ADVANCES IN HEATING EQUIPMENT: SAVING ENERGY BY NUMERICAL AND ANALYTICAL HEAT TRANSFER ENHANCEMENT TECHNIQUES

Alina Adriana Minea

Technical University "Gheorghe Asachi"
from Iasi, Bd. D. Mangeron no. 63
Iasi, 700050, Romania
E-mail: aminea@tuiasi.ro

Received 07 July 2016
Accepted 09 December 2016

ABSTRACT

In some industrial heating processes, fuel represents only a very small fraction of the total cost of manufacturing. But in most industrial heating processes, fuel is a considerable expense. Since the last decade of the twentieth century, embargos, wars, regulations, and deregulations have caused the costs of oil and gas to go through unsettling fluctuations. Costs of electric energy also have also risen, because of the increasing cost of fuels, wages, and equipment. The difference between fuel saving and fuel wasting often determines the difference between profit and loss; thus, heat saving is a must. The words "economy" and "efficiency," when used in their true sense in connection with industrial furnaces refer to the heating cost per unit mass of finished, sellable product.

Analytical, experimental and computational approaches represent three distinct methods to solve a given problem in heat transfer, fluid flow or energy efficiency. However, in actual practice, combinations of these three methods are used to obtain an approach that is best suited to a given problem.

Keywords: analytical, numerical, heating equipment, heat transfer, energy efficiency.

INTRODUCTION

The engineering cognizance of the need to increase the thermal performance of heating equipments, thereby effecting energy, material, and cost savings, as well as a consequential mitigation of environmental degradation, had led to the development and use of many heat transfer enhancement techniques. These methods have in the past been referred to with various terms, two of which are augmentation and intensification.

There is an enormous database of technical literature on the subject, now estimated at over 8000 technical papers and reports, which has been disseminated periodically in numerous bibliographic reports [1 - 19]. This literature documents the extensive research efforts devoted to establishing the conditions under which enhancement techniques will improve the heat or mass transfer in various applications. The efforts, initiated more than 140 years ago, when the first attempt to enhance heat transfer coefficients in condensing steam was reported in the classical study by J. P. Joule (1861), continued to be a major research and development activity.

Enhancement techniques essentially reduce the thermal resistance in conventional heating equipment by promoting higher convective heat transfer coefficient with or without surface area increases (as represented by fins or extended surfaces). On the other hand, heat exchange systems in spacecraft, electronic devices, and medical applications, for example, may rely primarily on enhanced thermal performance for their successful operation. The commercialization of enhancement techniques, where the technology has been transferred from laboratory to full-scale industrial use of those that are more effective and workable, has also led to a larger number of patents [2].

Sixteen different enhancement techniques have been identified by Bergles et al. [6 - 10]. They can be classified broadly as passive and active techniques. The use of two or more techniques (passive and/or active) in conjunction constitutes a compound enhancement.

The effectiveness of any of these methods is strongly dependent on the mode of heat transfer (single-phase free or forced convection, pool boiling, forced convection boiling or condensation, and convective mass transfer), and type and process application of the heat based equipment (heating equipments, heat exchangers etc).

Furthermore, as mentioned earlier, any two or more of these techniques (passive and/or active) may be employed simultaneously to obtain a compound enhancement in heat transfer that is greater than that produced by only one technique itself. Some promising applications, for example, are in heat or mass exchangers where one technique may preexist; this is particularly so when the existing enhancement is from an active method.

Methodology

Analytical, experimental and computational techniques are the three main approaches that may be involved for solving a problem in heat transfer [11 - 19].

Analysis

The equations, obtained from the conservation principles to describe and predict practical thermal transport processes, are generally too complicated to be solved analytically and computational methods are needed to obtain the desired solution. Analytical solutions are usually obtained only for few simplified and idealized circumstances. Examples of these are transient, lumped and steady-state one-dimensional processes that lead to ordinary differential equations with constant properties that result in linear partial differential equations, and simple convective processes with a known flow field. Various analytical techniques are available for solving linear differential equations and small systems of linear equations. However, the solution procedures are often quite involved and frequently lead to analytical expressions that may themselves require computation to obtain useful results [12]. Examples of these are the integral transform methods and the method of separation of variables, that are used for solving a variety of conduction problems. They lead to complicated integrals, series solutions and transcendental algebraic equations, which are then solved numerically. In addition, analytical techniques are usually not very versatile, requiring different techniques for different boundary conditions, geometries and material property variations. Analytical solutions for radiation and convective transport can generally be obtained for extremely simplified circumstances and very few practical cases can be considered by this approach.

Despite the limited applicability, complexity of the method and cumbersome analytical approach, the importance of analytical solutions can hardly be exaggerated. Analytical solutions provide the means to validate the numerical model and establish the accuracy of the results. This is done by considering a relatively simple problem that is amenable to an analytical solution and by comparing the results with those obtained by the numerical procedure applied to the same problem [12].

The main difference between the analytical and numerical approaches is that analytical methods obtain a solution that is valid everywhere in the region and at all times, within the constraints of the mathematical model, whereas numerical methods obtain results only at a finite number of discrete points and at finite time intervals. This makes it particularly attractive to use analytical methods for regions that are difficult to discretize and for short times for which numerical solutions may not be valid [13]. Therefore, it is desirable to obtain analytical solutions whenever possible and couple these with the numerical solutions, if necessary, to cover the entire computational region.

Experimentation

Experiments are generally time consuming and expensive. Therefore, experimental results are usually obtained for fairly narrow ranges of operating conditions and for a selected number of configurations, dimensions and designs. However, experimental results are extremely valuable in validating the mathematical and numerical models for a given thermal process or system.

Although analytical results for simpler circumstances and physical characteristics of the numerical results help in checking the accuracy and validity of the numerical scheme, a comparison with the experimental results from an actual system such as a prototype, are necessary to establish quantitatively the level of accuracy and confidence in the predictability of the numerical model.

Combined approaches

Analytical, experimental and computational approaches represent three distinct methods to solve a given problem in heat transfer, fluid flow or energy efficiency. However, in actual practice, combinations of these three methods are used to obtain an approach that is best suited to a given problem [14]. As mentioned earlier, boundary conditions may be used on analysis and experimental results, and different regions may be modeled separately by different approaches.

Numerical results are obtained if analytical methods cannot be used. Similarly, experimental inputs are built into many specialized commercial software packages, used to simulate certain types of systems or processes. Clearly, inputs from analysis and experiments, as well as comparisons of numerical results with those from these approaches, are generally desirable and necessary in order to obtain valid, realistic and accurate predictions from the numerical model for a given transport process or system. Benchmark solutions are established in several areas, using well defined problems and numerical results from a variety of methods. Comparison with analytical and experimental results may also be used in the developing of this solution. An example of such benchmark solutions is the computed laminar natural convection flow in a rectangular enclosure with isothermal vertical sides at a given temperature [14].

RESULTS AND DISCUSSION

Batch ovens and low-temperature batch furnaces (200 - 760°C) are in a range where convection capability may exceed radiation capability. Convection is used for effective heating in this temperature range where radiation is weak or has a “shadow problem”, because it travels only in straight lines. Increasing the convection heat transfer rate is generally accomplished by using circulating fans, by using high-velocity burners, by judicious load placement and spacing, and by enhanced heating. In this work, a new method for enhancing convection, by placing inclined panels in the enclosure, is proposed. The convection enhancement leads to diminishing the heating time and thus - to lower energy consumption for heating [11].

A mathematical model for heat transfer enhancement in the studied heating equipment.

The studied heating equipments are batch furnaces of low volume, working at medium temperatures. The studied heating cycle is up to 500°C. In this temperature range, convection capability is exceeding radiation. The aim of this study is to save energy by reducing heating time and this goal can be achieved by augmenting heat transfer in the heated enclosure. All experiments and the CFD analysis presented further are sustaining the idea of introducing two inclined radiant panels in the furnace chamber. The physical explanation is that the presence of the panels is of benefit for the total heat transfer coefficient.

The two involved mechanisms for heating in the considered enclosures are radiation and free convection. Radiation is simply enhanced by enlarging the heat transfer surface, thus the heat, transferred by radiation, is increasing. Convection heat transfer is strongly influenced by the panels’ position, which determines the air rate between the walls and the panels.

A useful approach of this method is the dimensionless analysis, explained before. Further on, it will keep in mind equations, derived by Andreozzi [17] with the help of dimensionless analysis and the associated parametric study. These equations were determined for chimney-type channels in internal natural convection flows for rectangular enclosures. This analysis can be successfully applied to heating equipments with panels inside. Each panel is forming a chimney-type channel between the heating region and the panel itself, creating two chimney-type channels in the enclosure. These channels are directing the flow and are modifying the natural convection.

For our specific case of a furnace we can consider the Andreozzi [17] dimensions factors as:

$\left(\frac{B}{b}\right)=1$, $\left(\frac{L}{L_h}\right)=1$ and $\left(\frac{L_h}{B}\right)=1$ and the equations are:

$$\Delta\Psi = 3,590 Ra^{0.2323} \quad (1)$$

$$Nu = \left\{ \begin{array}{l} \left[0.570\left(Ra\right)^{0.306}\right]^{-11} \\ +\left[1.155\left(Ra\right)^{0.161}\right]^{-11} \end{array} \right\}^{-\frac{1}{11}} \quad (2)$$

Combining these equations in terms of Ra number, one can obtain:

$$Nu = \left\{ \begin{array}{l} \left[0.1058\left(\Delta\Psi\right)^{1.3173}\right]^{-11} \\ +\left[0.4763\left(\Delta\Psi\right)^{0.6931}\right]^{-11} \end{array} \right\}^{-\frac{1}{11}} \quad (3)$$

and if the Nu number correlation is applied, one can get for panels of height L:

$$h = \frac{k}{L} \left\{ \begin{array}{l} \left[0.1058\left(\Delta\Psi\right)^{1.3173}\right]^{-11} \\ +\left[0.4763\left(\Delta\Psi\right)^{0.6931}\right]^{-11} \end{array} \right\}^{-\frac{1}{11}} \quad (4)$$

or for inclined panels at an angle α

$$h = \frac{k}{L\cos\alpha} \left\{ \begin{array}{l} \left[0.1058\left(\Delta\Psi\right)^{1.3173}\right]^{-11} \\ +\left[0.4763\left(\Delta\Psi\right)^{0.6931}\right]^{-11} \end{array} \right\}^{-\frac{1}{11}} \quad (5)$$

As a conclusion, equation (5) can be used to model the heat transfer coefficient based on panel length and inclination, thermal conductivity of the heating fluid and the associated stream function. This equation was considered as a base for energy saving possibilities in an electrical furnace with natural convection. Later on, the experimental and numerical tests will consider increasing air velocity as a base for enhancement of natural convection that can lead to augmentation of the overall heat transfer in the enclosure and decreasing of the heating time. Thus, this technique will go on minimizing the energy consumption for heating "thin parts" in the furnace.

Modeling possibilities for saving energy in heating equipments

Heat may be transferred from a furnace to the surface of a load by convection and/or radiation. Inside the load, heat is transferred by conduction. Quite different from this basic process of (a) external heat generation, (b) heat transfer to the load and subsequently (c) internal heat conduction, is the mechanism of inductive, conductive or microwave heating whereas the heat is generated directly inside the metal load due to Joule heat and molecular effects.

External Heat Transfer. The heat flux transferred from a furnace to a load is described by [2]:

$$q = q_{conv} + q_{rad-solid} + q_{rad-gas} \ [W/m^2] \quad (6)$$

where as the three components involve three very different principles of heat transfer mechanisms:

Convection due to the relative velocity w of the furnace atmosphere (fluid) to the load surface

Solid body (electromagnetic) radiation emitted and reflected by the furnace walls to the load, following Stefan Boltzmann's law. The geometrical relations of surfaces and the emission coefficients of load and furnace surfaces have to be considered as well.

Gas (electromagnetic) radiation of the radiating molecules in the furnace atmosphere to the load, considering composition and thickness as well as the temperature of the gas body at some higher power.

The heat which caused the temperature change of the load of a heat capacity c_p [J/kgK] is [2]:

$$Q = c_p \cdot dm \cdot \left(\frac{\partial T}{\partial \tau} \cdot d\tau\right) \text{ or}$$

$$c_p \cdot \rho \cdot dV \cdot \left(\frac{\partial T}{\partial \tau} \cdot d\tau\right) = = c_p \cdot \rho \cdot dx \cdot dy \cdot dz \cdot \frac{\partial T}{\partial \tau} \cdot d\tau \quad (7)$$

Thus, the differential equation of the temperature field T (x,τ) is derived as:

$$\frac{\partial T(x,\tau)}{\partial \tau} = -a \cdot \frac{\partial^2 T(x,\tau)}{\partial x^2} \quad (8)$$

where a is the thermal diffusivity.

However, the problem in practical engineering calculations is that all the material properties, especially c_p and k, show a more or less significant temperature dependency for most metals. This can be evaluated by using mean material property values over a temperature range to calculate indicative results.

The mathematical solution of equation (8) for real problems in industrial heating processes is not very practicable. This is even truer as furnace processes often cover wide temperature ranges. On the other hand, the real temperature profiles and their development over time are of interest to understand the heating process in detail. For this purpose numerical solutions and the finite difference method may be used. Doing this, equation (8) is transformed into a difference equation by using discrete values for the temperature T at the time n, n+1 and so forth, with a time step Δt. The location is indexed with j-1, j, j+1 and so forth, in a grid with a mesh distance Δx [2].

Heating of a "thin" load. "Thin" in a thermodynamically sense is a load, which does not develop significant temperature differences over the cross section, while being exposed to external heat transfer. So, temperature differences within the load may be neglected, the load is at the uniform temperature only. A „lumped analysis", treating the load as a "black box" at a uniform temperature is possible, when heating geometrically thin metal plates, foils or loads with high heat conductivity. The Biot number, Bi is used to characterize the thermodynamical thickness of a load, Bi describes the resistance of internal heat transfer to the resistance of external heat transfer [2].

$$\dot{q} = \frac{k}{s} \cdot \left(T_0 - T_{x=s}\right) \sim h \cdot \left(T_f - T_o\right) \quad (9)$$

$$\frac{\left(T_0 - T_{x=s}\right)}{\left(T_f - T_o\right)} \sim \frac{h}{k} \cdot s = \frac{\left(\frac{s}{k}\right)}{\left(\frac{1}{h}\right)} \quad (10)$$

$$Bi = \frac{h}{k} \cdot s \quad (11)$$

A body is considered to be "thin", when Bi < 0.5 and "thick", when Bi > 0.5. This is somewhat arbitrary and tells nothing more, than that the temperature "tension" in the load is 10 % of the temperature "tension" between the furnace and the load surface. Depending on the accuracy required, this limit between thin and thick in literature was also found to be 0.2 [14, 16]. In the later case, the temperature differences within the cross section have to be considered, for temperature and energy calculations.

Considering a heating process in a furnace with a temperature Tf over a wide temperature range, the load starting at a temperature of Tw1 and ending at Tw2 , a medium load temperature Tm, for which a medium heat transfer coefficient can be determined, can be assumed. The heating time t may be calculated as follows:

$$h \cdot A \cdot \left(T_f - T_w\right) \cdot \partial \tau = V \cdot \rho \cdot c_p \cdot \partial T_w \quad (12)$$

$$\frac{h \cdot A}{\rho \cdot c_p \cdot V} \cdot \partial \tau = \frac{\partial T_w}{\left(T_f - T_w\right)} \quad (13)$$

where as

$$\frac{A}{V} = \frac{A}{A \cdot s} = \frac{1}{s} \quad \text{for the plate}$$

$$\frac{A}{V} = \frac{2r\pi \cdot L}{r^2 \pi \cdot L} = \frac{2}{r} \quad \text{for the cylinder} \quad (14)$$

$$\frac{A}{V} = \frac{4R^2\pi}{\left(\frac{4R^3\pi}{3}\right)} = \frac{3}{R} \quad \text{for the sphere}$$

The parameter s is for a single sided heated plate; the thickness of the plate, respectively - for the half of the thickness for a double sided heated plate:

$$h \cdot \left(T_f - T_w\right) \cdot \partial \tau = \rho \cdot s \cdot c_p \cdot \partial T_w \Rightarrow$$
$$\Rightarrow \frac{h}{c_p \cdot (\rho \cdot s)} \partial \tau = \frac{1}{\left(T_f - T_w\right)} \cdot \partial T_w \quad (15)$$

After integration and applying boundary conditions:

$\tau = \tau_1 = 0$: $\quad C = \ln\left(T_f - T_{w1}\right)$

$\tau = \tau_2$: $\quad C = \ln\left(T_f - T_{w2}\right) + \frac{h \cdot \tau}{c_p \cdot \rho \cdot s}$

the heating time τ can be estimated with:

$$\tau = -\frac{c_p \cdot \rho \cdot s}{h} \cdot \ln\frac{\left(T_f - T_{w2}\right)}{\left(T_f - T_{w1}\right)} \qquad (16)$$

Moreover, saving energy possibilities for heating thin loads may rely on combining equation (4) with (16):

$$\tau = -\frac{c_p \cdot \rho \cdot s}{\frac{k}{L}\left\{\left[0.1058\left(\Delta\Psi\right)^{1.3173}\right]^{-11} + \left[0.4763\left(\Delta\Psi\right)^{0.6931}\right]^{-11}\right\}^{-\frac{1}{11}}} \cdot \ln\frac{\left(T_f - T_{w2}\right)}{\left(T_f - T_{w1}\right)} \qquad (17)$$

where c_p, ρ, s are the specific heat, density and dimension of the load; L is the panel length (or the wall length for furnaces without panels); ΔΨ is the stream function (similar to Re number, depicting the air rate) inside the heated chamber and k is the conductivity of the air at operating temperature (Tf).

Or, if it considers the furnace with inclined panels at an angle, α, one can get:

$$\tau = -\frac{c_p \cdot \rho \cdot s}{\frac{k}{L\cos\alpha}\left\{\left[0.1058\left(\Delta\Psi\right)^{1.3173}\right]^{-11} + \left[0.4763\left(\Delta\Psi\right)^{0.6931}\right]^{-11}\right\}^{-\frac{1}{11}}} \cdot \ln\frac{\left(T_f - T_{w2}\right)}{\left(T_f - T_{w1}\right)} \qquad (18)$$

If further a furnace with known maximum power supply, P, working for a specific time, τ is considered, the consumed energy can be written as:

$$E = P\tau = P\frac{c_p \cdot \rho \cdot s}{\frac{k}{L}\left\{\left[0.1058\left(\Delta\Psi\right)^{1.3173}\right]^{-11} + \left[0.4763\left(\Delta\Psi\right)^{0.6931}\right]^{-11}\right\}^{-\frac{1}{11}}} \cdot \ln\frac{\left(T_f - T_{w2}\right)}{\left(T_f - T_{w1}\right)} \qquad (19)$$

and for inclined panels:

$$E = P\frac{c_p \cdot \rho \cdot s}{\frac{k}{L\cos\alpha}\left\{\left[0.1058\left(\Delta\Psi\right)^{1.3173}\right]^{-11} + \left[0.4763\left(\Delta\Psi\right)^{0.6931}\right]^{-11}\right\}^{-\frac{1}{11}}} \cdot \ln\frac{\left(T_f - T_{w2}\right)}{\left(T_f - T_{w1}\right)} \qquad (20)$$

From equations (19) and (20), one can notice the influence of the heating time and stream function, respectively, on the energy consumption of a certain furnace working at maximum heating power. This equation is the base for optimization of furnace heating for "thin loads", where no special prescriptions are needed for heating. The saving energy technique applied and described in this publication, considered different types of panels (thick, thin, standard and perforated), under different inclinations, α. The chimney type effect on air rate in natural convection was studied and final energy consumption was monitored through total heating time, as expressed in equation (20).

As a conclusion, equation (20) can be considered as a mathematical model that describes the energy consumption of a furnace with inclined panels. This equation is very important, since it describes the possibilities of saving energy by decreasing the heating time for "thin loads", and increasing the air rate inside the furnace chamber, thus by intensifying convection and convection coefficient, respectively. Parameters considered as variables for the experimental and numerical study are the panels' position, surface and thickness that can lead to increasing the air velocity by creating the chimney effect with a result in augmenting natural convection. This equation is based on the dimensionless approach for natural convection heat transfer in chimneys type enclosures.

CFD possibilities for saving energy in heating equipments

A very popular approach for modeling heat transfer is the fixed grid enthalpy in which the fluid and the solid zones are not treated separately, but a common grid is used in the entire region. For solving the heat transfer problem, simulation of radiation and convection in a closed domain was considered. An important first step for establishing the model accuracy was to identify the boundary conditions and the properties of materials involved in the process [18]. Moreover, preliminary tests were carried out to verify the accuracy of the numerical solution. One global variable, Nu, was monitored to analyze the numerical solution independence of the mesh spacing, as is outlined below.

Governing equations and boundary conditions

The 2-D Navier–Stokes and energy equations have been used to describe the flow and heat transfer in the heated enclosure. The following assumptions had been adopted: (1) the heating fluid was Newtonian and incompressible, assumed as an ideal gas, (2) the flow was laminar, (3) the enclosure was heated by a constant heat flux from the resistors area, (4) variable thermophysical properties were considered for the fluid and (5) constant thermophysical properties were considered for the solids. Similar to any fluid mechanic and heat transfer problem, a numerical study for a fluid in a particular geometry is achieved by solving the main conservation laws for the flow. In general, a computational fluid dynamics (CFD) code follows three main steps when solving a problem [19]: integration of the conservation equations over the generated control volumes; changing of the obtained integral equations into algebraic equations with the aid of discretization methods; and finally – application of numerical iterative methods to solve the algebraic equations. Incompressible flow, Newtonian behavior, Boussinesq approximation for buoyancy force in natural convection and steady-state condition, seem to be reasonable simplifications when doing a numerical process. Also, the air flow in the enclosure can be considered 2D since in z direction there are no significant processes. Moreover, inserting the panels, air flow is modified mainly in y direction (along the panel and between the heated wall and the panel). In the 2D Cartesian coordinates and transient flow, the governing equations are as follows:

- Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{21}$$

- Momentum equation:

$$\rho\left(\frac{\partial u}{\partial \tau}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{22}$$

$$\rho\left(\frac{\partial v}{\partial \tau}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \tag{23}$$

- Energy equation:

$$\rho C_p\left(\frac{\partial T}{\partial \tau}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}\right)=k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right) \tag{24}$$

Referring to basic equations for the considered case, for air at maximum temperature of 500°C, the ratio and the Rayleigh number are:

$$\frac{Gr}{\mathrm{Re}^2}=4.13>1 \text{ and } Ra=Gr\,\mathrm{Pr}=0.38\cdot 10^6 \tag{25}$$

The ratio exceeds unity and we have to expect a strong buoyancy contribution to the flow. Consequently, Ra does not exceed 108, so it indicates buoyancy-induced laminar flow. Thus, the under relaxation factors of the solved equations were decreased in order to obtain more accurate results.

In the furnace chamber, different inclinations of the panels, varying from 0° to 21°, were considered. The constant heating flux boundary condition was adopted to heat up the furnace at maximum real electric power. The heating fluid was considered to be air. The isolation materials' properties were adopted from the furnaces instructions manual and considered as Cerafiber, produced by Thermal Ceramics.

Thermophysical properties

In order to obtain credible numerical results when modeling, one must focus on available correlations and choose the most proper one to apply for each of the various properties of a material. This becomes essential to know when solving the governing conservation equations. These properties might include thermal conductivity, viscosity, density and heat capacitance, depending in what form the governing equations are written. The effective physical properties of the materials studied can be evaluated using some well known classical formulas.

In these simulations, for all solid materials (isolation, resistors etc) the thermophysical properties were considered constant at a medium temperature value. The fluid, i.e. air properties were considered variable with temperature, as follows: the density was considered similar with that of the ideal gas; the heat capacitance, cp, was governed by a piecewise polynomial law with 8 coefficients, the thermal conductivity was defined by eight points, depending on temperature increasing; viscosity was varied by the Sutherland model, with three coefficients. All other properties (molecular mass, absorption coefficient, etc) were kept constant at a medium value. These assumptions helped us for simulating the real conditions, and are used largely in the literature [3-5, 13, 14, 17, 18].

Geometry

The computational domain is designated by a classic and an oval 2D furnace. The interior is 190x220 mm. The panels were inserted in the heating area. The different position of the panels was obtained using the journal files from Gambit, by modifying the angle of the second coordinated system. The grid was made axis symmetric, in order to ease the geometry creation and to easily modify it. This assumption decreases the computational effort and is realistic in terms of heating symmetry that is applicable for the studied furnaces.

Grid validation was our first preoccupation. Subsequently, several grids were tested in terms of the Nu number, until the results were grid-independent. Moreover, a finer grid was considered near the walls, in order to save computational time and to reveal the most important boundary processes.The heating was considered constant from the two lateral walls. The temperature was measured supplementary in a point in the middle of the furnace, at 3 cm from the ground. A denser grid with finer elements was picked for the oval furnace, since the processes and the air flow are different from the standard furnace version.

The grid has 3 parts: resistors, walls and interior. In Fig. 1 is the classic furnace with no panels, which is the standard version. In Fig. 2 is presented the grid obtained by introducing the panels, following the real experimental test and in Fig. 3 is presented the grid, obtained by introducing the panels in the oval enclosure.

The grid for both furnaces has 3 parts: resistors, walls and interior. For example, for the oval furnace, the resistors have 2586 nodes and 3881 elements; the walls have 8936 nodes and 15878 elements and at the interior there are 13308 nodes and 13049 elements.

Considering the fact that the grid is an unstructured one, with relatively high differences between the start and the maximum values of the applied size function, we have chosen a pressure-based 2D laminar viscous model with implicit formulation and unsteady solver, for the computation process. For radiation the Discrete Ordinates model was preferred. The following solver settings were applied: second order and body force weighting interpolation schemes for density and pressure fields, respectively and SIMPLEC algorithm for the pressure-velocity coupling method. The algebraic "discretized

Table 1. Variable x for the grid.

Simulation case number	x, mm
1	10
2	15
3	17
4	19
5	21

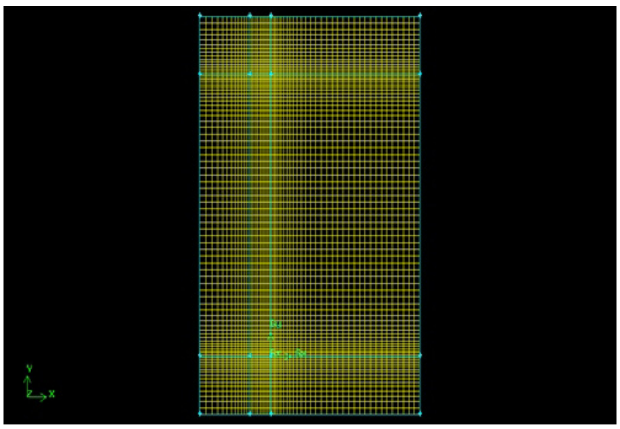

Fig. 1. The grid and boundary conditions for the standard furnace.

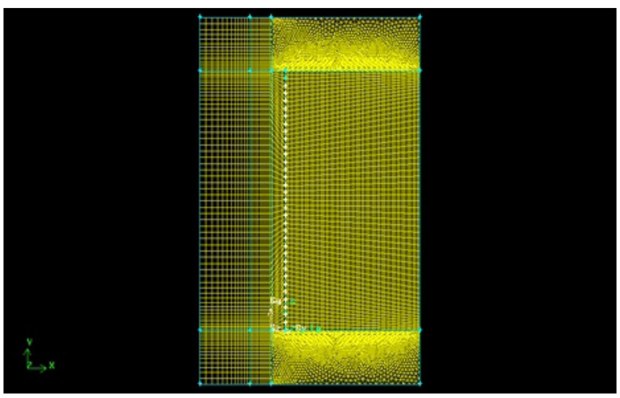

Fig. 2. Grid detail and boundary conditions for the standard furnace with panels.

Table 2. Report on heat transfer [1].

Simulation case number	Radiation Heat Transfer Rate, W		Total Heat Transfer Rate, W	
	wall-shadow	wall.2-shadow	wall-shadow	wall.2-shadow
1	2302.37	12.25	2390.09	2556.48
2	2343.94	14.08	2431.38	2531.53
3	2365.90	14.39	2453.04	2512.19
4	2389.96	15.65	2476.78	1487.57
5	2415.57	17.07	2502.34	1459.14

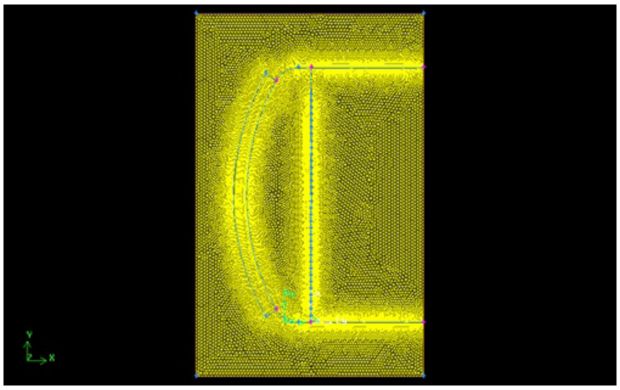

Fig. 3. Grid detail and boundary conditions for the oval furnace with panels at 0 degrees.

equations" resulting from this spatial integration process were sequentially solved throughout the physical domain considered. FLUENT solves the systems obtained from discretization schemes, using a numerical method. The residuals resulting from the integration of the governing equations were considered as convergence indicators. Fluent solves the linear systems resulting from discretization schemes, using a point implicit (Gauss–Seidel) linear equation solver in conjunction with an algebraic multigrid method. During the iterative process, the residuals were carefully monitored. For all simulations performed in the present study, converged solutions were considered when the residuals resulting from an iterative process for all governing equations were lower than 10^{-6}.

The solution domain is discretized with a mesh with non-uniform spacing in the horizontal and vertical direction, which allows the boundary layers to be resolved without an excessive number of grids. Moreover, all cases were carefully selected and modified using some new features of Fluent, by importing the boundary conditions and geometry modification from the journal files and geometry re-creation by reading journals. These features were used in order to avoid errors and to be able to establish a solid base of comparison.

Simulation on an optimum curved space

The numerical investigations were started from the geometry of the oval manufactured furnace, presented before. The aim was to establish a more conveniently curved heating space. Simulations were carried out in the same conditions for every grid obtained, and for the same number of iterations and an iteration step. In this work, all simulations, involving convection in a closed domain and a DO model for radiation heat transfer were completed. Table 1 shows the varieties of configurations used, depending on the value of x from Fig. 4. This variable, x, is shaping the chamber geometry.

However, as the value of x increases, the difference in the heating temperature turns out to be more significant.

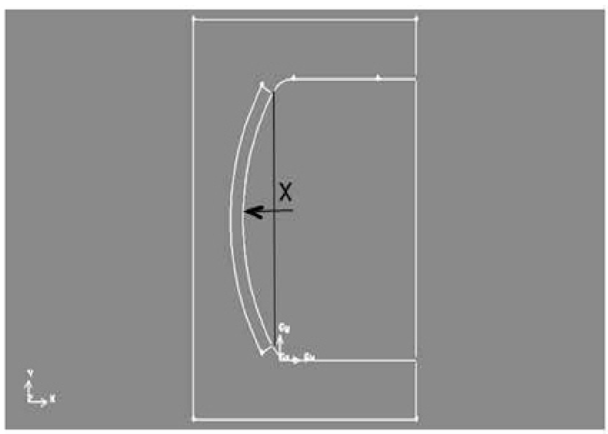

Fig. 4. Configuration of the studied geometry [1].

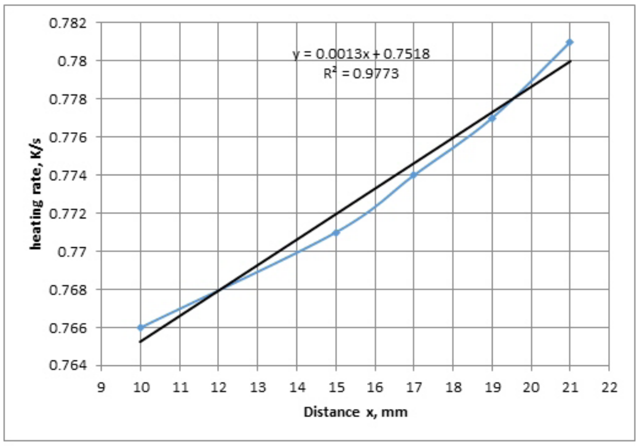

Fig. 5. Simulation final results interpretation [1].

The final heating temperature was estimated by monitoring the iteration time. As an example, at x = 10 mm, Fig. 5 shows the values of final heating temperature T at the default interior region, i.e. the central point of the oval geometry, which corresponds to the charge position in a furnace. From the time - temperature history, heating rates at different locations could be easily calculated. The final simulation results are presented in Fig. 4, only for the heating rate.

Fig. 5 shows a trend line for the simulation data, described by:

$$y = 0,001\ x + 0,751 \qquad (26)$$

So, it can be said that the air rate inside the oval furnace increases along with the curvature of the interior geometry.These results were obtained on the bases of intensifying heat transfer along the interior walls. From the simulation one can get information regarding the heat transfer in all cases. These results are given in Table 2.

CONCLUSIONS

Considering the presented case of an electric furnace, which at 500°C works mostly on natural convection, the numerical approach is the only theoretical approach that is available; otherwise the mathematical system of equations cannot be handled in simplifications that do not alter the physical processes. Possibility for alteration of the physical processes exists in any simulation, so the author believes that an experimental validation is mandatory, when available. Yet, any experiment can be amended by errors generated by the instrumentation, thermocouples' precision or position, initial or final conditions. So, when possible, numerical analysis is welcome. In this situation, the author has used the similarity variable method and the streamline function that describes very well the fluid rate in the enclosure. Also, for the specific case of panels' insertion, the chimney effect was adopted.

Equations (17) and (19) were chosen to describe the heating and the energy consumption for the considered equipment. A comparison between different heating regimes was accomplished and variation of heating time was observed when introducing the panels and this was in agreement with equation (18). The explanation is that by introducing the panels, the air rate in the boundary layer, v, increases and a decreasing of heating time, τ is noticed. Considering equation (20), the energy consumption by heating at maximum power is decreasing, while the heating time is decreasing and air rate is increasing.

As a final conclusion, a possibility for saving energy in the studied furnaces was identified. It is realized as an enhanced convection heat transfer, by changing the air rate profile, intensifying the air circulation in the enclosure and creating a chimney effect, through the introduction of metallic panels.

REFERENCES

1. A. A.Minea, Simulation of Heat Transfer Processes in an Unconventional Furnace, J. of Eng. Thermophysics, 19, 4, 2010, 31-38.
2. A.A.Minea, Advances in industrial heat transfer, CRC press Taylor & Francis, 2012
3. M.Mulas,S.Chibbaro,G.Delussu,I.Di Piazza,M. Talice, Efficient parallel computations of flows of arbitrary fluids for all regimes of Reynolds, Mach and Grashof numbers , Int. J. of Numerical Methods for Heat & Fluid Flow, 12, 6, 2002, 637-657.
4. A. Andreozzi, O. Manca, V. Naso, Natural convection in vertical channels with an auxiliary plate, Int. J. of Numerical Methods for Heat & Fluid Flow, 12, 6, 2002, 716-734.
5. O.Manca, M. Musto, V. Naso, Experimental Investigation of Natural Convection in an Asymmetrically Heated Vertical Channel with an Asymmetric Chim-

ney, Heat Transfer, 2005, 3-12.

6. A.E. Bergles, New Frontiers in Enhanced Heat Transfer, ASME, New York, 2000.
7. A.E. Bergles, V. Nirmalan, G. Junkhan, R.L Webb, Bibliography of Augmentation of Convective Heat and Mass Transfer, II, Report HTL-31, ISU-ERI-Ames-84221, Heat Transfer Laboratory, Iowa State University, Ames, IA, 1983.
8. A.E. Bergles, M.K. Jensen, E.F.C. Somerscales, R.M. Manglik, Literature Review of Heat TransferEnhancement Technology forHeat Exchanges in Gas-Fired Applications, Chicago, 1991.
9. A.E. Bergles, M.K. Jensen, B. Shome, Bibliography on Enhancement of Convective Heat and Mass Transfer, Troy, NY, 1995.
10. A.E. Bergles, M.K. Jensen, B. Shome, The Literature on Enhancement of Convective Heat and Mass Transfer, J. Enhanced Heat Transfer, 4, 1, 1996, 1–6.
11. W. Trinks, M.H. Mawhinney, R.A. Shannon, R.J. Reed, J.R. Garvey, Industrial Furnaces, Sixth Edition, John Wiley & Sons, 2004.
12. I. Pop, D.B. Ingham, Convective Heat Transfer: Mathematical and Computational Modelling of Viscous Fluids and Porous Media, Elsevier, USA, 2001.
13. A. Bejan, A. Krauss, Heat transfer handbook, Willey and sons, USA 2003.
14. W.S. Janna, Engineering Heat Transfer – second edition, CRC Press, 2001.
15. P.K. Sahoo, M.I.A. Ansari., A.K. Datta, A computer based iterative solution for accurate estimation of heat transfer coefficients in a helical tube heat exchanger, J. Food Eng., 58, 2003, 211-214.
16. B.V. Karlekar, R.M. Desmond, Heat Transfer, West Publishing Co., St. Paul, l982.
17. A. Andreozzi, B. Buonomo, O. Manca, Thermal management of a symmetrically heated channel–chimney system, Int. J. of Thermal Sciences, 48, 2009, 475-487.
18. Y. Jaluria, K.E. Torrance, Computational heat transfer, 2nd ed., Taylor and Francis, New York, 2003.
19. Y. Jaluria, Numerical Study of the Thermal Processes in a Furnace, Numer. Heat Transfer, 7, 1984, 211-224.