INTERNATIONAL JOURNAL OF
ENERGY AND ENVIRONMENT

Volume 2, Issue 5, 2011 pp.783-796
Journal homepage: www.IJEE.IEEFoundation.org

Experimental investigations and CFD study of temperature distribution during oscillating combustion in a crucible furnace

J. Govardhan[1], G.V.S. Rao[2], J. Narasaiah[3]

[1] Department of Mechanical Engineering, AVN Institute of Engineering & Technology, A.P., India.
[2] Department of Mechanical Engineering, PIRM Engineering College, A.P., India.
[3] Department of Mechanical Engineering, PRRM Engineering College, A.P., India.

Abstract
As part of an investigation few experiments were conducted to study the enhanced heat transfer rate and increased furnace efficiency in a diesel fired crucible furnace with oscillating combustion. The results of experimental investigations of temperature distribution inside the crucible furnace during oscillating combustion are validated with the numerical simulation CFD code. At first pragmatic study of temperature distribution inside a furnace was carried out with conventional mode of combustion at certain conditions and later transient behavior similar to that is conducted with oscillating combustion mode with the same conditions. There found to be enhanced heat transfer rate, reduced processing time and increased furnace efficiency with visibly clean emissions during the oscillating combustion mode than the conventional combustion mode. In the present paper the temperatures inside the furnace at few designated points measured by suitable K type thermo-couples are compared with the CFD code. The geometric models were created in ANSYS and the configuration was an asymmetric one for computational reason. The experimental and numerical investigations produce similar acceptable results. The presented results show that the 3D transient model appeared to be an effective numerical tool for the simulation of the crucible furnace for melting processes.

Keywords: Temperature distribution; Oscillating combustion; crucible furnace; furnace efficiency; heat transfer.

1. Introduction
In view of the impact on economy due to ever increasing energy prices globally and problems associated with global warming with the methods of energy utilization especially in the melting processes, there is a clear need for the heat transfer industries to focus on energy efficient methods and implementation of new technologies. The proposed new technologies shall be capable of utilizing variety of fuel resources with optimum release of heat energy and low emissions. Conventional combustion is generally used in the heat transfer industries for various melting operations. These systems using air-fuel mixture for combustion can be changed into oscillating combustion mode by introducing oscillations in the fuel flow rate as a parameter to improve the furnace performance. The furnaces which operate at high temperature produce large quantities of emissions are sometimes less productive and less efficient. There is a need to develop a technology that reduces emissions while increasing thermal efficiency for any furnace. John. C

and Wagner, in his NOx emission reduction by oscillating combustion technology urged industries to make their furnaces less polluting and more productive, whether they are firing with ambient-temperature, pre-heated air, oxygen enriched air, or oxygen [1]. Optimizing emission levels from combustion operations, which include a system for optimizing combustion, was generated by oscillating the fuel by an oscillating valve is a recognized technology for the reduction of NOx and improvement of furnace efficiency in industrial furnaces [2]. Oscillating combustion is an innovative technology which utilizes forced oscillations which create alternately successive fuel-rich and fuel-lean flame zones within the furnace, leading to increased heat transfer rate. The increased heat transfer rate results in enhanced furnace efficiency and its productivity with reduction in fuel consumption. Oscillating combustion is relatively a simple process and a new methodology for overall improvement performance of an aluminum foundry furnace. Overall Equipment Effectiveness (OEE) of an oil fired furnace which is a metric for total productive maintenance initiative has been calculated from the experiments conducted. Delabroy O, Louedin O. *et al.* described that oscillating combustion system is a low-cost, low NOx, high efficient technology and can be integrated in any combustion system whose principle is based on a cyclical perturbation of the gas line [3]. The results of the experiments conducted during oscillating combustion led to significant reduction in fuel consumption, highly cost effective which results in revenue savings, enhanced heat transfer rate and increased furnace efficiency. To achieve these improvements it is important to ensure the thermal energy of the hot gases to be absorbed by the load and the furnace walls. Temperature distribution in the furnace is paramount for the optimization of the thermal energy. The thermal behavior of flame as well as combustion products is very complex due to turbulence, chemical reactions and radiative heat exchange. The intensity of heat transfer from hot gases to the load is a function of temperature distribution inside the furnace. Generally, the temperature distribution throughout a body varies with location and time. Temperature distribution in the crucible of a furnace is an important operation variable that is a function of the materials used in construction, temperature in the metal-refractory interface etc. [4]. The study of temperature distribution and changes of induction heating furnace can offer theory support to choose and determine a reasonable heating system in actual production by using numerical simulation [5]. The numerical simulation has been identified as a suitable tool for better understanding of the phenomena of turbulent combustion normally prevails inside the furnaces. Wei Dong has employed CFD technology as an effective computer simulation tool to study and develop the new combustion concepts, phenomenal and progresses in advanced industrial furnaces and boilers [6]. A 3D numerical simulation with experimental validation of a gas-fired self-regenerative crucible furnace was presented by Francisco cadavid et al. Turbulence, radiation and chemical reactions are simulated using the software Gambit V2 and Fluent V 6.2 [7]. The difficulty in measuring the load temperatures inside a reheating gas furnace may be circumvented by appropriate use of numerical models [8]. A general mathematical model was devised for the numerical simulation of heat transfer of the transient temperature distribution in the load during the heating and cooling periods is of interest for the design of the furnace and the metallurgical control of the process [9]. An approach based on the assumption of periodicity of the composite structure is utilized to develop a method for the calculation of transient temperature profiles in layered, fibrous or particulate composites. The results show that the predicted temperatures agreed well with the measured ones [10]. The CFD simulation was applied to investigate the cause of non-uniform heat distribution by the temperature measurement in the radiant section of a nine-burner heater [11]. Systematic experimentation with numerical simulation was carried out to study the distribution of temperature and CO concentration in a FLOX combustion chamber. To study periodically oscillating combustion processes, turbulent heat release with slow chemical kinetics (turbulence-chemistry interaction), thermal and mechanical interaction of combustion chamber walls with hot gas flows, thermal radiation and pollutant formation in flames (e.g. soot, NO_x), Dr.-Ing. habil. B. Noll carried out in the Numerical Simulation research at the Institute of Combustion Technology, Stuttgart [12]. This present investigation describes the experimental study and numerical modeling of the oscillating combustion in a crucible furnace and also presents the experimental measurements validity with the results of numerical simulations. The geometric models were created in ANSYS and the configuration was an asymmetric one for computational reason. The geometric models created in 3-D to show different temperature points in a furnace. The boundary conditions for 13:1 air-fuel ratio and standard k-ε equation are chosen for modeling to generate better results in this case. Results show that the reasonable temperature distribution of the furnace can be obtained by optimizing the arrangement of the designated points in the furnace combustor according to the experimental measurements and numerical results to make the furnace combustor structure design more efficient. Experimental results were validated with the numerical simulation and good agreements were found between simulation and experimental measurements.

2. Computational model

2.1 Points of temperature measurements

T_1 = Temperature of the aluminum load in the crucible during the melting operation.

T_2 = Temperature of the hot gases 10 cm away from the inner walls of the combustion chamber of the furnace.

T_3 = Temperature of the hot gases at the entry of the stack.

2.2 Model geometry and mesh

The geometry of the crucible furnace along with the crucible for melting process used in this study is shown as Figure 1. The total furnace volume = 0.0829 m^3, Number of test points = 3, Chromel-Alumel (K-type) thermocouples used - Temperature range of -18^{0}C to 1372^{0}C; Accuracy = ± 0.5 % ; Sensitivity = 40 μV/^{0}C. A sensing probe and Digital temperature indicators used to read the temperatures. In order to understand temperature distribution in the furnace for optimization during the oscillating combustion mode five air-fuel ratios (13:1, 14:1, 15:1, 16:1 and 17:1) each two above and two below the stoichiometric ratio with three loads of aluminum were tested. Numerical simulation was carried out for 13:1 air-fuel ratio at 20 kg of load since better results were achieved in this case.

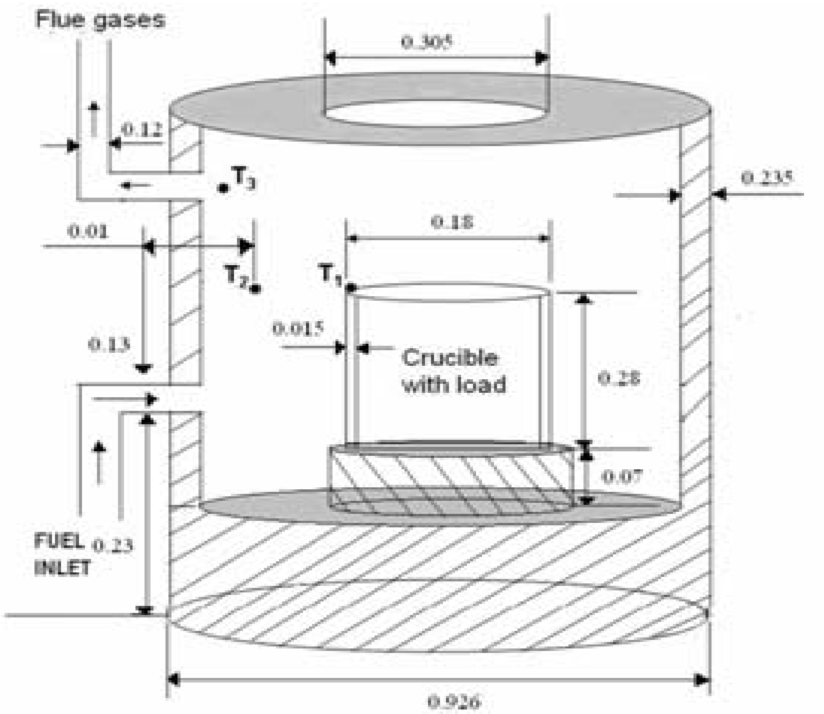


Figure 1. Schematic diagram of the crucible furnace for model geometry (in metre)

2.3 Basic governing equations

Conservation equations of mass can be written as

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho \upsilon_i)}{\partial x_i} = S_i \tag{1}$$

where Si is the mass source in the system. For multi component system, the mass balance can be expressed as

$$\frac{\partial(\rho m_i)}{\partial t} + \frac{\partial(\rho u_i m_i)}{\partial t} = -\frac{\partial j_{i',i}}{\partial x_i} + R_{i'} + S_{i'} \tag{2}$$

where m_i is a local mass fraction of each species in the system, j_i is a diffusion flux of species i' which arises due to the concentration gradients, R_i is the mass rate of creation or depletion by chemical reaction, S_i **:** is the mass rate of any other sources.
For laminar flows of dilute gas system, the diffusion flux meets the Fick's law as.

$$j_{i',i} = \partial D \, D_{i',m} \frac{\partial m_i}{\partial x_i} \tag{3}$$

where $D_{i',m}$ is the diffusion equations of momentum can be described as Navier-Stokes equations as

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + s_{mi} \tag{4}$$

where ρ the static pressure is τ_{ij} is the stress tensor. s_{mi} is the momentum source in *i* direction.

The stress tensor τ_{ij} is given by $\tau_{ij} = \mu(\frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_i}) - \frac{2}{3}\mu \frac{\partial u_i}{\partial u_j} \rho_{ij}$ (5)

where μ is the molecular viscosity.
Conservation equations of energy equations can be written as

$$\frac{\partial(\rho h)}{\partial t} + \frac{\partial(\rho h u_i)}{\partial x} = \frac{\partial p}{\partial t} + u_i \frac{\partial p}{\partial x_i} - \frac{\partial(k \frac{\partial T}{\partial x_i})}{\partial x_i} - \frac{\partial h_i j_i}{\partial x_{ij}} - \tau_{ik} \frac{\partial u_i}{\partial x_k} + s_h \tag{6}$$

where: $h = m_j h_j$

$$h_{j'} = {}_{Tref}^{T} c_{pj} \, dT \tag{7}$$

The energy source due to chemical reaction can be expressed as

$$S_{h,reaction} = \Sigma_j \left| h_{f'}^{o} + {}_{Trefj'}^{Tref} C_{pj} \, dT \right| R_{j'} \Gamma\Gamma \tag{8}$$

And the energy source due to radiation will be calculated in radiation models.

2.4 Transport equations for the standard k-ε model
In the k- ε model, the k and ε can be obtained from the following transport equations as

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right)}{\partial x_i} + G_k + G_b - \rho\varepsilon \tag{9}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho u_i \varepsilon)}{\partial x} = \frac{\partial\left[\left(\mu + \frac{\partial l}{\partial \varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]}{\partial x_i} + C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k + (1 - C_{3\varepsilon})G_b\right) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{10}$$

where G_k is the generation of k due to the turbulent stress as

$$G_k = -\rho u_i' u_j' \frac{\partial u_j}{\partial x_i} = \tau_{ij} \frac{\partial u_j}{\partial x_j} \tag{11}$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, is the generation of turbulence kinetic energy due to buoyancy. $C_{1\varepsilon}, C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε , respectively.

2.5 Modeling of turbulent viscosity

The turbulent viscosity, μ_t can be evaluated by k and ε as

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{12}$$

where C_μ is a dimensionless constant.

In the k- ε model, the k and ε can be obtained from the turbulent transport equations as

$$G_k = \mu_t S^2 \tag{13}$$

where S is the modulus of the mean strain rate Sij as

$$S \equiv \sqrt{S_{ij} S_{ij}} \tag{14}$$

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{15}$$

$$G_b = \beta g_i \frac{\mu_t}{p_{rt}} \frac{\partial T}{\partial x_i} \tag{16}$$

where P_{rt} is the turbulent prandtl number for temperature or enthalpy (0.85), and β is the coefficient of thermal expansion as

$$\beta = \frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p \tag{17}$$

The standard *k- ε* model constants $C_{1\varepsilon}, C_{2\varepsilon}, C_\mu, \sigma_k, \sigma_\varepsilon$ have the following standard values as

$$C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$$

2.6 The standard k- ε turbulence model

A brief description of the standard k-**ε** model is included. The flow was assumed to be turbulent the effects of molecular viscosity are negligible. The standard k-ε model is therefore valid for fully turbulent flows. The simplest models of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k- **ε** model in FLUENT falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding and J.S.Woo, J. Szekey et al [13]. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model.

3. Numerical procedure

ANSYS FLUENT version 12, CFD commercial software was used which solves the governing equations based on the boundary conditions numerically by the finite volume method. The geographical model was created and the configuration was asymmetric and three dimensional since turbulence has to be treated as three-dimensional. Figure 2 shows the grid structure to mesh the model for numerical simulation. A structured 3D mesh with 29,563 quadrilateral cells applied on the domain considered. To ward off the expected sudden changes in fluid flow close to the burner exit and around the fuel nozzle (mixing and reaction zone) the grid was refined. To optimize the performance effects of oscillating combustion attention was drawn towards the temperature distribution and the heat transfer rate. The standard k-ε-model was used to describe the effect of turbulence in the flow field. For the description of the diffusion combustion a model was used which solved the transport phenomena of every relevant gas phase species. Eddy dissipation model was used to account the effects of turbulence on the chemical conversion rates. P1 model was used for the radiative heat transfer in the furnace combustor. The standard wall functions option for the near wall treatment was applied as well as the no slip condition at the wall. At the chamber wall no slip boundary conditions and no species flux normal to the wall surface are applied. The thermal boundary condition on the chamber wall is observed as adiabatic wall condition. A rapid convergence has been reached at 10^{-8} accuracy with implicit multi-grid and a coupled solution of the momentum and continuity equation. The inlet temperature of air is considered to be uniform at 303K. A fixed uniform velocity for steady state 58 m/s is specified at the air inlet. The velocity of fuel during the oscillating combustion was 13 m/s is specified at the diesel inlet with ambient condition and the mass flow rate of fuel was 1.32 g/s.

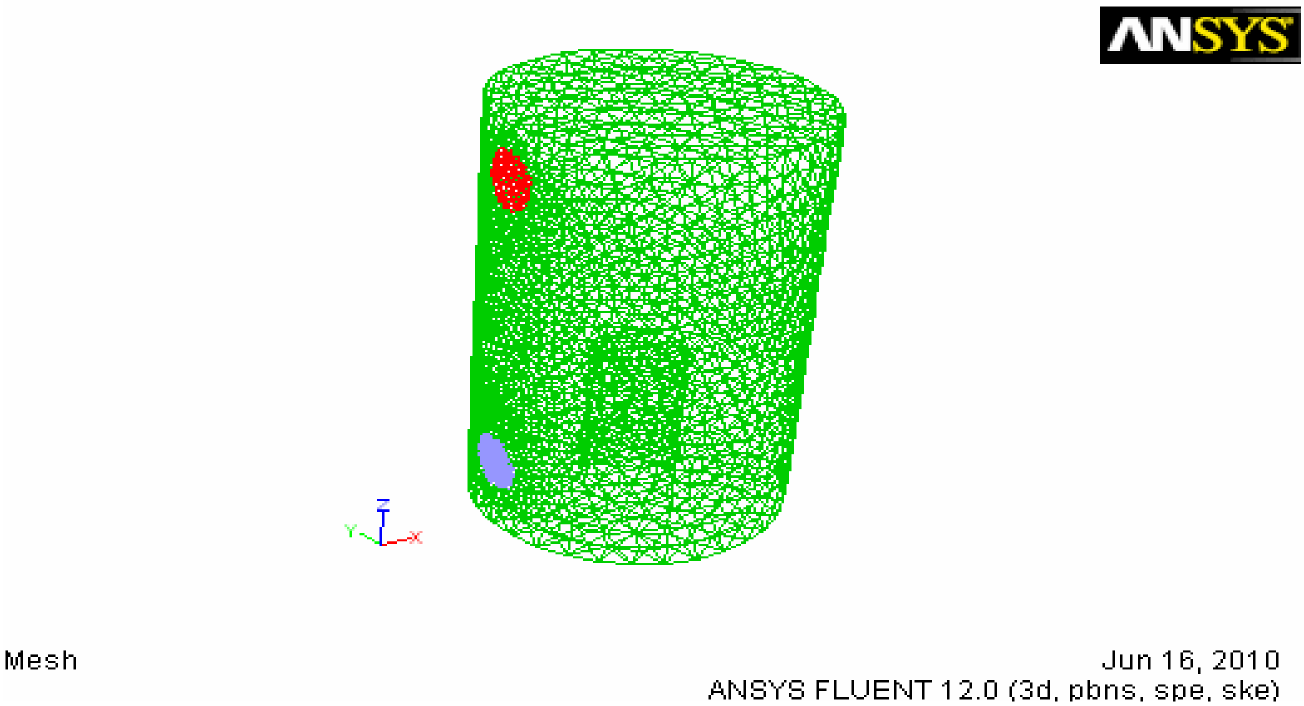


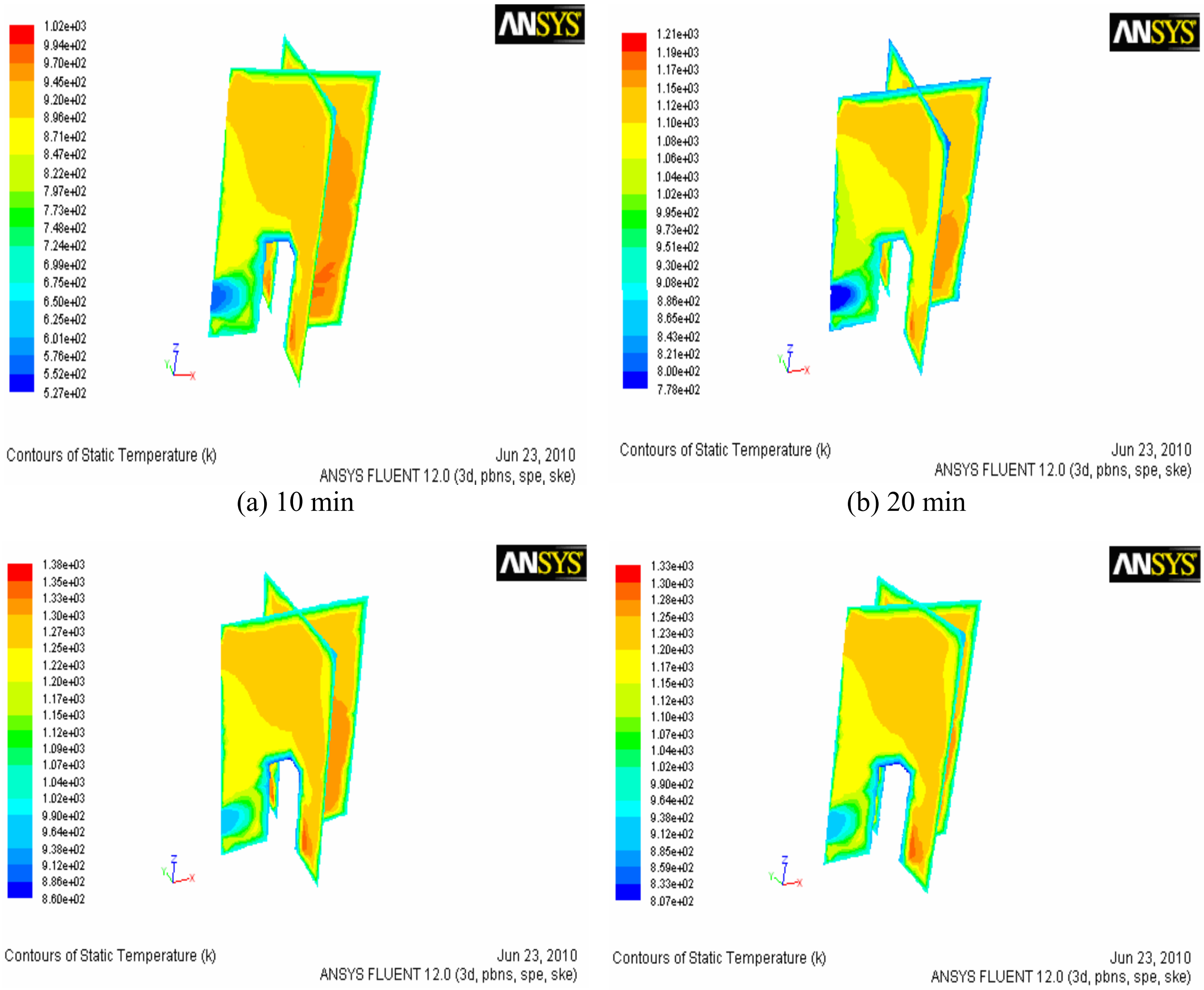
Figure 2. Computational model- grid mesh

3.1 Boundary conditions

Combustion air inlet temperature	Ambient
Combustion air inlet velocity	58 m/s (constant)
Fuel Used	Diesel ($C_{10}H_{22}$)
Fuel inlet temperature	Ambient
Fuel inlet velocity	13 m/s
Mass of fuel	1.32 g/s
Air-fuel ratio	13:1
Time-step	10 minute
Burner and combustor	Asymmetric, Three Dimensional
Commercial CFD package	ANSYS FLUENT-12
Turbulent Model	Standard k-ε
Transport equation	for species transport
Formulation	Implicit
Chemical conversion rate/mixing	Eddy dissipation model
Radiation heat transfer	P.1 model
Conservation equations (for effect of fluctuations)	PDF (Double delta function)

3.2 Oscillating combustion temperature contours

The oscillating combustion temperature contours and the contours of path lines of temperature are shown in Figures 3 and 4.



(a) 10 min (b) 20 min

(c) 30 min (d) 40 min

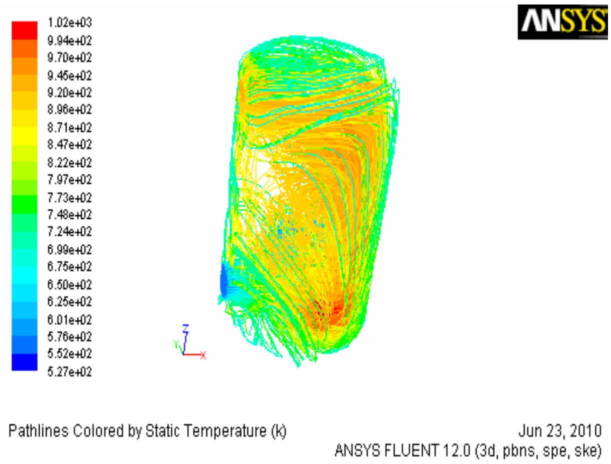
Figure 3. Temperature contours (a, b, c, d)

4. Experimental results and discussions

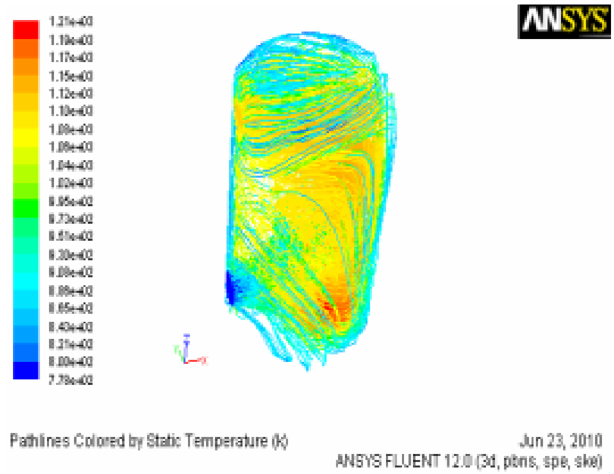
3D design modeled due to turbulent combustion by using a CFD code. This provides the information about the temperature distribution in the furnace otherwise difficult to characterize. The measurements are difficult to perform for practical reasons related to the operations and quite extreme fluctuating conditions in the crucible furnace combustor. Modeling validation is performed by the comparison of the following computed and experimental measured data. Tests are performed on crucible furnace with diesel as fuel at ambient conditions. Modeling was carried out on the furnace for temperature distribution in the furnace during oscillating combustion. The results of ANSYS FLUENT CFD code used for the analysis of turbulent flows to study temperature distribution have been validated against the experimental results.

4.1 Temperature distribution during the oscillating combustion mode

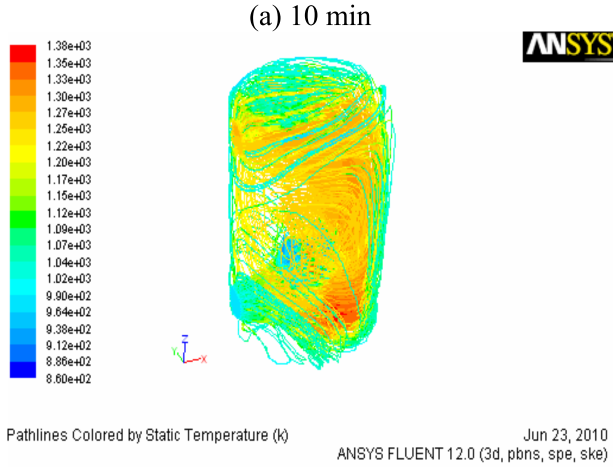
Thermal convection is an important factor for heat and mass transfer from the hot gases in the furnace to the load during the oscillating mode. Temperature distribution in the furnace is completely coupled with the flow, because the load melt density is the function of temperature. Strong, turbulent flow heat exchange is expected in the furnace while the thermal conductivity effect should be reasonable during this mode of combustion. During this simulation and experiments the mass flow rate of fuel was perturbed deliberately causing oscillations in the flow resulting in oscillating combustion. The inlet velocity of the fuel was changed where as the combustion air inlet velocity kept constant. During the oscillating mode of combustion the turbulent flame travels rapidly around the furnace and the high turbulence and high temperature fields of oscillating combustion flame transfer thermal energy into the load by impingement and due to more luminous and fuel rich zone flame. Enhanced heat transfer rate to the load was observed and this can be attributed to the existing temperature gradients in different zones of the furnace combustor during the oscillating combustion.



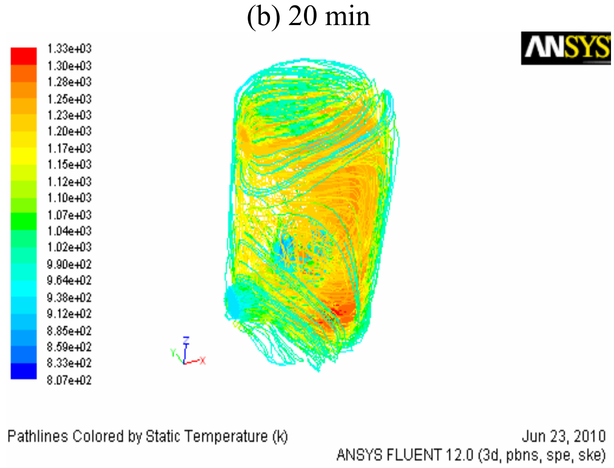
(a) 10 min



(b) 20 min



(c) 30 min



(d) 40 min

Figure 4. (a-d) Contours of path lines of temperature

4.2 Validation

The time-averaged value was used to present the temperature at the chosen points in the furnace combustor. The time-averaged temperature fields are shown for experimental and numerical simulation. The oscillating combustion temperature contours and the contours of path lines of temperature are shown in Figures 3 and 4. The temperature distribution during the oscillating combustion was compared with the experimental results which are summarized in Table 1 and Figure 5 for the temperatures T1, T2 and T3. Experimental results are shown with deviation percentage with the CFD results. Figure 5 shows the deviations of experimental measurements with numerical results and also the magnitude of temperatures obtained during oscillating combustion. The T1 temperature point for experimental and simulation predictions are in quite convincing and the deviations are negligible. But there are few deviations noticed at T2 and T3 which are discussed as.

Table 1. Temperature values for T1, T2 and T3

Time	T1 (K)		T2 (K)		T3 (K)	
(min)	Exp	CFD	Exp	CFD	Exp	CFD
10	575	594	621	871	735	871
20	778	843	791	1040	1043	1080
30	921	912	977	1150	1051	1150
40	974	938	1053	1170	1123	1222

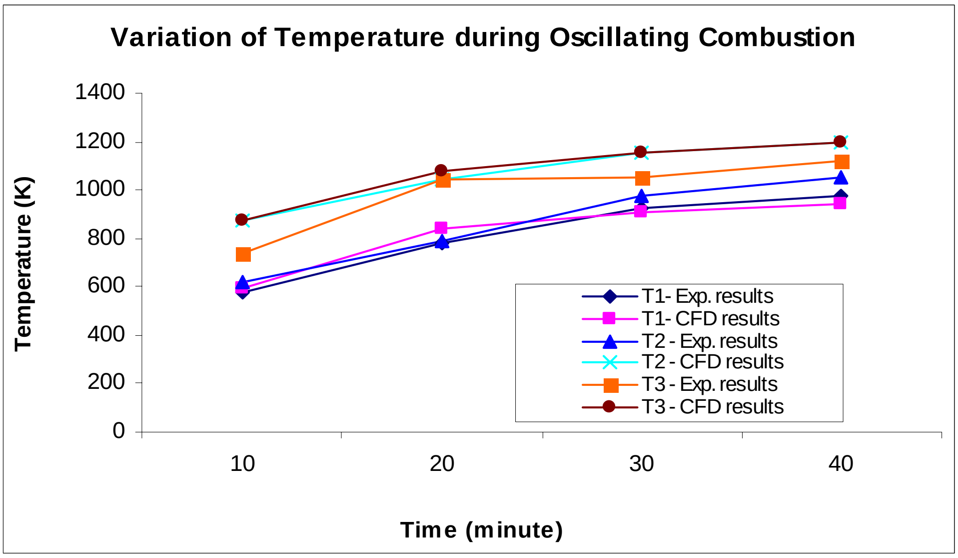


Figure 5. Comparison of experimental and CFD results

4.3 Comparison of experimental measurements and CFD results on temperature distribution

It can be seen from Table 2 and Figure 6 that for T1 the experimental measurements and CFD results are hand in hand, grows almost linearly from start to end with minor deviations around 3.1% to 7.7% which can be treated as small. But from Table 3 and Figure 7 the deviation found to be in disagreement for T2, which varies from 28.7% to 23.9% from beginning to 20 minutes of time interval. Perhaps this may require some finer meshing for flow simulation. The use of eddy-dissipation model for combustion seems to cause some over predictions. It also can be seen from Table 4 and Figure 8 that the temperature T3 was found to be high at all time-intervals but the deviation was found to be in control. This is due to the fact that most of the radiation flux was available at this designated point and near the furnace wall. Predictions of CFD temperatures are also seem to be closer to the experimental values. The deviation was found to be slightly high in the beginning around 15.6% but changed to 3.42% to 8% during the next part of the time-steps.

Table 2. Deviation rate for temperature T1

Time (min)		10	20	30	40
T1 (K)	CFD	594	843	912	938
	Exp.	575	778	921	974
	Deviation %	3.1	7.7	-0.9	-3.8

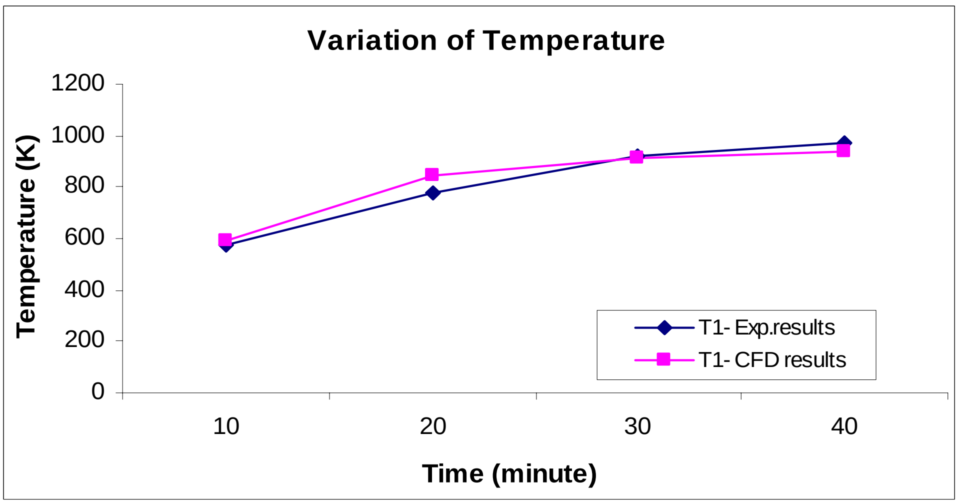


Figure 6. Comparison of temperature T1

Table 3. Deviation rate for temperature T2

	Time (min)	10	20	30	40
T2 (K)	CFD	871	1040	1150	1200
	Exp.	621	791	977	1053
	Deviation %	28.7	23. 9	15	12.2

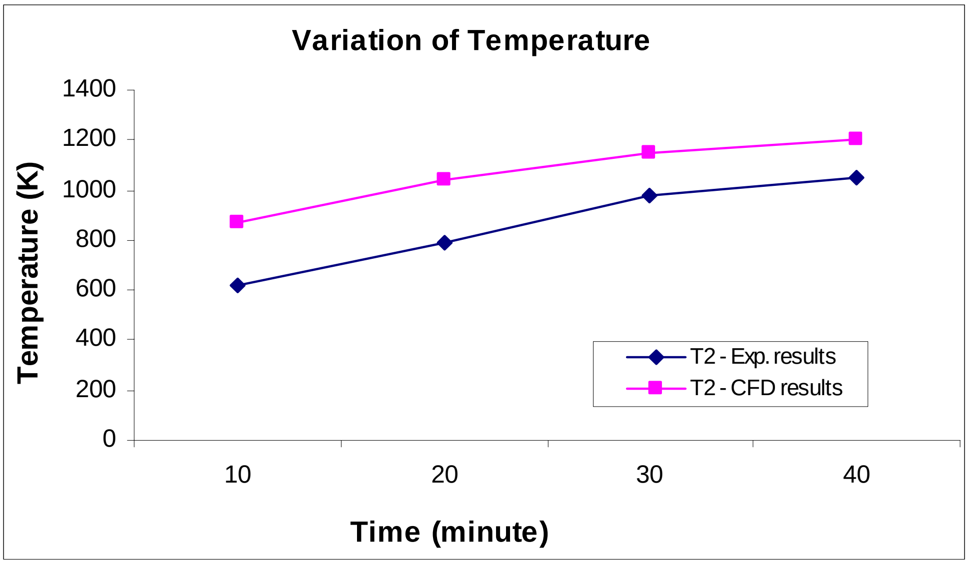


Figure 7. Comparison of temperature T2

Table 4. Deviation rate for temperature T3

	Time (min)	10	20	30	40
T3 (K)	CFD	871	1080	1150	1222
	Exp.	735	1043	1051	1123
	Deviation %	15.5	3.42	8.6	8

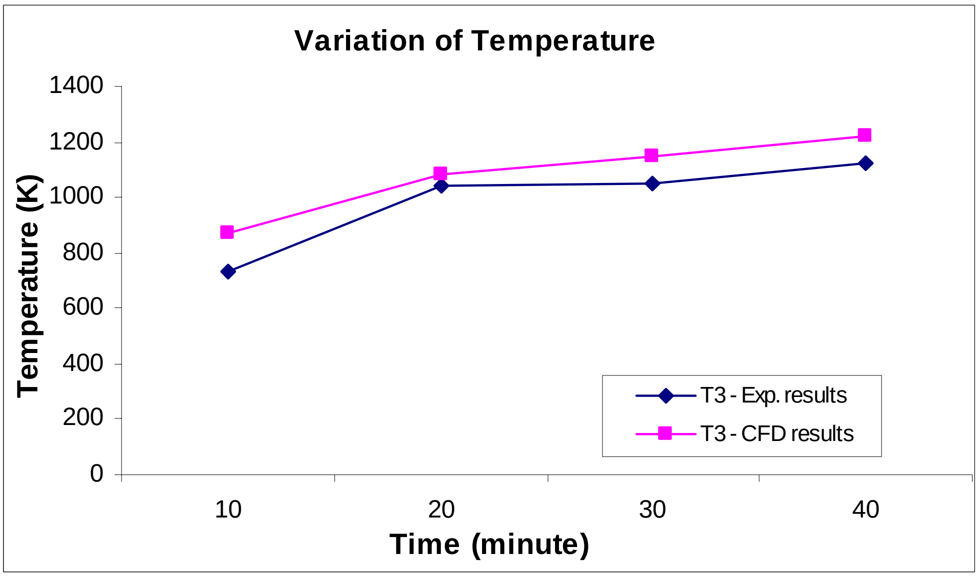


Figure 8. Comparison of temperature T3

There found to be slight disagreements or deviations of experimental results from the CFD simulation results. It was noticed that the experimental results were in good agreement with CFD predictions for the boundary conditions chosen during steady state combustion mode. However, some disagreements were noticed during the oscillating combustion mode. Though the deviations are slightly high but are well in control. In the beginning temperature point T2 was found to be with a deviation of 28.7% which is disagreeable but reduced to 12.25% by the end of the process time. Also, the temperature point T3 was seen with a deviation of 15.6% in the beginning but was well in control during the next time-steps of the operation. The noticeable changes during oscillating combustion were due to the change in the boundary condition of the inlet velocity of the fuel for the numerical simulation. Based on the change, the predictions of the CFD computations were found to be high. However, the temperatures of the same designated points for T2 and T3 during the tests were found to be low during oscillating combustion mode. This can be attributed to the phenomenon of oscillating combustion during the experiments. It can also be stated that the salient features of the oscillating combustion are such that due to the oscillations in fuel flow the combustion becomes oscillated which results in alternately successive fuel-rich and fuel-lean flame zones. As said earlier, the fuel-rich zone is highly luminous, radiative and turbulent, longer in length and enhances the heat transfer rate to the load. The fuel-rich and fuel-lean zones eventually mix up in the furnace combustor after the heat has been transferred from the fuel-rich zone to the load and escapes in to the stack. This results in the reduction in the crown temperature of the furnace combustor thereby reduced temperatures of the designated points which are seen from the experimental values. Also, the slight disagreements between the tests and modeling may be assumed by

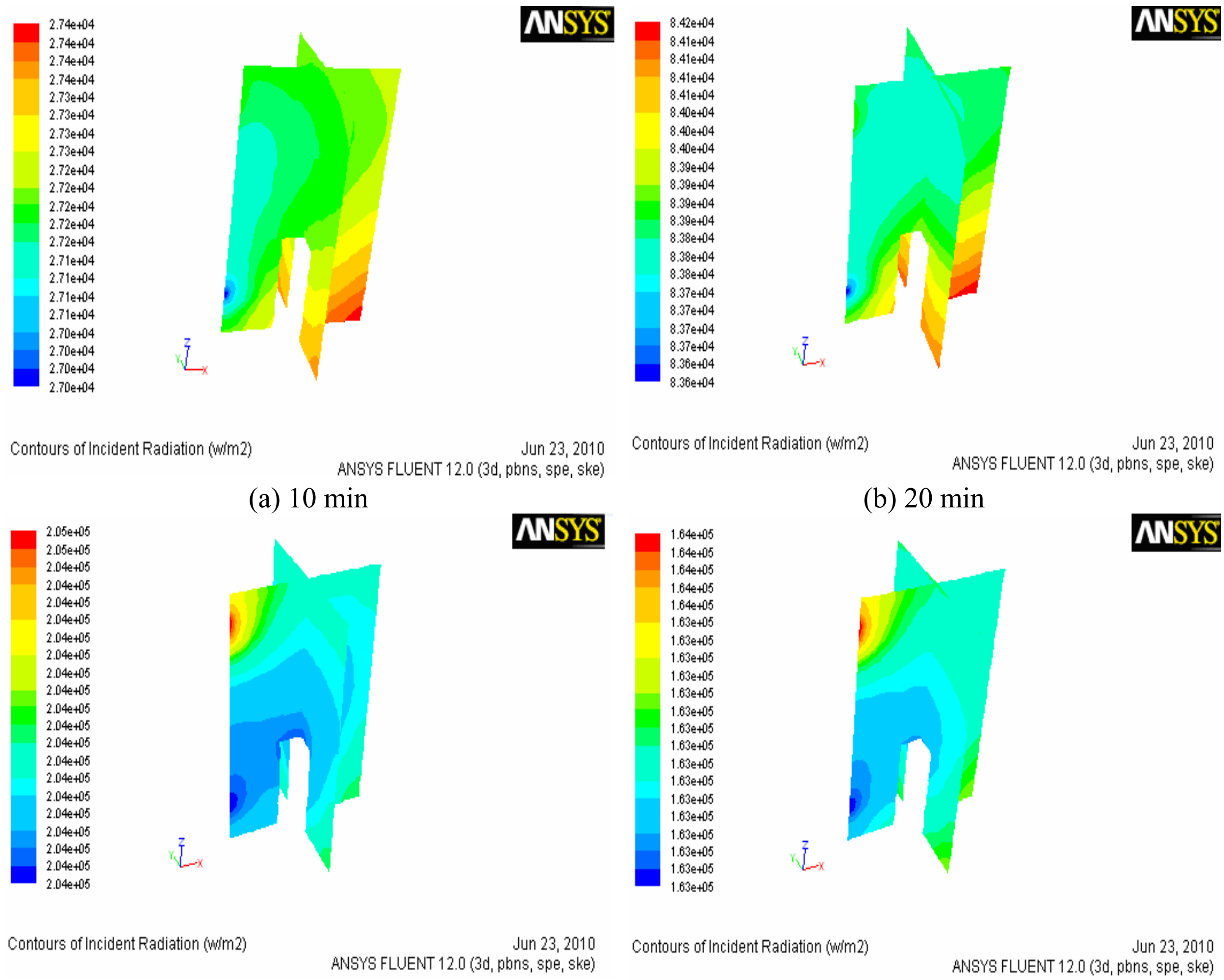
- The inadequate boundary conditions of the inlet fuel and combustion air velocities and flow rates. Sometimes due to the small variations in the boundary conditions the simulation results may appear to be varying with those of measured during the experiments.
- A portion of the error could also be addressed due to the measurement errors due to the deterioration of thermo-couples and escape of hot gases resulting in heat loss.
- The other portion of errors arises from the computational errors caused by the assumptions in the turbulence, combustion and the radiation models adopted to simulate the problem.
- Attention has to be drawn towards the evaluation of thermal conductivity of the silicon graphite crucible insulation that affects dramatically the thermal field in the furnace. Nevertheless, the CFD measurements are in good agreement with the experimental results.

4.4 Numerical analysis of radiation contours

Table 5 presents the numerical results of three-dimensional radiation contours obtained from the Figure 9 (a-d) during oscillating combustion mode and are presented for the radiation at the entry, middle and at the exit points in the furnace combustor. The results show that enhanced heat transfer rate was observed during the oscillating combustion mode. This was due to the high turbulence and luminous flame zone of the oscillating combustion providing enhanced heat transfer rate. There is a noticeable variation in the magnitude of radiation in the radiation profiles. This may be due to the assumed heat transfer coefficient in the boundary conditions for the radiation model, since the heat transfer coefficient depends upon the thermal conductivity and diffusion coefficient of the flow field.

Table 5. Radiation contours –numerical simulation

Radiation (W/m^2)	Oscillating Combustion mode		
Zone & Time	Entry	Middle	Exit
10 min	2.7xe4	2.72xe4	2.74xe4
20 min	8.36 xe4	8.38 xe4	8.38 xe4
30 min	2.04 xe5	2.04 xe5	2.04 xe5
40 min	1.63 xe5	1.63xe5	1.64 xe5



(a) 10 min (b) 20 min

(c) 30 min (d) 40 min

Figure 9 (a-d). Contours of incident radiation

5. Conclusion

The following conclusions were arrived from the experimental investigations.

- Temperature difference is a potential for driving the radiation flux. The temperatures at the load observed during the oscillating combustion indicates high heat transfer rate.
- Heat transfer rate in the furnace found to be more during oscillating mode due to oscillations in the fuel flow rate and these conditions are important in many applications of heat transfer industries.
- The increase in heat transfer rate is from 2.72% to 8.45% and increase in furnace efficiency from 2% to 6.54%.
- Fuel savings increases from 7% to 28% during the oscillating combustion mode depending upon the operating condition than conventional combustion mode.
- There was considerable increase in furnace efficiency during the oscillating combustion mode.
- The melting time was lower with oscillating combustion mode.
- Decrease in 19 minutes of melting time was observed during oscillating combustion mode.
- During the oscillating combustion mode, the combustion process was extended over a larger volume in the furnace with fuel-rich and fuel-lean zones of combustion and subsequent mixing results in low furnace crown temperature, reduced stack gas temperature and reduced NOx formation.
- The experimental temperature contours validation with the CFD numerical simulation predictions was found to be in good agreement.
- Visibly apparent clean and reduced flue gas volumes with low emissions were observed.

Acknowledgements

The author is grateful to the Management of A.V.N. Institute of Engineering & Technology, Ibrahimpatnam (M) for their support and P.R.R.M. Engineering College, Shabad, R. R. Dist., Andhra Pradesh, India., for providing the facilities for the execution of this experimental analysis in the Production Technology Laboratory of the Department of Mechanical Engineering.

References

[1] John, C and Wagner. NOx emission reduction by oscillating combustion. GTI, 2004.

[2] Rajani Varagani K. Advanced control system for enhanced operation of oscillating combustion in combustors. Air Liquide-Houston, TX,US 2004.

[3] Delabroy O, Louedin O. et al. Oxy combustion for reheat furnace. Joint International Combustion Symposium 2001: 5: 217-240.

[4] Luis Felip Verdeja, Roberto Gonzales and Alejandro Ordonez. Using FEM to determine temperature distribution in a blast furnace crucible. JOM, Feb 2000.

[5] Shu-guangFu. The Numerical Analysis of Induction Heating Furnace Temperature Field of numerical simulation to study the temperature. Computational Intelligence and Software Engineering, CiSE 2009, International Conference, pp 1-5, 2009.

[6] Wei Dong. Design of Advanced Industrial Furnaces using Numerical Modeling Method. Heat and furnace technology – Royal Institute of Technology, Stockholm, Sweden.

[7] Francisco cadavid, Bernardo Herava and Andres Amell. Numerical simulation of the flow streams behavior in a self regeneration crucible furnace. Science and technology of gases and rational use of energy group, Columbia, 2009.

[8] P.Marino, A.Pignotti and D.Sollis. Numerical model of steel slab reheating in pusher furnace, Latin American applied research.32:257-262, 2002.

[9] Sergio Pissanetzky, Numerical simulation of the transient temperature distribution inside a close-packed array of cylindrical tubes during heating and cooling under high vacuum. Nuclear Engineering and Design, Volume 56, Issue 2, pp. 59-368, 1980.

[10] Zbigniew M. Grabowski, Eric H. Jordan, and Kevin P. Walker. Transient temperature distribution in a composite with periodic microstructure. Composites engineering, Volume 4, Issue 11, pp. 1055-1072, 1994.

[11] Eddy h. Chui, Applications of CFD modeling in Canadian industries, Fourth International NTNU Trondheim, Norway 6- 8 June 2005.

[12] Dr.-Ing. habil. B. Noll. Numerical Simulation, German Aerospace Center, Institute of Combustion Technology, Numeric Simulation, Stuttgart.

[13] B.E Launder, D.B.Spalding and J.S Woo, J. and Szekely. Lectures in mathematical models of turbulence, A study on the mathematical modeling of turbulent recirculating flows in gas-stirred ladles. Journal of metallurgical and material transactions B, Vol 21, No.2, pp. 269-277, 1990.

J. Govardhan is a Ph.D candidate in the Department of Mechanical Engineering, Osmania University. He obtained his Bachelor's degree in Mechanical Engineering from Institution of Engineers (India) in 1991 and Master's degree from Delhi University in 1996. His area of research is Oscillating Combustion. He developed an oscillating valve and incorporated it in the test equipment and studied the effects of oscillations during the combustion and its performance characteristics such as process time, specific energy consumption, thermal efficiency and emissions. Applied CFD numerical simulations and compared the experimental results with that of CFD predictions and the experimental results are found to be in good agreement. He has published few papers in International Journals and also presented papers in International Conferences.

E-mail address: govardhan58@yahoo.co.in

GVS. Rao graduated in Science and Engineering from Andhra University in 1996 and 1969 respectively. He received his Master's degree (M.E) with Thermal Science as specialization from Regional Engineering College, Warangal in 1972 under Osmania University. He pursued his Doctorial Work and received Ph.D from Indian Institute of Technology, Madras in 1978. He served in Research and Development of Bharat Heavy Electricals Limited (BHEL) and designed, developed combustion system for M.H.D Power generation of a pilot plant, and National Project with participation of BHEL and BARC of DAE. For 23 years in BHEL served in different Engineering areas and R&D. His main areas of interest are combustion, gasification and new energy developments. He has published many articles in various International Journals and Conferences.
E-mail address: termarrow@yahoo.com

J. Narsaiah, working as Assistant Professor in PRRMEC, Shabad. He has done his M.Tech in Industrial Engineering and Management from JBIET (JNTU Hyderabad) in 2009 and B.Tech., in Mechanical Engineering from Vasavi College of Engineering (OU) in 2003. His area of interest is in the field of combustion and reduction in emissions.
E-mail address: jaligari_jns@yahoo.co.in