Numerical analysis of the transient heat transfer in high temperature chamber furnaces

K S Krumov and N Y Penkova
University of Chemical Technology and Metallurgy, Sofia, Bulgaria
E-mail: kkrumov@uctm.edu

Abstract. An algorithm for numerical analysis of the transient thermal processes in high temperature gas chamber furnaces for ceramic firing is developed. It is based on mathematical models of the conjugate heat transfer, including combustion, turbulent flows of the torch and gas mixtures, heat transfer through radiation, conduction and convection. The algorithm is used for analysis and improvement of the efficiency of a furnace for firing of technical ceramics.

1. Introduction
The intermittent chamber furnaces are widely used in the ceramic industry because of the possibilities for flexible thermal regimes for firing of different ceramic ware. The firing process includes heating and cooling of the production by increasing, keeping and decreasing of the temperature in the furnace space with the time (so-called temperature curve). That variation depends on the ceramic material, geometry of the fired articles and their arrangement in the space. The non-stationary temperature field in the furnace usually is maintained by means of an adjustable combustion process at unsteady fuel and airflow through the burners. The efficiency of such thermal aggregates is relatively small due to the high thermal losses to the environment by the exhausted gases, heat exchange with the environment and heat accumulation by the furnace envelopes [1]. It also depends on the temperature and velocity fields in the furnace space – they have to be uniform to ensure faultless firing of the ceramics. Computational fluid dynamics and heat transfer, based on finite volume method are useful tools for investigation of the possibilities for improvement of the efficiency of chamber furnaces [2, 3, 4 and 5]. The complete finite volume analysis of the conjugate heat transfer can be performed at a geometrical model, including the solid and gaseous domains of the furnace. It requires large computer resources and computational time at fine finite element meshes and minor steps of the unsteady firing process. Such complex study of the coupled combustion and heat transfer in chamber furnaces at industrial operating conditions was not published so far. An algorithm for investigation of the transient thermal processes in periodically working chamber furnaces via different approaches is presented in this paper. It is validated and applied for improvement of the construction of a gas furnace for firing of technical ceramics.

2. Conceptions for modelling and numerical analysis of the transient heat transfer in gas chamber furnaces
The conjugate heat transfer in the gas chamber furnaces covers coupled transient processes:
- combustion at unsteady turbulent fuel and air flows;
- heat transfer by convection, radiation and conduction from the hot gas mixture to the furnace envelopes and fired ware, arranged on auxiliary refractory construction;
- heat transfer by conduction in the solid media. Usually it is accompanied by endothermic and
exothermic chemical and phase change processes in the fired materials;

- surface to surface radiation heat exchange by the solid surfaces, seeing each other.

For a complex analysis of the processes above the geometrical model has to include:

- gas domain of the internal furnace space and the burners where fuel and airflows are fed and combustion proceeds (Part 1);
- solid domain of the ware to be fired, arranged on auxiliary refractory construction (Part 2);
- solid envelopes of the furnace: multilayer walls, floor and arch, made by refractory bricks and insulation (Part 3).

The geometrical model is discretized by a finite element mesh – finer in the fluid regions and coarse in the solid regions. The different domains have to share common faces in order to simplify the mesh and the connections between the nodes at the interfaces. The equations below are solved to model the transient conjugate heat transfer in the furnace.

- **Fluid domain (Part 1)**: continuity equation, momentum equations, energy equation, standard κ-ε model turbulence model, boundary layer model [6, 7], ideal gas relations for the air, fuel components and products of combustion, eddy dissipation combustion model and radiation model P1 [8].

- **Solid domains (Part 2 and 3)**: energy equation with non-zero heat generation rate in the ceramic bodies to be fired to model the thermal effects of endo- and exothermic processes in the material (if there exist any). The surface to surface radiation exchange can be neglected at relatively near temperatures of the solid surfaces.

The temperature in the furnace changes several times during the firing regime: the physical properties of the gases, surrounding elements, furnace masonry and thermal treated products have to be accepted as functions of the temperature.

The initial values of the variables to be computed can be accepted as follows:

- zero velocity components and pressure (according a reference pressure) for each node in the gas space;
- zero mass fractions of fuels components and products of combustion in the gas space: it contains air only (nitrogen and oxygen);
- initial temperatures of the gas and solid space according to the technology stages and the temperature curve.

The models can be solved numerically by CFX of FLUENT at ANSYS Workbench [8] at the boundary conditions in Table 1. The fields of the pressure, gas fractions, velocity components, turbulence parameters and temperatures in the fluid and solid space are obtained as results. The heat fluxes and heat transfer coefficients at all fluid/solid interfaces are computed as output derived variables. That approach (named Variant 1) requires large computer resources and computational time.

Another way to solve the problem (Variant 2) is to generate separate geometrical models of the internal space and furnace envelopes and to simulate consequently the heat transfer in them to calibrate the boundary conditions at the interface between the gas space and the walls. Variant 2 is implemented at the steps below.

Generation of geometrical model of Part 1 and 2 (internal furnace space), and a distinct model of Part 3 (solid furnace envelopes).

Numerical simulating of the transient heat transfer in the furnace envelopes (Part 3) according to the boundary conditions in Table 2. The heating and cooling rates are computed according to the temperature curve, accepted at the technology. Two subsequent simulations have to be done. The final nodal temperatures after the first simulation are used as initial ones for a second numerical simulation of the heat transfer. The variation of the heat fluxes to the internal surfaces with the time is determined as results of the second simulation. Their integration over the time gives the accumulated heat by the furnace envelopes and thermal losses to the environment.

Computation of the moment values of artificial heat transfers coefficients $U*$ for the furnace envelopes by equation (1):

$$U' = \frac{\dot{q}_{in}}{\Delta T}$$  \hspace{1cm} (1)
where: \(q_{st}\) - heat flux, W/m²; \(\Delta T\) - temperature difference between the internal temperature according the boundary condition \(T_0\) in Table 2 and the ambient temperature: \(\Delta T = T_0 - T_0\) 

Numerical simulation of the transient processes in the furnace space (combustion and conjugate heat transfer in Part 1 + Part 2) at the boundary conditions in Table 3.

Comparison of measured temperatures in fixed points in the gaseous domain to the computed nodal temperatures in the same points. In the case of differences exceeding accepted one, \(\Delta T\), used at the temperature boundary conditions in Table 2 are corrected. The steps 2, 3 and 4 are repeated until the measured and calculated temperatures are close enough - it is considered that the model of heat transfer with the environment is calibrated and validated.

Both approaches for numerical simulation are demonstrated on Figure 1.

**Table 1.** Boundary conditions for complex CFD analysis of the furnace (Part 1+Part 2+Part 3).

| Boundaries | Boundary conditions |
|------------|---------------------|
| 1. External surfaces of vertical walls, floor and arch | \(-\lambda \frac{\partial T_{st}}{\partial n} + h_{sc}(T - T_0) = 0\) \(h_{sc} = A \varepsilon T_{st} + \sigma (T^{4}_f - T^{4}_i)\) \(\Delta T = T - T_0\) where: \(h_{sc}\) - heat transfer coefficient by convection and radiation from the external wall surfaces to the environment, W/m²K⁻¹; \(T_0 = 293\)K; \(\lambda\) - thermal conductivity of the external layer, W/m²K⁻¹; \(\varepsilon\) - emissivity of the surfaces; \(\sigma\) - Stefan-Boltzmann constant; \(A\) is a constant that can be accepted as follows [5]: \(A = \begin{cases} \text{3.3 at arch} \\ \text{2.6 at vertical walls} \\ \text{1.6 at floor} \end{cases}\) |
| 2. Inlets | \(T_{air} = T_{air}(t)\) Velocity, normal to the boundary: \(V_{n, air} = V_{n, air}(t)\) Oxygen and nitrogen mass fractions: \(g_{O_2} = 0.22\); \(g_{N_2} = 0.78\) |
| 2.1 – Air inlets | Air temperature: \(T_{air} = T_{air}(t)\) Velocity, normal to the boundary: \(V_{n, air} = V_{n, air}(t)\) Oxygen and nitrogen mass fractions: \(g_{O_2} = 0.22\); \(g_{N_2} = 0.78\) |
| 2.2 – Fuel inlets | \(T_{f} = T_{in} = const.\) Velocity, normal to the boundary: \(V_{n, fuel} = V_{n, fuel}(t)\) Mass fractions of the i-th fuel components: \(g_i\) |
| 3. Exhaust gases outlet | \(p = 0\) Pa according a reference pressure |

**Table 2.** Boundary conditions for thermal analyses of the furnace envelopes (Part 3).

| Boundaries | Boundary conditions |
|------------|---------------------|
| 1. Internal surfaces of arch, vertical walls and floor | Temperature change mode: \(T_{st} = \begin{cases} T_{in} + C_s \tau & \text{at } \tau \leq t_{heating} \\ T_{max} - \Delta T & \text{at } t_{heating} \leq \tau \leq t_{cooling} \\ T_{max} - \Delta T - C_c \tau & \text{at } t_{cooling} \leq \tau \leq t_{cooling} \end{cases}\) \(C_s=\text{heating rate, } \text{K}^{-1}; C_c=\text{cooling rate, } \text{K}^{-1}; t_{heating}=\text{time duration of the heating period, } \text{s}; t_{cooling}=\text{time duration of the period with constant maximal temperature } T_{max} \text{ in the furnace, } \text{s}; t_{cooling}=\text{time duration of cooling period, } \text{s}; T_{in}=\text{initial temperature, } \text{K.}\) If \(C_s=\text{const.}: C_s = \frac{T_{max} - \Delta T - T_0}{t_{cooling}}\) Temperature differences: \(\Delta T = \begin{cases} 0\text{K at arch} \\ 50\text{K at vertical walls} \\ 100\text{K at floor} \end{cases}\) |
| 2. Burner walls | \(T = T_{max}\) |
| 3. External surfaces | \(-\lambda \frac{\partial T_{st}}{\partial n} + h_{sc}(T - T_0) = 0\) |
Numerical simulation of the conjugate heat transfer (CHT) in chamber furnaces

**Variant 1**

Geometrical model, including Part 1, 2 and 3.

CFD analysis at boundary conditions in Table 1:
1) Heat transfer at the external surfaces
2) Velocities, temperatures and mass fractions on the inlets
3) Gage pressure on the outlets

Transient fields of pressure, velocities, temperatures, concentrations and other variables.

**Variant 2**

Step 1. Separate geometrical models of Parts 1 + Part 2 and Parts 3.

Step 2. Simulation of transient heat transfer in Part 3 at boundary conditions in Table 2.

Step 3. Determination of artificial heat transfer coefficients $U^*$, reflecting the accumulated heat and thermal losses.

Step 4. CFD analysis of Parts 1+2 at boundary conditions in Table 3:
1) Heat transfer at the internal surfaces using $U^*$
2) Velocities, temperatures and mass fractions on the inlets
3) Gage pressure on the outlets

---

**Figure 1.** Algorithm for numerical simulation of the transient heat transfer in gas chamber furnaces. Table 3. Boundary conditions for CFD analysis of the furnace space (Part 1 and 2).

| Boundaries | Boundary conditions |
|------------|---------------------|
| 1. Internal surfaces of vertical walls, floor and arch | $-\lambda \frac{\partial T}{\partial n} + U^*(T - T_{\text{inlet}}) = 0$ |
| where $U^*$ is different for arch, floor and walls |
| 2. Inlets | Air temperature: $T_{\text{air}} = T_{\text{air}}(\tau)$ |
| 2.1 – Air inlets | Velocity, normal to the boundary: $V_{n,\text{air}} = V_{n,\text{air}}(\tau)^*$ |
2.2 – Fuel inlets

Oxygen and nitrogen mass fractions: \( g_{O_2} = 0.22; g_{N_2} = 0.78 \)

Velocity, normal to the boundary: \( V_{n,fuel} = V_{n,fuel}(\tau) \)

Mass fractions of the fuel components \( i \): \( g_i \)

3. Exhaust gases outlet

\( p = 0 \) Pa according a reference pressure

* The moment values of the air and fuel flows are determined by in situ measurements at working aggregates or can be computed by a thermal balance of the furnace and mass balance of the combustion process.

3. Numerical investigations

Variant 2 of the algorithm above is used to analyse and improve the performance of a high temperature chamber furnace for firing of technical ceramics. The heating periods includes increasing of gas temperature to approximately 1600°C for 15 hours and keeping that temperature for an hour. These processes are realized by subsequent rising of the fuel (natural gas) and air flows. The fuel and the combustion air are conducted in the furnace by six burners (Figure 3). The ceramic articles (chucks with relatively small sizes) are arranged on a refractory construction (checker work). During the operations of the furnace non-uniform temperature and velocity fields in the furnace space are established, resulting in unsatisfactory firing of the production and subsequent wastes. Detail information about the furnace and the production is not possible due to confidential rules.

Initially a geometrical model of the ¼ of furnace envelopes, divided by symmetry planes, was generated. The arch, floor and walls are included in the geometry, the burners are neglected. Transient temperature fields are simulated numerically at the overall firing regime to determine the final temperature field after the cooling period that is the initial one for the next heating process (Figure 2). The instantaneous values of artificial overall heat transfer coefficients \( U^* \) are calculated for the established heating period.

The geometrical model of the internal furnace space (Parts 1 and 2) is obvious from Figure 3. The articles to be fired are not present in the geometrical models: they are modelled by increasing of the density and the roughness of the horizontal checker work elements. The geometry of the burners is generated according their technical data.

A numerical simulation of the transient combustion, fluid flow and heat transfer processes were implemented at boundary conditions, corresponding to Table 3. The variations of fuel and air flows and temperatures with the time at the inlets of the burners are determined at in situ measurements. They are used to compute the inlet velocities.

The models are calibrated and validated comparing the controlled (measured) temperatures in the furnace space to the computed ones.

The conducted studies proved the inappropriate topology and capacity of the burners and the inability to maintain a uniform temperature field in the furnace chamber by adjusting the combustion process (Figure 3). The power of the burners is higher than the necessary one, computed by a thermal balance. That results in difficulty at the flow control and non-fully combustion in the top row burners (Figure 5). Their positions on the relatively small furnace volume lead to non-uniform thermal loads to the furnace envelopes and the ceramic ware.

A variant for reconstruction of the furnace, changing the burner installation and auxiliary refractory checker work is accepted for further investigations (Figure 4). It is accepted to change the number of the burners, their positions and their power in order to ensure possibilities for smooth flow control, uniform forced convection in the furnace space and full combustion of the fuel.
Figure 2. Initial temperature field in the furnace envelope (at the start of the heating process).

Figure 3. Gas streamlines and temperature fields at the moment of the firing process at the existing construction.
Numerical simulations of the transient combustion process, temperature, velocity and pressure fields are implemented in accordance to the conception for reconstruction. An increase of time for circulation of hot gases in chamber space at the new burner topology in comparison to the existing one is established. It is due to the extended flow path provoked by the counter current action of the lower and upper burners. As results, complete combustion of the fuel is observed (Figures 5 and 6).

A reduction of the fuel consumption per firing cycle of 48% is expected after the reconstruction of the furnace aggregate as a result of wastes decreasing, possibility for smooth flow control and a full combustion.
4. Conclusions
The proposed algorithm for numerical investigation of the transient conjugate heat transfer allows obtaining of detail information of the thermal and fluid flow processes in periodically working gas chamber furnaces. The rapid development of the computer equipment’s is a precondition for its complex use and improvement. The algorithm can be applied for analysis of the efficiency of furnaces and connected with them burner installations at operation conditions and project stages.

References
[1] Valtchev G, Rashev a V and Georgieva M 1998 Possibilities for reduction of fuel consumption in bread ovens Proceedings of University of Food Technology. XLIII (3) p 65-69
[2] Yordanov K, Mechkarova T, Stoyanova A and Zlateva P 2018 Determination of the temperature of cathode unit of indirect plasma burner through a computer simulation model Proceedings of the Second International Scientific Conference IITI 2017 - Advances in Intelligent Systems and Computing vol 680 (Springer Cham) p 403-409
[3] Kaushal P, Mohammadazhar A and Arjunsinh M 2018 Development of mathematical model for heat transfer analysis of cylindrical induction furnace wall International Journal of Advance Research and Innovative Ideas in Education. 4(6) p 451-461
[4] Chougule, V N, Daud O F, Gupta G R, Shelke V H and Ugale R V 2013 Numerical analysis of combustion furnace performance by CFD In Second National Conference on Recent Developments in Mechanical Engineering p 26-27
[5] Zashkova L 2008 Mathematical modelling of the heat behaviour in the ceramic chamber furnaces at different temperature baking curves Simulation Modelling Practice and Theory. 16 p 1640–1658
[6] Launder B and Spalding D 1974 The numerical computation of turbulent flows Computer Methods in Applied Mechanics and Engineering 3 p 269-289
[7] Spalding D 1958 Heat transfer from surfaces of non-uniform temperature Journal of Fluid Mechanics. 4 p 22-32
[8] ANSYS Release 16 2016 © SAS IP Inc

Acknowledgments
The study has been financially supported by the Operational Programme "Science and education for smart growth" 2014-2020 of the European Union cofounded by the European Social Fund through the project BG05M2OP001-2.009-0015 “Support for the development of capacity of doctoral students and young researchers in the field of engineering, natural and mathematical sciences”. The authors acknowledge the research ideas of National Program Young scientist and postdoctoral students, funded by Bulgarian Ministry of Education.