domain assumed is stationary	

**4.1 Fluid Properties:**

Viscosity	1.09e-5 Ns/m <sup>2</sup>
Specific Heat	1004.4 J/kg-K
Conductivity	0.032411 W/mK
Thermal Expansivity	0.003356/K
Density	1.2 Kg/m <sup>3</sup>

**5. Boundary Conditions**

The boundary conditions following were applied as given in the paper. The above mentioned assumptions were applied to the boundary conditions and then given input to the solver. Mass flow rate is given as combined and the solver divides it among the available inlet pipes.

Table 3 Boundary Conditions

<b>INLET</b>	Mass Flow Rate – 0.37 kg/s
	Boundary Type – Inlet
	Flow Regime – Subsonic
	Turbulence - 5% Medium Intensity
<b>WALL</b>	No-Slip type
	Adiabatic
<b>OUTLET</b>	Boundary Type- Outlet
	Turbulence- Medium Intensity

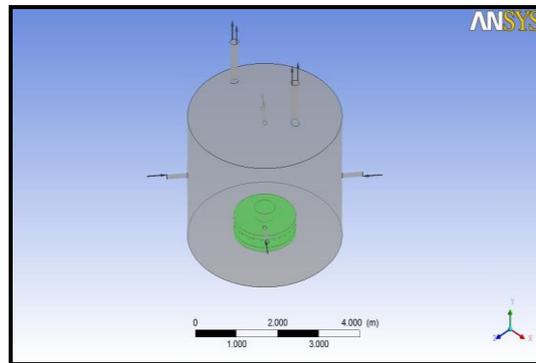


Fig 4: Boundary Conditions

**V. RESULTS AND DISCUSSION**

**5.1 General Gas Flow Pattern**

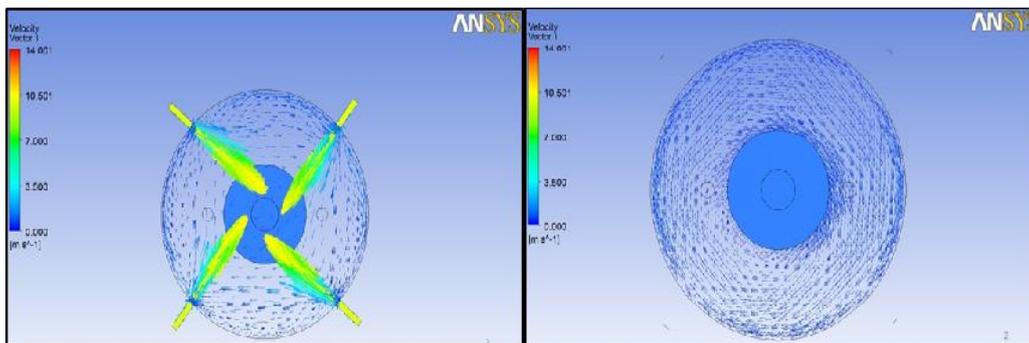


Fig 5: Gas flow pattern

As is described earlier, there are four oil burners installed slightly above the top surface of the impeller. The two opposing burners have a small angle of 6.5° so that the flame jets do not meet in the center of the furnace, and more stable and slightly rotating flames are formed. Figure 5 shows the general gas flow pattern obtained from the final model at three different cross sections. From Figure 5(a), 5(b) it can be seen that the jets from the four burners form a counter clockwise rotating flow. This rotating flow extends from the burner level to the top of the furnace upward and to the bottom of the furnace downwards. From Figure 5(c) it can be seen that near the center of the furnace between the two pair of burners, a downward gas flow is formed, which is ended near the impeller top-surface. Below the impeller the velocity is much lower, which is not efficient for the heating process of the lower part of the impeller.

**5.2 Temperature Evolution Results of the job**

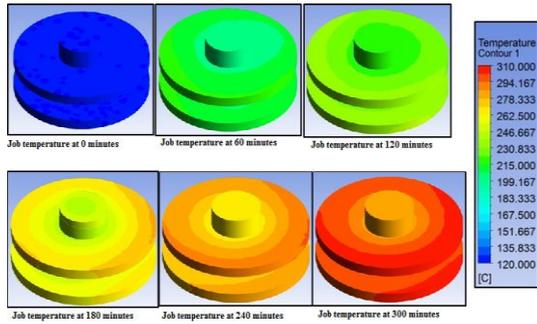


Fig 6: Temperature Evolution results of job

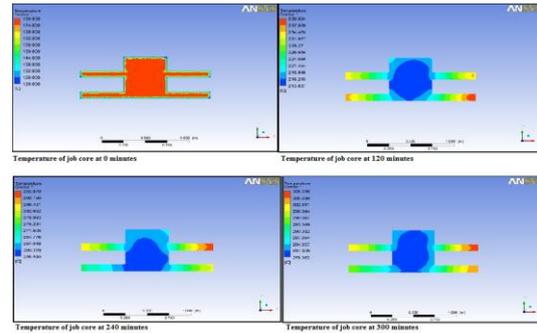


Fig 7: Temperature evolution at job core

**5.2 Temperature Evolution Results at the Job Cross Section:**

Fig.7 illustrates a temperature contours from the final transient simulation model. Due to the limited space in the paper, only a few time steps were chosen from the 15 time steps (time 0 – 5 hours). From these snap-shots, the heating processes of the furnace space and the impeller can be clearly seen along the heating process. The temperature of the impeller is higher on the top. The lower parts of the impeller have some delay in temperature increase. As was mentioned before the heating of lower parts is slow due to slower flow at the lower parts of the furnace. Hence the flow patterns support the temperature results.

**5.3 Thermocouples in Furnace Space:**

As per SAE standard AMS 2750D, 9 thermocouples are used for temperature measurement. The variation of temperature at the individual thermocouples is shown with the progress in time. The points are monitored continuously during the progress of solution.

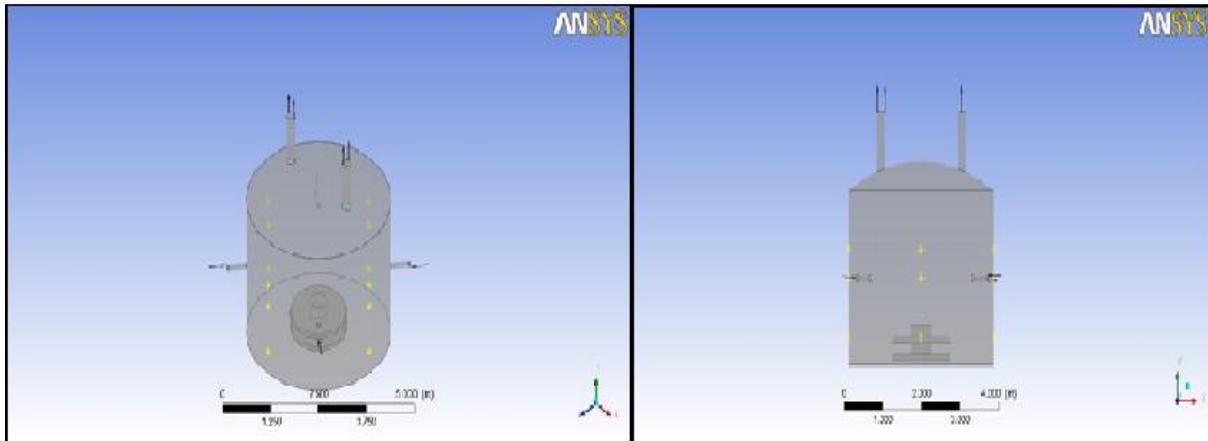


Fig 8: Placement of points in space

The placement of the points in two views is shown in fig. 8. The points are placed at 0.5, 1.5, and 2 meters from the furnace level. Hence they lie at job and burner level and space. Four such points are placed at 90 degrees to each other.

**VI. CONCLUSION**

The results obtained by simulation are documented as above. The heating profile was reasonably predicted by the simulation. Through the CFD simulation, not only the surface temperature could be properly predicted, but also the temperature within the metal components could be simulated. The latter is crucial for the heat treatment process, but is not possible to measure directly in practice. The results obtained by us are found to be in acceptable limit ( $\pm 5\%$ ). This indicates an extremely accurate range. Hence it is proved that the method of analysis used for the current paper is accurate and can now be applied to an industrial furnace.

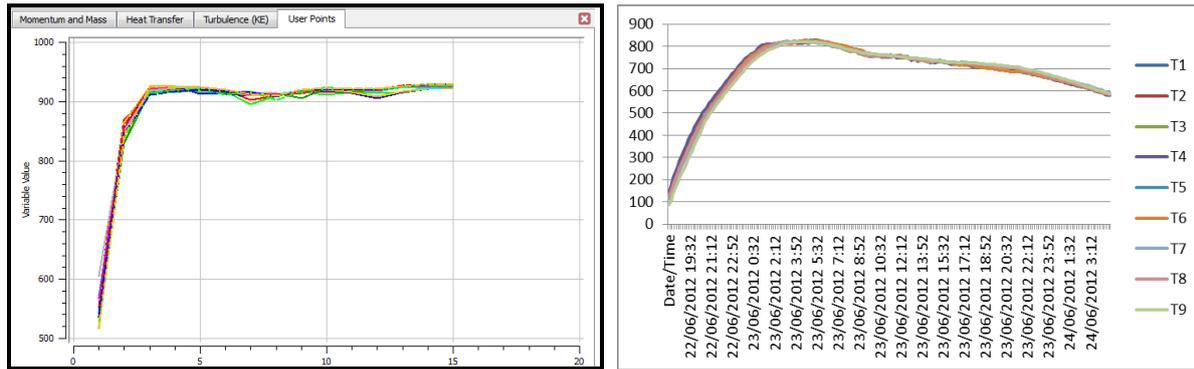


Fig 9: Variation of temperatures (a) Simulation results, (b) Actual results

## VII. ACKNOWLEDGEMENT

The authors would like to thank BCUD University of Pune for sponsoring this project vide BCUD/OSD/522 dated. 26/02/2011.

## REFERENCES

- [1] J. Blazek , Computational fluid dynamics: principles and applications (Amsterdam, Elsevier Publications, 2001)
- [2] ANSYS Inc., Ansys CFX.13 Theory Guide, 2010
- [3] R. W. Lewis, P. Nithiarasu, K. N. Seetharamu, Fundamentals of the finite element method for heat and fluid flow (New Jersey, John Wiley & Sons, Ltd,2008)
- [4] M. M. Rathore , Engineering heat and mass transfer (New Delhi, Laxmi Prakashan, 2006)
- [5] W. Trinks, M. H. Mawhinney, R. A. Shannon, R. J. Reed and J. R. Garvey, Industrial furnaces (New Jersey, John Wiley & Sons, Inc., 2004)
- [6] T. J. Chung, Computational Fluid Dynamics (Cambridge, Cambridge University Press, 2003 )
- [7] J. Kang , R. Purushothaman, Y. Rong A. K. Singh and L. Zhang , Computer Aided Heat Treatment Planning System for Quenching and Tempering , Integrated Computational Materials Engineering: Lessons from Many Fields, TMS (The Minerals, Metals and Materials Society), 2007
- [8] C. A. García, J. C. Moreno-Piraján and F. Sánchez, Simulation and Flow Analysis for a Brick Furnace, Electronic Journal Of Environmental, Agriculture and Food Chemistry,5, 2006
- [9] Y. J. Jang and S. W. Kim , An Estimation of a Billet Temperature during Reheating Furnace Operation, International Journal of Control, Automation, and Systems, 1(5), 2007, 43-50
- [10] R. A. de Jong , M. A. Reuter and Y. Yang, Use of CFD to Predict the Performance of a Heat Treatment Furnace, Fourth International Conference on CFD in the Oil and Gas, Metallurgical & Process Industries SINTEF / NTNU Trondheim, Norway, 2005.
- [11] R. Boom, J. R. Post, M. A. Reute, E. Scheepers, Y. Yang and B. Zhou, Computational Fluid Dynamics Simulation of Pyro metallurgical Processes, Fifth International Conference on CFD in the Process Industries CSIRO, Melbourne, Australia , 2006, 13-15
- [12] P. Diwakar , V. Mehrotra, , R. Vallavanatt, Troubleshooting Furnace Operations using Computational Fluid Dynamics (CFD), Proceedings of ASME-PVP' 04 5th International Bi-Annual ASME/JSME Symposium on Computational Technology for Fluid/Thermal/Chemical/Stressed Systems with Industrial Applications an Diego (La Jolla), California, 2004
- [13] E. H. Chui , Applications Of CFD Modeling In Canadian Industries, Fourth International Conference on CFD in the Oil and Gas, Metallurgical & Process Industries, SINTEF / NTNU Trondheim, Norway, 2005
- [14] R. Mehta and S. Sahay, Heat Transfer Mechanisms and Furnace Productivity During Coil Annealing: Aluminum vs. Steel, ASM International, 2008
- [15] Doe, Fundamentals handbook thermodynamics, heat transfer and fluid flow( Volume 1, 2 and 3, U.S. Department of Energy FSC-6910, Washington, D.C. 20585)